## MULTIPHYSICS MODELING USING CONSOL

A FIRST PRINCIPLES APPROACH

ROGER W. PRYOR



## MULTIPHYSICS MODELING USING COMSOL®

## A FIRST PRINCIPLES APPROACH

## ROGER W. PRYOR, PHD





JONES AND BARTLETT PUBLISHERS Sudbury, Massachusetts BOSTON TORONTO LONDON SINGAPORE World Headquarters Jones and Bartlett Publishers 40 Tall Pine Drive Sudbury, MA 01776 978-443-5000 info@jbpub.com www.jbpub.com

Jones and Bartlett Publishers Canada 6339 Ormindale Way Mississauga, Ontario L5V 1J2 Canada Jones and Bartlett Publishers International Barb House, Barb Mews London W6 7PA United Kingdom

Jones and Bartlett's books and products are available through most bookstores and online booksellers. To contact Jones and Bartlett Publishers directly, call 800-832-0034, fax 978-443-8000, or visit our website, www.jbpub.com.

Substantial discounts on bulk quantities of Jones and Bartlett's publications are available to corporations, professional associations, and other qualified organizations. For details and specific discount information, contact the special sales department at Jones and Bartlett via the above contact information or send an email to specialsales@jbpub.com.

#### Copyright © 2011 by Jones and Bartlett Publishers, LLC

All rights reserved. No part of the material protected by this copyright may be reproduced or utilized in any form, electronic or mechanical, including photocopying, recording, or by any information storage and retrieval system, without written permission from the copyright owner.

COMSOL and COMSOL Multiphysics are trademarks of COMSOL AB. COMSOL materials are reprinted with the permission of COMSOL AB.

#### **Production Credits**

Publisher: David Pallai Editorial Assistant: Molly Whitman Production Director: Amy Rose Associate Production Editor: Tiffany Sliter Associate Marketing Manager: Lindsay Ruggiero V.P., Manufacturing and Inventory Control: Therese Connell Composition: Glyph International Cover Design: Scott Moden Cover Image: © Photo courtesy of COMSOL, Inc. (www.comsol.com) Printing and Binding: Malloy, Inc. Cover Printing: Malloy, Inc.

#### Library of Congress Cataloging-in-Publication Data

Pryor, Roger W.
Multiphysics modeling using COMSOL : a first principles approach / Roger W. Pryor.
p. cm.
Includes index.
ISBN-13: 978-0-7637-7999-3 (casebound)
ISBN-10: 0-7637-7999-7 (casebound)
1. Physics—Data processing. 2. Physics—Mathematical models. 3. Physics—Computer simulation. 4.
Engineering—Data processing. 5. Engineering—Mathematical models. 6. Engineering—Computer simulation. 7. COMSOL Multiphysics. I. Title.
QC52.P793 2010
530.15'18—dc22

2009033347

#### 6048

Printed in the United States of America 13 12 11 10 09 10 9 8 7 6 5 4 3 2 1

### Contents

Preface ix Introduction x

#### Chapter 1 Modeling Methodology 1

Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers 1 Hardware Considerations 1 Coordinate Systems 2 Implicit Assumptions 5 1D Window Panes Heat Flow Models 6 1D Single-Pane Heat Flow Model 6 1D Dual-Pane Heat Flow Model 16 1D Triple-Pane Heat Flow Model 27 First Principles Applied to Model Definition 35 Common Sources of Modeling Errors 37 Exercises 38

#### Chapter 2 Materials and Databases 39

Materials and Database Guidelines and Considerations 39
COMSOL<sup>®</sup> Material Library Module: Searchable Materials Library 40
MatWeb: Searchable Materials Properties Website 43
PKS-MPD: Searchable Materials Properties Database 48
References 61
Exercises 62

#### Chapter 3 1D Modeling 63

1D Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers 63
1D Modeling Considerations 63
Coordinate System 64

1D KdV Equation: Solitons and Optical Fibers 64 COMSOL KdV Equation Model 65 First Variation on the KdV Equation Model 77 Second Variation on the KdV Equation Model 83 1D KdV Equation Models: Summary and Conclusions 90 1D Telegraph Equation - 90 COMSOL 1D Telegraph Equation Model 92 First Variation on the Telegraph Equation Model 98 Second Variation on the Telegraph Equation Model 105 1D Telegraph Equation Models: Summary and Conclusions 110 References 112 Exercises 112

#### Chapter 4 2D Modeling 113

2D Guidelines for New COMSOL <sup>®</sup> Multiphysics <sup>®</sup> Modelers 1	13
2D Modeling Considerations 113	
Coordinate System 114	
2D Electrochemical Polishing (Electropolishing) Theory 115	
COMSOL 2D Electrochemical Polishing Model 118	
First Variation on the 2D Electrochemical Polishing Model	130
Second Variation on the 2D Electrochemical	
Polishing Model 147	
2D Electrochemical Polishing Models:	
Summary and Conclusions 165	
2D Hall Effect Model Considerations 167	
2D Hall Effect Model 171	
First Variation on the 2D Hall Effect Model 186	
Second Variation on the 2D Hall Effect Model 203	
2D Hall Effect Models: Summary and Conclusions 221	
References 222	
Exercises 223	

#### Chapter 5 2D Axisymmetric Modeling 225

2D Axisymmetric Guidelines for New COMSOL® Multiphysics® Modelers 225 2D Axisymmetric Modeling Considerations 225 2D Axisymmetric Coordinate System 227 Heat Conduction Theory 228

2D Axisymmetric Heat Conduction Modeling 229 2D Axisymmetric Cylinder Conduction Model 229 First Variation on the 2D Axisymmetric Cylinder Conduction Model 240 Second Variation on the 2D Axisymmetric Cylinder Conduction Model, Including a Vacuum Cavity 250 2D Axisymmetric Cylinder Conduction Models: Summary and Conclusions 263 2D Axisymmetric Insulated Container Design 265 2D Axisymmetric Thermos\_Container Model 265 First Variation on the 2D Axisymmetric Thermos\_Container 285 Model Second Variation on the 2D Axisymmetric Thermos\_Container Model 301 2D Axisymmetric Thermos\_Container Models: Summary and Conclusions 316 References 318 Exercises 319

#### Chapter 6 2D Simple Mixed-Mode Modeling 321

2D Mixed-Mode Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers 321 2D Mixed-Mode Modeling Considerations 321 2D Coordinate System 322 2D Axisymmetric Coordinate System 324 Joule Heating and Heat Conduction Theory 324 Heat Conduction Theory 325 2D Resistive Heating Modeling 326 2D Resistive Heating Model 326 First Variation on the 2D Resistive Heating Model 345 Second Variation on the 2D Resistive Heating Model, Including 366 Alumina Isolation 2D Resistive Heating Models: Summary and Conclusions 388 2D Inductive Heating Considerations - 389 2D Axisymmetric Coordinate System 389 2D Axisymmetric Inductive Heating Model 393 First Variation on the 2D Axisymmetric Inductive Heating Model 411 Second Variation on the 2D Axisymmetric Inductive Heating Model 433

2D Axisymmetric Inductive Heating Models: Summary and Conclusions 453 References 457 Exercises 457

#### Chapter 7 2D Complex Mixed-Mode Modeling 459

2D Complex Mixed-Mode Guidelines for New COMSOL <sup>®</sup>
Multiphysics <sup>®</sup> Modelers 459
2D Complex Mixed-Mode Modeling Considerations 459
2D Coordinate System 461
Electrical Impedance Theory 461
2D Electric Impedance Sensor Model: Basic 464
Basic 2D Electric Impedance Sensor Model: Summary
and Conclusions 475
2D Electric Impedance Sensor Model: Advanced 477
2D Electric Impedance Sensor Models: Summary
and Conclusions 492
Generator and Power Distribution Basics 493
2D AC Generators: Static and Transient 498
2D AC Generator Model (2D_ACG_1): Static 498
2D AC Generator Model (2D_ACG_2): Transient 522
2D AC Generators, Static and Transient Models: Summary
and Conclusions 529
2D AC Generator: Sector—Static and Transient 530
2D AC Generator Sector Model (2D_ACGS_1): Static 531
2D AC Generators, Static Sector Model: Summary and
Conclusions 555
2D AC Generator Sector Model (2D_ACGS_2): Transient 557
2D AC Generators, Static and Transient Sector Models:
Summary and Conclusions 568
References 569
Exercises 570

#### Chapter 8 3D Modeling 571

3D Modeling Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup>
Modelers 571
3D Modeling Considerations 571
3D Coordinate System 572

Electrical Resistance Theory 573 Thin Layer Resistance Modeling Basics 575 3D Thin Layer Resistance Model: Thin Layer Approximation 577 3D Thin Layer Resistance Model, Thin Layer Approximation: Summary and Conclusions 591 3D Thin Layer Resistance Model: Thin Layer Subdomain 593 3D Thin Layer Resistance Models: Summary and Conclusions 608 **Electrostatic Modeling Basics** 611 3D Electrostatic Potential Between Two Cylinders 613 3D Electrostatic Potential Between Two Cylinders: Summary and Conclusions 622 3D Electrostatic Potential Between Five Cylinders 622 3D Electrostatic Potential Between Five Cylinders: Summary and Conclusions 633 Magnetostatic Modeling Basics 634 3D Magnetic Field of a Helmholtz Coil 636 3D Magnetic Field of a Helmholtz Coil: Summary and Conclusions 649 3D Magnetic Field of a Helmholtz Coil with a Magnetic Test Object 652 3D Magnetic Field of a Helmholtz Coil with a Magnetic Test Object: Summary and Conclusions 666 References 669 Exercises 670

#### Chapter 9 Perfectly Matched Layer Models 671

Perfectly Matched Layer Modeling Guidelines and Coordinate Considerations 671
PML Theory 671
PML Models 676
2D Dielectric Lens Model, with PMLs 676
2D Dielectric Lens Model, with PMLs: Summary and Conclusions 690
2D Dielectric Lens Model, without PMLs 691
2D Dielectric Lens Model, with and without PMLs: Summary and Conclusions 702
2D Concave Mirror Model, with PMLs 707 2D Concave Mirror Model, with PMLs: Summary and Conclusions 723
2D Concave Mirror Model, without PMLs 724
2D Concave Mirror Model, with and without PMLs: Summary and Conclusions 736
References 744
Exercises 745

#### Chapter 10 Bioheat Models 747

Bioheat Modeling Guidelines and Coordinate Considerations 747
Bioheat Equation Theory 747
Tumor Laser Irradiation Theory 749
2D Axisymmetric Tumor Laser Irradiation Model 749
2D Axisymmetric Tumor Laser Irradiation Model: Summary and Conclusions 768
Microwave Cancer Therapy Theory 769
2D Axisymmetric Microwave Cancer Therapy Model 770
2D Axisymmetric Microwave Cancer Therapy Model: Summary and Conclusions 794
References 794
Exercises 795

Index 797



viii

### Preface

The purpose of this book is to introduce hands-on model building and solving with COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software to scientists, engineers, and others interested in exploring the behavior of different physical device structures on a computer, before actually going to the workshop or laboratory and trying to build whatever it is.

The models presented in this text are built within the context of the physical world (applied physics) and are explored in light of first principles analysis techniques. As with any other method of problem solution, the information contained in the solutions from these computer simulations is as good as the materials coefficients and the fundamental assumptions employed in building the models.

The primary advantage in combining computer simulation and first principles analysis is that the modeler can try as many different approaches to the solution of the same problem as needed to get it right (or at least close to right) in the workshop or laboratory the first time that device components are fabricated.

#### Acknowledgments

I would like to thank David Pallai of Jones and Bartlett Publishers for his ongoing encouragement in the completion of this book. I would also like to thank the many staff members of COMSOL, Inc., for their help and encouragement in completing this effort.

I would especially like to thank my wife, Beverly E. Pryor, for the many hours that she spent reading the manuscript and verifying the building instructions for each of the models. Any errors that remain are mine and mine alone.

Roger W. Pryor, Ph.D.

## Introduction

COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software is a powerful finite element (FEM), partial differential equation (PDE) solution engine. The basic COMSOL Multiphysics software has eight add-on modules that expand the capabilities of the basic software into the following application areas: AC/DC, Acoustics, Chemical Engineering, Earth Science, Heat Transfer, MEMS, RF, and Structural Mechanics. The COMSOL Multiphysics software also has other supporting software, such as the CAD Import Module and the Material Library.

In this book, scientists, engineers, and others interested in exploring the behavior of different physical device structures through computer modeling are introduced to the techniques of hands-on building and solving models through the direct application of the COMSOL Multiphysics software, the AC/DC Module, the Heat Transfer Module, and the RF Module. Chapter 9 explores the use of perfectly matched layers (PML) in the RF Module. The final technical chapter (Chapter 10) explores the use of the bioheat equation in the Heat Transfer Module.

The models presented here are built within the context of the physical world (applied physics) and are presented in light of first principles analysis techniques. As with any other methodology of problem solution, the information derived from the modeling solutions through use of these computer simulations is only as good as the materials coefficients and the fundamental assumptions employed in building the models.

The primary advantage derived from combining computer simulation and first principles analysis is that the modeler can try as many different approaches to the solution of the same problem as needed to get it right (or at least close to right) in the workshop or laboratory before the first device components are fabricated and tested. The modeler can also use the physical device test results to modify the model parameters and arrive at a final solution more rapidly than by simply using the cut-and-try methodology.

#### Chapter Topics

This book comprises ten technical chapters. Its primary focus is to demonstrate to the reader the hands-on technique of model building and solving. The COMSOL Concepts and Techniques are shown in Figure 1. The COMSOL modules employed in the various

Chapter										
Concept/Technique	1	2	3	4	5	6	7	8	9	10
1D PDE modeling	•		٠							
2D Axisymmetric coordinates						٠				
2D Axisymmetric modeling					•					
2D Modeling				•					•	٠
3D Modeling								٠		
Animation			•	•	٠	•	•		•	٠
Azimuthal inductive heating						٠				
Bioheat equation										٠
Boundary conditions			٠							
Boundary integration					٠					
CAD drawings (geometry objects), export and import					•	٠				
Conductive media Pc				•	•	٠	•	٠		
Constants, import and export					•					
Cross-section plot				٠		٠		٠		
Cylindrical coordinates					•					
Deformed mesh – moving mesh (ALE)				•			٠			
Domain plot parameter							٠			
Electromagnetics				٠						
Electrostatic potentials								•		
Electro-thermal coupling						٠				
Floating contacts				•						
Free mesh parameters				٠	٠	٠				
Geometric assembly (pair creation across a boundary)							•			
Global equations							•			
Heat transfer coefficients	•				٠					
Imbalance-offset geometry				•						
Induction heating						٠				
In-plane electric currents							•			
In-plane te waves									٠	
Iterative solver								٠		
Lagrange parameters										٠
Laplacian operator								٠		
Magnetostatic modeling								٠		
Materials library	•	•			•		•			
Maximum element size				٠	٠	٠				
Mesh mapping							•			
Mixed-materials modeling						٠				
Mixed-mode modeling						•				
Opaque and transparent thermally conductive materials					٠					
Ordinary differential equation (ODE)							•			
Parametric solutions			٠		٠					
Perfectly matched layers									•	
Periodic point conditions							•			
Perpendicular induction currents							•			
Polyline drawings							•			
Quadrilateral mesh (Quad)				٠	•	٠				
Quasi-static solutions					•					
Reference frame							•			
Rotating machinery							•			
Scalar expressions					٠	٠				
Scalar variables						٠				
Static solutions					٠					
Subdomain mesh				٠	٠	٠				
Suppress subdomain									٠	
Surface integrals					٠					
Thin layer approximation								٠		
Thin layer subdomain								٠		
Time-harmonic analysis							٠			
Transient analysis				٠		٠				
Triangular mesh				•	•	٠				
Weak constraints	-	· · · · ·		•						

					Cha	apter				
Module	1	2	3	4	5	6	7	8	9	10
Basic	•	٠	•	•	•	•	٠	٠	•	•
AC/DC				•		•	٠	٠		
Heat Transfer					•					•
Materials Library		٠								

**FIGURE 2** COMSOL Modules Employed

models in specific chapters are shown in Figure 2, and the physics concepts and techniques employed in the various models in specific chapters are shown in Figure 3.

These grids link the overall presentation of this book to the underlying modeling, mathematical, and physical concepts. In this book, in contrast to some other books with which the reader may be familiar, key ancillary information, in most cases, is contained in the notes.

NOTE Please be sure to read, carefully consider, and apply, as needed, each note.

#### Chapter 1. Modeling Methodology

Chapter 1 begins the introduction to the modeling process by discussing the fundamental considerations involved: the hardware (computer platform), the coordinate systems (physics), the implicit assumptions (lower dimensionality considerations), and first principles analysis (physics). Three relatively simple 1D models are presented, built, and solved for comparison: one-pane, two-pane, and three-pane thermal insulation window structures. Comments are also included on common sources of modeling errors.

#### Chapter 2. Materials and Databases

Chapter 2 briefly introduces three sources of materials properties data: the COMSOL Material Library, MatWeb, and the PKS-MPD.

The COMSOL Material Library is a module that can be added to the basic COM-SOL Multiphysics software package to expand the basic library that is already included. It contains data on approximately 2500 materials, including elements, minerals, soil, metal alloys, oxides, steels, thermal insulators, semiconductors, and optical materials. Each material can have up to 27 defined properties. Each of those defined properties is available as a function of temperature.

MatWeb is an online searchable subscription materials properties data source. MatWeb has three classes of access: Unregistered (free limited feature access), Registered Member (free expanded feature access), and Premium Member (fee-based

					Cha	pter				
Physics Concepts	1	2	3	4	5	6	7	8	9	10
AC induction						•				
Anisotropic conductivity				•						
Antennas										٠
Bioheat equation										٠
Boltzmann thermodynamics					٠					
Complex AC theory							٠			
Complex impedance							٠			
Concave mirror									٠	
Coulomb gauge								٠		
Dielectric lenses									٠	
Distributed resistance								٠		
Electrical impedance theory							٠			
Electrochemical polishing				•						
Electrostatic potentials in different geometric configurations								٠		
Faraday's law				•						
First Estimate Review	•							•		
Foucault (eddy) currents						•				
Fourier's law						•				
Free-space permittivity								•		
Good first approximation	•					•				
Hall effect				•						
Hard and soft nonlinear magnetic materials							•			
Heat conduction theory	•				٠					
Helmholtz coil								•		
Information transmission			٠							
Insulated containers					٠					
Joule heating					•	•				
Lorentz force				•						
Magnetic field				•						
Magnetic permeability								•		
Magnetic vector potential								•		·
Magnetostatics								•		
Maxwell's equations	-				•			•	•	
Mechanical to electrical energy conversion					-		•	-	-	
Microwave irradiation							-			•
Newton's law of coding					•					-
Ohm's law		-			-	•	•	•		
Optical (laser) irradiation						-	-	-		•
Pennes's equation										•
Perfectly matched layers: 2D planar, 3D cartesian cylindirical and										-
Spherical									٠	İ
Perfusion										•
Planck's constant					•					<u> </u>
Power transmission grids, AC and DC	-	-		-	-		•			<u> </u>
Power transmission, AC and DC	-				-		•			<u> </u>
Reactance	-	-				-	•			<u> </u>
Semiconductor dual carrier types	-	<u> </u>		•			-			<u> </u>
Skin depth	-	-		L.			•			<u> </u>
Soliton waves	+		•			<u> </u>	-			<u> </u>
Telegraphs equation	+		•							<u> </u>
Thin layer resistance	+		-					•		<u> </u>
I HILL I AVEL I UNISIAILUU	1	1		1						<u> </u>
Vacuum					•					

access to all features, plus selected data storage and modeling software formatted data export). MatWeb has 69,000 data sheets for materials, including plastics, metals, ceramics, semiconductors, fibers, and various other commercially available materials.

PKS-MPD (Pryor Knowledge Systems—Materials Properties Database) is a new searchable materials properties database with data on more than 4000 materials, including elements, minerals, soil, metal alloys, oxides, steels, thermal insulators, semiconductors, optical materials, and biomaterials (tissue). Each material can have up to 43 defined properties. Each of those defined properties is associated with the temperature of measurement and the frequency of measurement. The collection of defined properties for each materials property datum is exportable in a format suitable for use with the COMSOL Multiphysics software.

#### Chapter 3. 1D Modeling

The first half of Chapter 3 models the 1D KdV equation and two variations. The KdV equation is a powerful tool that is used to model soliton wave propagation in diverse media (e.g., physical waves in liquids, electromagnetic waves in transparent media). It is easily and simply modeled with a 1D PDE mode model.

The second half of Chapter 3 models the 1D telegraph equation and two variations. The telegraph equation is a powerful tool that is used to model wave propagation in diverse transmission lines. It can be used to thoroughly characterize the propagation conditions of coaxial lines, twin pair lines, and microstrip lines, among other things. The telegraph equation is easily and simply modeled with a 1D PDE mode model.

#### Chapter 4. 2D Modeling

The first half of Chapter 4 models the 2D electrochemical polishing model. This model is a powerful tool that can be used to model surface smoothing for diverse projects (e.g., microscope samples, precision metal parts, medical equipment and tools, large and small metal drums, thin analytical samples, vacuum chambers).

The second half of Chapter 4 models the 2D Hall effect. The 2D Hall Effect model is a powerful tool that can be used to model Hall effect magnetic sensors for sensing fluid flow, rotating and linear motion, proximity, current, pressure, and orientation.

#### Chapter 5. 2D Axisymmetric Modeling

The first half of Chapter 5 models three 2D axisymmetric cylinder conduction models. From a comparison of the three models, it can be readily observed that the presence of a vacuum cavity significantly reduces the rate of heat flow through the model and raises the equilibrium temperature at the surface receiving the heat flux. The second half of Chapter 5 models three 2D axisymmetric thermos container models. From a comparison of the three models, it can be readily observed that the presence of a vacuum cavity significantly reduces the rate of heat flow through the model and the associated heat loss.

#### Chapter 6. 2D Simple Mixed-Mode Modeling

The first half of Chapter 6 models three 2D resistive heating models. These models are more illustrative of the mixed-mode modeling concept than they are directly amenable to the comparison of calculated values. They present different examples of the diversity of applied scientific and engineering model designs that can be explored using electro-thermal coupling and transient analysis. These models also demonstrate the significant power of relatively simple physical principles, such as Ohm's law and Joule's law.

The second half of Chapter 6 models three 2D axisymmetric inductive heating models. These models demonstrate the difference in level of complexity between single-coil and multi-coil models. In the Inductive\_Heating\_1 model, the concept of inductively produced heating is introduced. In the Inductive\_Heating\_2 model, the concept of inductively produced heating is applied to a practical application (a heated crucible) so as to present one example of the diverse applied scientific and engineering model designs that can be explored using electro-thermal coupling and transient analysis. In the Inductive\_Heating\_3 model, the crucible is filled with a commonly used metal for melting.

These models are examples of the good first approximation type of model. In other words, they demonstrate the significant power of relatively simple physical principles, such as Ohm's law and Joule's law, when applied in the COMSOL Multiphysics modeling environment. They could, of course, be modified by the addition of calculations, insulating materials, and heat loss through convection, among other changes.

#### Chapter 7. 2D Complex Mixed-Mode Modeling

The first third of Chapter 7 introduces two 2D electric impedance sensor models: basic and advanced. Those models employ high-frequency currents -1 MHz alternating currents AC—to explore the differential impedance within a body of material in a noninvasive fashion. Such currents are applied to the material of the modeled body to locate volumes that differ in impedance from the impedance of the bulk material by monitoring the local impedance.

The basic version models the location of a fixed-volume impedance difference. The advanced version models the location of a fluctuating difference volume, as might be seen in a medical application measuring lung function. 2D electric impedance tomography research is currently exploring the application of this type of impedance sensing measurement technology to the detection of breast cancer, lung function, brain function, and numerous other areas.

The second third of Chapter 7 introduces two 2D AC generator models: static and transient. These models generate low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. They demonstrate the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of rotating machines for the conversion of mechanical energy to electrical energy.

The last third of Chapter 7 introduces two 2D AC generator sector models: static and transient. These models generate low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. They demonstrate the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of a rotating machine for the conversion of mechanical energy to electrical energy. An ordinary differential equation (ODE) is incorporated into the sector model to handle the torque-related aspects of the model calculations.

#### Chapter 8. 3D Modeling

The first third of Chapter 8 models the 3D thin layer resistance model, thin layer approximation, and the thin layer resistance model, thin layer subdomain. The first model employs the thin layer approximation to solve a model by replacing the center domain with a contact-resistance identity pair. Such an approximation has broad applicability. It is important to note that the use of the thin layer approximation is applicable to any problem in which flow is described by the divergence of a gradient flux (e.g., diffusion, heat conduction, flow through porous media under Darcy's law).

The application of the thin layer approximation is especially valuable to the modeler when the differences in domain thickness are so great that the mesh generator fails to properly mesh the model or creates more elements than the modeling platform can handle ("run out of memory" problem). In those cases, this approximation may enable a model to be solved that would otherwise fail.

A direct comparison is made of the model solutions by comparing the results obtained from the cross-section plots. As seen from the examination of those plots, the only substantial difference between the two solutions is the electrical potential difference across subdomain 2 (the thin layer). Thus the modeler can choose the implementation that best suits his or her system and time constraint needs, without suffering excessive inaccuracies based on the approximation method.

The second third of Chapter 8 introduces the 3D electrostatic potential model. This modeling technique demonstrates one of the methods that can be used by the modeler to explore electrostatic potentials in different geometric configurations. It can be applied to both scientific and engineering applications (e.g., ranging from X-ray tubes and particle accelerators to paint sprayers and dust precipitators). The 3D\_ESP\_2 model is typical of those that might be found in a particle beam analyzer or a similar engineering or scientific device.

The last third of Chapter 8 models the 3D magnetic field of a Helmholtz coil. This model demonstrates the magnetic field uniformity of a Helmholtz coil pair. This magnetostatic modeling technique can be applied to a diverse collection of scientific and engineering applications (e.g., ranging from magnetometers and Hall effect sensors to biomagnetic and medical studies).

A related model, the 3D magnetic field of a Helmholtz coil with a magnetic test object, demonstrates the magnetic field concentration when a high relative permeability object lies within the field of the Helmholtz coil. This magnetostatic modeling technique can be applied to a diverse collection of scientific and engineering test, measurement, and design applications.

#### Chapter 9. Perfectly Matched Layer Models

The first half of Chapter 9 introduces the 2D dielectric lens models, with and without perfectly matched layers (PMLs). [The PML model best approximates a free space environment (no reflections).] Comparison is made between the two models. The differences in the electric field, *z*-component visualizations between the PML and no-PML models amount to approximately 2%. Depending on the nature of the problem, such differences may or may not be significant. What these differences show the modeler is that he or she needs to understand the application environment well so as to build the best model. For other than free space environments, the modeler needs to determine the best boundary condition approximation using standard practices and a first principles approach. Always do a first principles analysis of the environment before building the model.

The second half of Chapter 9 introduces the 2D concave mirror models, with and without PMLs. There are only small differences in the electric field, *z*-component visualizations between the PML and no-PML models for the concave mirror. This lack of large differences between the PML and no-PML models again shows the modeler that he or she needs to understand the relative importance of the modeled values to evaluate the application and the application environment so as to build the best model.

#### Chapter 10. Bioheat Models

The bioheat equation plays an important role in the development and analysis of new therapeutic medical techniques (e.g., killing of tumors). If the postulated method raises the local temperature of the tumor cells without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

The results (estimated time values) from the model calculations will significantly reduce the effort needed to determine an accurate experimental value. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C (315.15 K) to 60 °C (333.15 K).

The first half of Chapter 10 models the bioheat equation as applied with a photonic heat source (laser). The second half of the chapter models the bioheat equation as applied with a microwave heat source.

NOTE Executable copies of each model and related animations are available in **full** color on the accompanying DVD.



# 1

## **Modeling Methodology**

#### In This Chapter

Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers Hardware Considerations Coordinate Systems Implicit Assumptions
1D Window Panes Heat Flow Models 1D Single-Pane Heat Flow Model 1D Dual-Pane Heat Flow Model 1D Triple-Pane Heat Flow Model
First Principles Applied to Model Definition Common Sources of Modeling Errors

#### Guidelines for New COMSOL® Multiphysics® Modelers

#### **Hardware Considerations**

There are two basic rules to selecting hardware that will support successful modeling. First, new modelers should be sure to determine the minimum system requirements that their version of COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software needs before borrowing or buying a computer to run their new modeling software. Second, these new modelers should run their copy of COMSOL Multiphysics software on the best platform with the highest processor speed and the most memory obtainable: The bigger and faster, the better. It is the general rule that the speed of model processing increases directly as a function of the processor speed, the number of platform cores, and the available memory.

**NOTE** The number of platform cores is equal to the number of coprocessors designed into the computer (e.g., one, two, four, eight, . . .).

The platform that this author uses is an Apple<sup>®</sup> Mac Pro<sup>®</sup>, running Mac OS X<sup>®</sup> version 10.5.x, and also running Parallels Desktop<sup>®</sup> 3.x with Microsoft<sup>®</sup> Windows XP<sup>®</sup>.

This Mac Pro has four 3 GHz cores and 16 GB of RAM. This platform, as configured, is more powerful, more versatile, more stable, and more cost-effective than other potential choices. It can handle complex 3D models in short computational times (more speed, more memory)—that is, in minutes instead of the hours that may be required by less powerful systems. This configuration can run any of the 32-bit COMSOL Multiphysics software, when using COMSOL Multiphysics Version 3.4. The Apple hardware is configured for 64-bit processing and will run at the 64-bit rate when using COMSOL Multiphysics Version 3.5. If new modelers desire a different 64-bit operating system than Macintosh OS X, then they will need to choose either a Sun<sup>®</sup> or a Linux<sup>®</sup> platform, using UNIX<sup>©</sup> or a PC with a 64-bit Microsoft Windows operating system.

The "3 GHz" specification is the operating speed of each of the cores and the "16 GB" is the total shared random access memory (RAM). The "64-bit" refers to the width of a processor instruction.

Once the best available processor is obtained, within the constraints of your budget, install your copy of COMSOL Multiphysics software, following the installer instructions. Once installed, COMSOL Multiphysics software presents the modeler with a graphical user interface (GUI). For computer users not familiar with the GUI concept, information in such an interface is presented primarily in the form of pictures with supplemental text, not exclusively text.

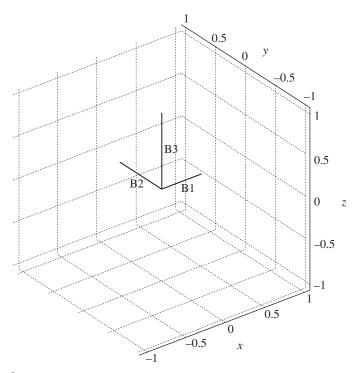
#### **Coordinate Systems**

Figure 1.1 shows the default (x-y-z) coordinate orientation for COMSOL modeling calculations. This coordinate system is based on the right-hand rule.

The right-hand rule is summarized by its name. Look at your right hand, point the thumb up; point your first finger away from your body, at a right angle (90 degrees) to your thumb; and point your second finger at a right angle to the thumb and first finger, parallel to your body. Your thumb represents the *z*-axis, your first finger represents the *x*-axis, and your second finger represents the *y*-axis.

In this right-handed coordinate system, x rotates into y and generates z. If you have a need to convert your model from the x-y-z frame to a Spherical Coordinate frame, then the transformation can be implemented using built-in COMSOL mathematical functions. The x-y-z to spherical coordinate conversion is achieved through the following equations:

Spherical radius (r): 
$$r = \operatorname{sqrt}(x^2 + y^2 + z^2)$$
 (1.1)



**FIGURE 1.1** 3D GUI example of the Cartesian coordinate system (*x*-*y*-*z*)

x-y plane rotational angle $\phi$ (phi):	$phi = a \tan^2(y, x)$	(1	.2	)
--	------------------------	----	----	---

*x-z* plane rotational angle  $\theta$  (theta): theta =  $a \cos(z/r)$  (1.3)

The built-in function sqrt(argument) indicates that COMSOL Multiphysics will take the positive square root of the argument contained between the parentheses. The built-in function *a* tan2(argument) indicates that COMSOL Multiphysics will convert the argument contained between the parentheses to an angle in radians (in this case, phi). The built-in function *a* cos(argument) indicates that COMSOL Multiphysics will convert the argument contained between the parentheses to an angle in radians (in this case, phi). The built-in function *a* cos(argument) indicates that COMSOL Multiphysics will convert the argument contained between the parentheses to an angle in radians (in this case, theta).

To employ these spherical conversion equations in your model, you will need to start COMSOL Multiphysics, select "3D" in the Model Navigator Screen, select the desired Application Mode, and click the OK button. Using the pull-down menu, select Options > Expressions > Scalar Expressions and then enter equations 1.1, 1.2, and 1.3 in the Scalar Expressions window, as shown in Figure 1.2.

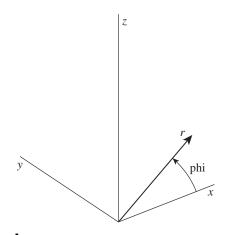
Name	Expression	Unit	Description
r	sqrt(x^2+y^2+z^2)	m	spherical radius
phi	atan2(y,x)	rad	rotation counterclockwise in X-Y plane
theta	acos(z/r)	rad	rotation clockwise in Z-X plane
( -		- 10	) • •

**FIGURE 1.2** COMSOL Multiphysics 3D Scalar Expressions window with the spherical coordinate transform equations entered

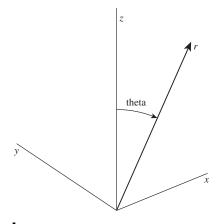
NOTE When a list of operations is presented sequentially (A > B > C > D), the modeler is expected to execute those operations in that sequence in COMSOL.

Once these equations are available in the model, the x-y-z coordinates can be converted as shown in Figures 1.3 and 1.4.

**NOTE** The rotational sense of an angular transform is determined by viewing the angular rotation as one would view a typical analog clock face. The vector  $\mathbf{r}$  rotating from x into y for a positive angle phi is counterclockwise, because phi equals zero at the positive x-axis. The vector  $\mathbf{r}$  rotating from z into x for a positive angle theta is clockwise, because theta equals zero at the positive z-axis.







**FIGURE 1.4** Spherical coordinate transform angle and rotational sense for  $(\theta)$  theta

As all potential COMSOL modelers know, the Cartesian (x-y-z) and Spherical Coordinate (r-phi-theta) reference frames are not the only coordinate systems that can be used as the basis frame for Multiphysics models. In fact, when you first open the Model Navigator, you are given the option and are required to choose one of the following modeling coordinate systems: 1D, 2D, 3D, Axial Symmetry (1D), or Axial Symmetry (2D). The coordinate system that you choose determines the geometry and specific subgroup of COMSOL Application Modes that can be applied in that selected geometry (geometries).

An Application Mode is the initial collection of equations, independent variable(s), dependent variable(s), default settings, boundary conditions, and other properties that are appropriate for the solution of problems in that branch of physics (e.g., acoustics, electromagnetics, heat transfer). As indicated by the name "Multiphysics," multiple branches of physics can be applied within each model. Diverse models that demonstrate the application of the multiphysics concept will be explored in detail as this book progresses.

#### **Implicit Assumptions**

A modeler can generate a first-cut problem solution as a reasonable estimate, by choosing initially to use a lower-dimensionality coordinate space than 3D (e.g., 1D, 2D Axisymmetric). By making a low-dimensionality geometric choice, a modeler can significantly reduce the total time needed to achieve a detailed final solution for the chosen prototype model. Both new modelers and experienced modelers alike must be especially careful to fully understand the underlying (implicit) assumptions, unspecified conditions, and default values that are incorporated into the model as a result of simply selecting the lower-dimensionality geometry.

A first-cut solution is the equivalent of a back-of-the-envelope or on-a-napkin solution. Solutions of this type are relatively easy to formulate, are quickly built, and provide a first estimate of whether the final solution of the full problem is deemed to be (should be) within reasonable bounds. Creating a first-cut solution will often allow the modeler to decide whether it is worth the time and money required to create a fully implemented higher-dimensionality (3D) model.

Space, as all modelers know, comes with four basic dimensions: three space dimensions (x-y-z) and one time dimension (t). For example, the four dimensions might be *x*-*y*-*z*-*t*, or *r*- $\phi$ - $\theta$ -*t*. Relativistic effects can typically be neglected, except in cases of high velocity or ultra-high accuracy. Neither of these types of problems will be covered in this book.

Note Relativistic effects typically become a concern only for bodies in motion with a velocity approaching that of the speed of light ( $\sim 3.0 \times 108$  m/s) or for ultra-high resolution time calculations at somewhat lower velocities.

The types of calculations presented within this book are typically for steady-state models or for relatively low-velocity transient model solutions. Any transient solution model can be solved using a quasi-static methodology.

In a steady-state model, the controlling parameters are defined as numerical constants and the model is allowed to converge at the equilibrium state defined by the specified constants. In the quasi-static methodology, a model solution to a problem is found by initially treating the model as a steady-state problem. Incrementally modifying the modeling constants then moves the model problem solution toward the desired transient solution.

#### 1D Window Panes Heat Flow Models

Consider, for example, a brief comparison between a relatively simple 1D heat flow model and the identical problem presented as a 3D model. The models considered here are those of a single-pane, dual-pane, or triple-pane window mounted in the wall of a building on a typical winter's day. The questions to be answered are Why use a dual-pane window?

#### 1D Single-Pane Heat Flow Model

Run the COMSOL Multiphysics application. Select "New" and then select "1D" in the Model Navigator. Then select "COMSOL Multiphysics." Select "Heat Transfer" followed by "Conduction" and then "Steady-state analysis." Click OK.

Name	Expression	Value	Description
T_in	70[degF]	294.261111[K]	Interior Temperature
T_out	0[degF]	255.372222[K]	Exterior Temperature
р	1[atm]	1.01325e5[Pa]	air pressure
		1	

**FIGURE 1.5** 1D Constants specification window

After the 1D workspace appears, enter the constant values needed for this model: Use the menu bar to select Options > Constants. Enter the following items: T\_in tab 70[degF] tab Interior Temperature tab T\_out tab 0[degF] tab Exterior Temperature tab p tab 1[atm] (in Version 3.4) or 1.01325e5[Pa] (for earlier versions) tab air pressure tab; see Figure 1.5. Click the Apply button. These entries in the Constants window define the Interior Temperature, the Exterior Temperature, and the air pressure for use in this model. Click on the disk icon in the lower-left corner of the Constants window (Export Variables to File) to save these constants as ModelOC\_1D\_WP1.txt for use in the comparison models to follow later in this chapter. Click the OK button.

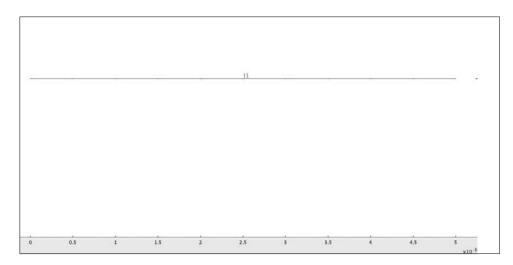
Next, draw a line to represent the thickness of a 1D window pane: Use the menu bar and select Draw > Specify Objects > Line. Enter the following: 0.000 space 0.005; see Figure 1.6. Leave the default Polyline, and click the OK button. Click the Zoom Extents icon in the toolbar.

Once the Zoom Extents icon is clicked, the specified 0.005 m line will appear in the workspace, as shown in Figure 1.7.

After the 0.005 m line has been created in the workspace, use the menu bar and select Physics > Subdomain Settings > 1. The window shown in Figure 1.8 contains the known properties of copper (Cu) as the default materials properties values.

Coord		ine
_	000 0.005	ОК
Style:	Polyline	¢ Cancel
Name:	11	Apply

FIGURE 1.6 1D Line specification window



**FIGURE 1.7** The 0.005 m line shown in the 1D workspace

NOTE COMSOL Multiphysics software is based on the Finite Element Method (FEM). To ensure that it is as easy as possible to use out of the box for modelers, both new and experienced, COMSOL inserts default materials properties and numerical parameters settings values to avoid singularities and other errors in the calculation of solutions. The modeler will need to verify that all materials and parameter values that are incorporated into their particular models are the appropriate values for the desired solution.

If copper is not the material of choice, as in this heat transfer model, then the materials property values need to be changed.

The implicit assumption here would be that the default values in the Subdomain Settings window are the correct values that the modeler needs to build the desired model. Specifically, that assumption would be true only if the modeler were building the heat transfer model using copper. In general, that implicit assumption is not correct. The modeler needs to know before building any new models what the approximate expected values are for the particular properties of the materials selected for use in the model. A number of sources have detailed materials properties values available: Some sources are available at no cost, while other sources have different levels of availability for different fees. Materials properties sources are discussed in Chapter 2.

For this model, click the Load button, and then select "Basic Materials Properties" and "Silica Glass." Click OK. All the appropriate values displayed in the Value/ Expression subwindows in the Subdomain Settings window are altered as the new values are loaded from the library. (Silica glass has a thermal conductivity value roughly 0.35% of that of copper.)

Once the thermal conductivity is loaded from the materials properties library, enter T\_in in the  $T_{ext}$  and  $T_{ambtrans}$  windows, as shown in Figure 1.9.

Click the Init button and enter T\_in as shown in Figure 1.10. Click OK. Setting the T\_in value as the initial temperature of the window pane (subdomain 1) allows for quicker convergence of the model and avoids any singularities.

			etaan	
Subdomains Groups		Physics Init	Element	Color/Style
Subdomain selection	-Thermal propert	ies and heat sources/sir	nks	
1	l ibrary material	Silica Glass	( load)	
	Quantity	Value/Expression	Unit	Description
	k	1.38[W/(m*K)]	W/(m·K)	Thermal conductivity
	Q	0	W/m <sup>3</sup>	Heat source
	h <sub>trans</sub>	0	W/(m <sup>3</sup> ·K)	Convective heat transfer coefficien
Group:	T <sub>ext</sub>	T_in	к	External temperature
Select by group	C <sub>trans</sub>	0	W/(m <sup>3</sup> ·K <sup>4</sup> )	User-defined constant
Active in this domain	Tambtrans	T_in	к	Ambient temperature

FIGURE 1.9 1D Subdomain Settings Physics window settings

0.0	Subdomain Settings - Heat Transfer by Conduction (ht)
Fquation $-V \cdot (kVI) = Q + h_{trans}(I_{ex})$	$t^{-1}$ ) + C <sub>trans</sub> (1 <sub>ambtrans</sub> <sup>4</sup> - 1 <sup>4</sup> ), 1 = temperature
Subdomains Grou	ps Physics Init Element Color/Style
Subdomain selection	T(t <sub>0</sub> ) T_in K Temperature
Group: Select by group Active in this domai	n
	Help Apply Cancel OK

**FIGURE 1.10** 1D Subdomain Settings Init window settings

Now the modeler needs to enter the appropriate Boundary Settings. Using the menu bar, select Physics > Boundary Settings. After the Boundary Settings window appears, as shown in Figure 1.11, select "1" in the Boundary Settings Boundary selection window.

Boundaries Groups		Coefficien	ts Color	]
Boundary selection	Boundary sources an	d constraints		
1	Boundary condition:	Heat flux	<b>*</b>	
2	Quantity q <sub>0</sub>	Value/Expression	Unit	Description
	h			Heat transfer coefficient
	T <sub>inf</sub>			External temperature
Group:	Const			Problem-dependent constant
Select by group	T <sub>amb</sub>			Ambient temperature
Interior boundaries	то			Temperature

**FIGURE 1.11** Boundary Settings window

$(k\nabla T) = q_0 + h(T_{inf} - T) + Co$	onst(1 <sub>amb</sub> - 1')			
Boundaries Groups		Coefficien	ts Color	
Boundary selection	Boundary sources an	d constraints		
2	Boundary condition:	Heat flux	\$	
	Quantity	Value/Expression	Unit	Description
	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
	h	15	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
	T <sub>inf</sub>	T_in	к	External temperature
Group:	Const	0	$W/(m^2 \cdot K^4)$	Problem-dependent constant
	T <sub>amb</sub>	T_in	к	Ambient temperature
Select by group	то	0	к	Temperature

FIGURE 1.12 Filled-in Boundary Settings window for boundary 1

Next, select "Heat flux" from the Boundary conditions pull-down menu. Enter 15 in the Heat transfer coefficient window (h). Enter T\_in in the External temperature window ( $T_{inf}$ ). Enter T\_in in the Ambient temperature window ( $T_{amb}$ ). Click the Apply button. Figure 1.12 shows the filled-in Boundary Settings window for boundary 1.

Now select "2" in the Boundary Settings Boundary selection window. Select "Heat flux" as the Boundary condition. Enter 15 in the Heat transfer coefficient window (h). Enter T\_out in the External temperature window ( $T_{inf}$ ). Enter T\_out in the Ambient temperature window ( $T_{amb}$ ). Click the Apply button. Figure 1.13 shows the filled-in Boundary Settings window for boundary 2. Click OK.

All the Subdomain Settings and the Boundary Settings have now been either chosen or entered. The next step is to mesh the model. In this simple model, all the modeler needs to do is use the toolbar and select "Initialize Mesh." Figure 1.14 shows the initial mesh. The line segments between the dots are the mesh elements.

To improve the resolution, the mesh will be refined twice. All the modeler needs to do is use the toolbar and select Refine Mesh > Refine Mesh. The refined mesh of the single-pane model now contains the 60 elements shown in Figure 1.15, rather than the original 15 elements shown in Figure 1.14.

For this model, the software will be allowed to automatically select the Solver and Solver Parameters. To solve this model, go to the menu bar and select Solve > Solve Problem. The solution is found almost immediately, in 0.014 second. The solution is plotted using the default Postprocessing values and is shown in Figure 1.16.

12

$(k\nabla T) = q_0 + h(T_{inf} - T) + C$				
Boundaries Groups		Coefficien	ts Color	
Boundary selection	Boundary sources	and constraints		
1	Boundary conditio	n: Heat flux	\$	
-	Quantity	Value/Expression	Unit	Description
	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
	h	15	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
	T <sub>inf</sub>	T_out	к	External temperature
Group:	Const	0	W/(m <sup>2</sup> ·K <sup>4</sup> )	Problem-dependent constant
	T <sub>amb</sub>	T_out	к	Ambient temperature
Select by group	τ <sub>ο</sub>	0	к	Temperature

**FIGURE 1.13** Filled-in Boundary Settings window for boundary 2

**NOTE** The precise length of time required for the solution of a given model depends directly on the configuration of the platform and the overhead imposed by the operating system.

The implicit assumption here would be that the default values in the Postprocessing window are the correct values that the modeler needs to plot the calculated results of the built model. Specifically, that assumption in general will not be true. For example, in the case of this model, the default plot is in Kelvins (K), when the modeler would probably prefer degrees Fahrenheit. Also, it would be helpful to show the change in temperature as a function of the distance into the window pane.

The modeler needs to know before building a new model what the approximate expected resultant values are for the particular properties of the materials selected for use in the model. A firm understanding of the basic physics involved and the appropriate conservation laws that apply to the model are required for analysis, understanding, and configuration of the Postprocessing presentation(s).

The plot presentation is changed as follows: Select Postprocessing > Plot Parameters from the menu bar. When the Plot Parameters window shown in Figure 1.17 appears, select "Line" (Figure 1.18) and then " $^{\circ}F$  (degF)" from the Unit pull-down bar.

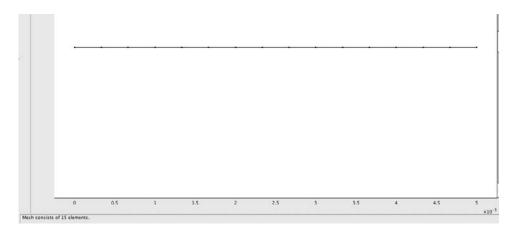


FIGURE 1.14 1D single-pane window with initialized mesh

Select "Use expression to color line." Click the Color Expression button to display the Line Color Expression window (Figure 1.19). Select "°F (degF)" from the Unit pulldown bar. Click OK, and then click OK again. The Plot Presentation will be rendered as shown in Figure 1.20.

#### **1D Single-Pane Analysis and Conclusions**

The 1D single-pane window model, though simple, reveals several fundamental factors about the physics of heat flow through the single-pane window. The interior temperature T\_in was established at 70 °F. The exterior temperature T\_out was established at 0 °F. The calculated temperature at the midpoint of the single pane is the median value

**FIGURE 1.16** 1D single-pane window solution plotted using default values

	Plot Parameters
	General Line Max/Min Animate
Plot type	Solution to use
Max/min marker	Time: Solution at angle (phase): 0 degrees
	Frame:
	Geometries to use
	Element refinement: 🗹 Auto 4
ot in: Main axes	🗧 🗌 Keep current plot
Smoothing)	Title

**FIGURE 1.17** 1D single-pane Plot Parameters General window

		Plot Pa	arameters		
(	General	Line	Max/Min	Anin	nate
🗹 Line plot					
🗹 Height data					
Predefined quantities	Temper	ature		\$	
Expression:	т				Smooth
Unit:	°F			\$	
<ul> <li>Use expression to</li> <li>Uniform color</li> </ul>	o color lines		Color	Expres	sion
	(	Help	) ( AI	oply )	(Cancel) (0

**FIGURE 1.18** 1D single-pane Plot Parameters Line window

of the interior and exterior temperatures, 35 °F. The temperature difference between the inner surface of the pane and the outer surface of the pane is approximately 2 °F.

The temperature difference between the air in the heated room (70 °F) and the interior surface of the single pane (35.9 °F) is approximately 34 °F. This temperature difference between the ambient temperature and the single-pane window will at least result in water vapor condensation (fogging) and heat loss to the exterior.

NOTE	When building	models, be sure	to save early and often.
------	---------------	-----------------	--------------------------

Line color data	
Predefined quantities:	Temperature Range
Expression:	Т
Unit:	°F \$

**FIGURE 1.19** 1D single-pane Line Color Expression window

FIGURE 1.20 1D single-pane window solution plotted using °F and Color Bar

#### **1D Dual-Pane Heat Flow Model**

This 1D model explores the physics of a dual-pane window with an air space between the panes. This model is parametrically similar to the single-pane window model for ease of comparison of the modeling results.

Run the COMSOL Multiphysics application. Select "New" and then select "1D" in the Model Navigator. Select COMSOL Multiphysics > Heat Transfer > Conduction > Steady-state analysis. Click OK.

After the 1D workspace appears, use the menu bar to select Options > Constants. Import the file ModelOC\_1D\_WP1.txt saved earlier. To import this file, click on the Folder icon in the lower-left corner of the Constants window. These imported entries, as shown in Figure 1.21, define the Interior Temperature, the Exterior Temperature, and the air pressure for use in this model and the models to follow.

Modeling a dual-pane window requires that three lines be drawn in the workspace window. The drawn lines represent the left (first) pane, the air space, and the right (second) pane, respectively. To use the menu bar to draw the first line, select Draw > Specify Objects > Line. Enter 0.000 space 0.005 in the window, as shown in Figure 1.22. Leave the default Polyline, and click the OK button. Next, use the menu bar to draw the second line. Select Draw > Specify Objects > Line. Enter 0.005 space 0.015 in the window, as shown in Figure 1.23. Leave the default Polyline, and click the OK button. Next, use the Menu bar to draw the window, as shown in Figure 1.23. Leave the default Polyline, and click the OK button.

Name	Expression	Value	Description
T_in	70[degF]	294.261111[K]	Interior Temperature
T_out	0[degF]	255.372222[K]	Exterior Temperature
p	1[atm]	1.01325e5[Pa]	air pressure
_			) + +

**FIGURE 1.21** 1D dual-pane Constants specification window

button. Finally, using the menu bar, draw the third line. Select Draw > Specify Objects > Line. Enter 0.015 space 0.020 in the window. Leave the default Polyline, and click the OK button. Figure 1.24 shows the results of the model line creation before clicking the Zoom Extents icon in the toolbar.

Once the Zoom Extents icon is clicked, the specified dual-pane Window model will appear in the workspace as shown in Figure 1.25.

Next, using the menu bar, select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select "1" in the Subdomain selection window. For this model, select Load > Basic Materials Properties > Silica Glass. Click OK. All the appropriate values displayed in the Value/Expression subwindows in the Subdomain Settings window are altered as the new values are loaded from the library. (Silica glass has a thermal conductivity value roughly 0.35% of that of copper.)

Once the thermal conductivity is loaded from the materials properties library, enter T\_in in the  $T_{ext}$  and  $T_{ambtrans}$  windows, as shown in Figure 1.26. Click the Apply button.

Next, select "2" in the Subdomain selection window. For this model, select Load > Basic Materials Properties > Air, 1 atm. Click OK. All the appropriate values displayed in the Value/Expression subwindows in the Subdomain Settings window are altered as the new values are loaded from the library.

Coord	inates	ОК
x: 0.	000 0.005	
Style:	Polyline	Cancel
Name:	11	Apply

**FIGURE 1.22** 1D dual-pane Line specification window for the left pane

Coord	inates	
x: 0.	005 0.015	
Style:	Polyline	Car
Name:	12	

**FIGURE 1.23** 1D dual-pane Line specification window for the air gap

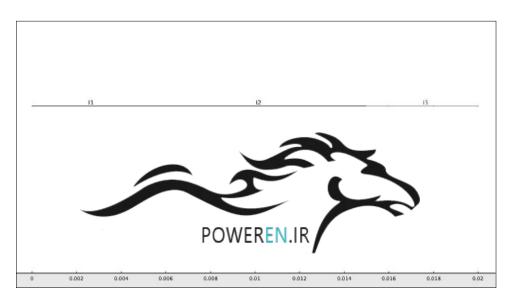
Once the thermal conductivity is loaded from the materials properties library, enter T\_in in the  $T_{ext}$  and  $T_{ambtrans}$  windows, as shown in Figure 1.27. Click the Apply button.

Next, select "3" in the Subdomain selection window. For this model, Select Library materials list > Silica Glass. The previously selected values loaded from the materials library are loaded into this subdomain.

Once the thermal conductivity is loaded from the materials properties library, enter T\_out in the  $T_{ext}$  and  $T_{ambtrans}$  windows, as shown in Figure 1.28. Click the Apply button.

Next, set the initial conditions for each subdomain (1, 2, 3) by clicking the Init button and then entering the initial conditions shown in Table 1.1. Click OK.

Now, the modeler needs to enter the appropriate Boundary Settings. Using the menu bar, select Physics > Boundary Settings, as shown in Figure 1.29.



**FIGURE 1.25** 1D dual-pane workspace after clicking the Zoom Extents icon

Once the Boundary Settings window appears, select "1" in the Boundary selection window. Select "Heat flux" as the Boundary condition. Enter 15 in the Heat transfer coefficient window (h). Enter T\_in in the External temperature window ( $T_{inf}$ ). Enter T\_in in the Ambient temperature window ( $T_{amb}$ ). Click the Apply button. Figure 1.30 shows the filled-in Boundary Settings window for boundary 1.

	+ C <sub>trans</sub> (1 <sub>ambtrans</sub> <sup>4</sup> - 1 <sup>4</sup> ),			
Subdomains Groups		Physics Init	Element	Color/Style
Subdomain selection	-Thermal propert	ties and heat sources/sir	nks	
1	Library material	Air, 1 atm	(load.)	
3	Quantity	Value/Expression	Unit	Description
	k	k(T[1/K])[W/(m*K)]	W/(m·K)	Thermal conductivity
	Q	0	W/m <sup>3</sup>	Heat source
	h <sub>trans</sub>	0	W/(m <sup>3</sup> ·K)	Convective heat transfer coefficient
	T <sub>ext</sub>	T_in	к	External temperature
Group:	C <sub>trans</sub>	0	] W/(m <sup>3</sup> ⋅K <sup>4</sup> )	User-defined constant
Select hy group			ТК	
Active in this domain	Tambtrans	T_in	ĸ	Ambient temperature

**FIGURE 1.27** 1D dual-pane Subdomain Settings Physics window settings, air gap

# Table 1.1 Subdomain Settings, Initial Conditions

Subdomain	1	2	3
Init	T_in	T_in	T_out

$V \cdot (kVI) = Q + h_{trans}(I_{ext}-I)$	trans' ambtrans	1012 M 1012 C		
Subdomains Groups	_	Physics Init	Element	Color/Style
Subdomain selection	Thermal propert	ties and heat sources/si	nks	
1 2	l ibrary material	Silica Glass	load	
3	Quantity	Value/Expression	Unit	Description
	k	1.38[W/(m*K)]	W/(m·K)	Thermal conductivity
	Q	0	W/m <sup>3</sup>	Heat source
	h <sub>trans</sub>	0	W/(m <sup>3</sup> ⋅K)	Convective heat transfer coefficien
Group:	T <sub>ext</sub>	T_out	к	External temperature
Group: 📫	C <sub>trans</sub>	0	W/(m <sup>3</sup> ·K <sup>4</sup> )	User-defined constant
Active in this domain	Tambtrans	T_out	к	Ambient temperature

**I FIGURE 1.28** 1D dual-pane Subdomain Settings Physics window settings, right pane

Boundaries Groups		Coefficient	s Color	
Boundary selection	Boundary sources an			
2	Boundary condition:	Thermal insulation	A Y	
3	Quantity	Value/Expression	Unit	Description
4	q <sub>0</sub>	0		Inward heat flux
	h	0		Heat transfer coefficient
	T <sub>inf</sub>	0		External temperature
Group:	Const	0		Problem-dependent constant
Select by group	Tamb	0		Ambient temperature
Interior boundaries	Τ <sub>ο</sub>	0		Temperature

**FIGURE 1.29** 1D dual-pane Boundary Settings window

Now select "4" in the Boundary Settings Boundary selection window. Select "Heat flux" as the Boundary condition. Enter 15 in the Heat transfer coefficient window (h). Enter T\_out in the External temperature window ( $T_{inf}$ ). Enter T\_out in the Ambient temperature window ( $T_{amb}$ ). Click the Apply button. Figure 1.31 shows the filled-in Boundary Settings window for boundary 4. Click OK.

Boundaries Groups	1	Coefficien	ts Color	
Boundary selection	Boundary sources an	Consideration		
2	Boundary condition:	Heat flux	¢	
3	Quantity	Value/Expression	Unit	Description
4	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
	h	15	W/(m <sup>2</sup> ·к)	Heat transfer coefficient
	T <sub>inf</sub>	T in	к	External temperature
	Const	0	W/(m <sup>2</sup> K <sup>4</sup> )	Problem-dependent constan
Group:	Tamb	T_in	к	Ambient temperature
Interior boundaries	Τ <sub>ο</sub>	0	к	Temperature

**FIGURE 1.30** Filled-in 1D dual-pane Boundary Settings window for boundary 1

		(a. 10)		
Boundaries Groups		Coefficien	ts Color	
Boundary selection	Boundary sources Boundary condition		•	
3	Quantity	Value/Expression	Unit	Description
4	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
	h	15	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
	T <sub>inf</sub>	T_out	к	External temperature
	Const	0	W/(m <sup>2</sup> ·K <sup>4</sup> )	Problem-dependent constant
Group: 🕴 🗘	T <sub>amb</sub>	T_out	к	Ambient temperature
Interior boundaries	Τ <sub>0</sub>	0	к	Temperature

FIGURE 1.31 Filled-in 1D dual-pane Boundary Settings window for boundary 4

At this point, a new modeler probably wonders why no conditions have been specified for boundaries 2 and 3. The COMSOL Multiphysics software default condition is to automatically establish continuity for interior boundaries. The numbers for boundaries 2 and 3 are grayed out to indicate that they are not available for setting. The default boundary settings can be overridden, if needed, by the advanced modeler by clicking the Interior boundaries check box to make boundaries 2 and 3 accessible.

Once all the Subdomain Settings and the Boundary Settings for this model have been either chosen or entered, the next step is to mesh the model. In this simple model, all the modeler needs to do is use the menu bar and Select "Initialize Mesh." Figure 1.32 shows the initial mesh with 16 elements.

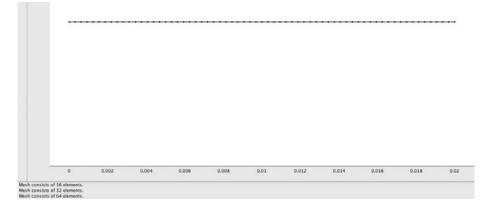
The mesh will be refined twice to improve the resolution. All the modeler needs to do is use the toolbar and select Refine Mesh > Refine Mesh. The refined mesh of the dual-pane model now contains the 64 elements shown in Figure 1.33, rather than the original 16 elements shown in Figure 1.32.

For this model, the software will be allowed to automatically select the Solver and Solver Parameters. To solve this model, go to the menu bar and select Solve > Solve Problem. The solution is found almost immediately, in 0.317 second (the time to solution will vary, depending on the platform). The solution is plotted using the default Postprocessing values and is shown in Figure 1.34.

**FIGURE 1.32** 1D dual-pane window with air gap with initialized mesh

The implicit assumption here would be that the default values in the Postprocessing window are the correct values that the modeler needs to plot the calculated results of the built model. Specifically, that assumption in general will not be true. The modeler needs to know before building a new model what the approximate expected resultant values are for the particular properties of the materials selected for use in the model. A firm understanding of the basic physics involved and the appropriate conservation laws that apply to the model are required for analysis, understanding, and configuration of the Postprocessing presentation(s).

The plot presentation is changed as follows: Select Postprocessing > Plot Parameters from the menu bar. When the Plot Parameters window shown in Figure 1.35 appears, select "Line" (Figure 1.36) and then "°F (degF)" from the Unit pull-down bar (Figure 1.37). Select "Use expression to color lines." Click the Color Expression button



**FIGURE 1.33** 1D dual-pane window with air gap with refined mesh

**FIGURE 1.34** 1D dual-pane window solution plotted using default values

	Plot Parameters
	General Line Max/Min Animate
Plot type	Solution to use
🗹 Line	Solution at time: 0
Max/min marker	Time:
🗹 Geometry edges	Solution at angle (phase): 0 degre
	Frame:
	Geometries to use
	Geom1
	Element refinement: 🗹 Auto 🛛 4
<u></u>	C Keep current plot
of in Main axes	
ot in: Main axes	

**FIGURE 1.35** 1D dual-pane Plot Parameters General window

🗹 Height data		
Predefined quantities:	Temperature	-
Expression:	Т	Smooth
Unit:	(K	
Line color		
O Use expression to	color lines Color Expr	ession
<ul> <li>Uniform color</li> </ul>	Color	

**FIGURE 1.36** 1D dual-pane Plot Parameters Line window

	Plot Parameters	
	General Line Max/Min Anima	ite
Line plot		
🗹 Height data		
Predefined quantities:	Temperature	
Expression:	Т	Smooth
Unit:	∫°F ♠	
Line color		
O Use expression to	color lines Color Express	ion
Uniform color	Color	
0		

**FIGURE 1.37** 1D dual-pane window solution set to use °F

🗹 Line plot 🗹 Height data		
Predefined quantities:	Temperature	)
Expression:	Т	Smooth
Unit:	_ °F*	)
Line color		
O Use expression to	color lines Color Expre	ession
🔘 Uniform color	Color)	

**FIGURE 1.38** 1D dual-pane window solution set to use Color Expression

(Figure 1.38). Select "°F (degF)" from the Unit pull-down bar (Figure 1.39). Click OK, and then click OK again. The plot presentation will be rendered as shown in Figure 1.40.

### **Dual-Pane Analysis and Conclusions**

The 1D dual-pane window model, though simple, reveals several fundamental factors about the physics of heat flow through the dual-pane window. The interior temperature T\_in was established at 70 °F, as in the single-pane model. The exterior temperature T\_out was established at 0 °F, as in the single-pane model. The calculated temperature at the midpoint of the single pane is the median value of the interior and

Predefined quantities:	Temperature Range
Expression:	Т
Unit:	°F \$

FIGURE 1.39 1D single-pane window solution set to use °F

FIGURE 1.40 1D dual-pane window solution plotted using °F and Color Bar

exterior temperatures, 35 °F. The temperature difference between the inner surface of the left pane and the outer surface of the left pane is approximately 0.48 °F. The temperature difference between the inner surface of the right pane and the outer surface of the right pane is also approximately 0.48 °F. The temperature difference between the inner surface of the dual-pane window is approximately 53 °F, as compared to approximately 2 °F for the single-pane window.

The temperature difference between the air in the heated room (70 °F) and the interior surface (61.5 °F) of the dual-pane window is approximately 8.5 °F. This small temperature difference will result in some heat loss and minimal water vapor condensation (fogging).

Compare the result for the dual-pane window to that of the single-pane window. The temperature difference between the air in the heated room (70 °F) and the interior surface of the single-pane window (35.9 °F) is approximately 34 °F. This temperature difference between the ambient temperature and the single-pane window will at least result in water vapor condensation (fogging) and heat loss to the exterior.

### **1D Triple-Pane Heat Flow Model**

This 1D model explores the physics of a triple-pane window with an air space between each pair of panes. This model is parametrically similar to the single-pane and dualpane models for ease of comparison of the modeling results.

Name	Expression	Value	Description	-
T_in	70[degF]	294.261111[K]	Interior Temperature	1
T_out	0[degF]	255.372222[K]	Exterior Temperature	1
p	1[atm]	1.01325e5[Pa]	air pressure	
_			) 4 >	

**FIGURE 1.41** 1D triple-pane Constants specification window

Run the COMSOL Multiphysics application. Select "New" and then "1D" in the Model Navigator. Select COMSOL Multiphysics > Heat Transfer > Conduction > Steady-state analysis. Click the OK button.

After the 1D workspace appears, use the menu bar to select Options > Constants. Import the file ModelOC\_1D\_WP1.txt saved earlier. To import this file, click on the Folder icon in the lower-left corner of the Constants window. These imported entries, as shown in Figure 1.41, define the Interior Temperature, the Exterior Temperature, and the air pressure for use in this model and the models to follow.

Modeling a triple-pane window requires that five lines be drawn in the workspace window. The drawn lines represent the left (first) pane, the first air space, the center (second) pane, the second air space, and the right (third) pane, respectively. Use the toolbar to draw the first line: Select Draw > Specify Objects > Line. Enter 0.000 space 0.005 in the window. Leave the default Polyline, and then click the OK button. Next, use the menu bar to draw the remaining four lines, as indicated in Table 1.2. Then, click the Zoom Extents icon in the toolbar. The finished workspace configuration is shown in Figure 1.42.

Line	Start	End
1	0.000	0.005
2	0.005	0.015
3	0.015	0.020
4	0.020	0.030
5	0.030	0.035

### Table 1.2 Triple-Pane Window Workspace Lines

**FIGURE 1.42** 1D triple-pane workspace after clicking the Zoom Extents icon

Next, using the menu bar, select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select "1" in the Subdomain selection window. For this model, select Load > Basic Materials Properties > Silica Glass. Click OK (see Figure 1.43). Enter the remaining Subdomain Settings as shown in Table 1.3.

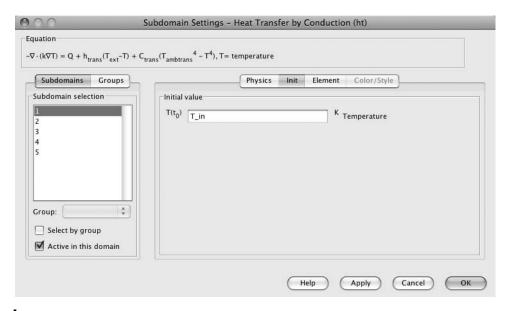
Subdomain	Material	T <sub>ext</sub>	T <sub>ambtrans</sub>
1	Silica glass	T_in	T_in
2	Air	T_in	T_in
3	Silica glass	T_in	T_in
4	Air	T_in	T_in
5	Silica glass	T_out	T_out

### Table 1.4 Subdomain Settings, Initial Conditions

Subdomain	1	2	3	4	5
Init Setting	T_in	T_in	T_in	T_in	T_out

Next, set the initial conditions for each subdomain (1, 2, 3, 4, 5) by clicking the Init button and then entering the initial conditions shown in Table 1.4. See Figure 1.44.

Having configured the Subdomain Settings for this model, the modeler needs to enter the appropriate Boundary Settings. Using the menu bar, select Physics > Boundary Settings. The Boundary Settings window appears, as shown in Figure 1.45.



**FIGURE 1.44** 1D triple-pane Subdomain Settings Init window settings

Boundaries Groups		Coefficier	nts Color	
Boundary selection	Boundary sources an	d constraints		
1	Boundary condition:	Heat flux	A V	
3	Quantity	Value/Expression	Unit	Description
5	q <sub>0</sub>			Inward heat flux
6	h			Heat transfer coefficient
	T <sub>inf</sub>			External temperature
	Const			Problem-dependent constant
Group:	Tamb			Ambient temperature
Select by group	To			
Interior boundaries	0			Temperature

**FIGURE 1.45** Blank 1D triple-pane Boundary Settings window

Once the Boundary Settings window appears, select "1" in the Boundary selection window. Select "Heat flux" as the Boundary condition. Enter 15 in the Heat transfer coefficient window (h). Enter T\_in in the Ambient temperature window ( $T_{inf}$ ). Enter T\_in in the External temperature window ( $T_{amb}$ ). Click the Apply button. Figure 1.46 shows the filled-in Boundary Settings window for boundary 1. Fill in the

	unu		_	
Boundaries Groups		Coefficien	ts Color	
Boundary selection	-Boundary sources ar	nd constraints		
2	Boundary condition	Heat flux	\$	
3	Quantity	Value/Expression	Unit	Description
4	q <sub>o</sub>	0	W/m <sup>2</sup>	Inward heat flux
6	h	15	W/(m <sup>2</sup> ⋅K)	Heat transfer coefficient
	T <sub>inf</sub>	T in	к	External temperature
Group:	Const	0	W/(m <sup>2</sup> K <sup>4</sup> )	Problem-dependent constan
	Tamb	T_in	к	Ambient temperature
Select by group	To	0	— к	Temperature

**FIGURE 1.46** Filled-in 1D triple-pane Boundary Settings window for boundary 1

Boundary	1	2	3	4	5	6
condition	Heat flux	*	*	*	*	Heat flux
h	15	*	*	*	*	15
T <sub>inf</sub>	T_in	*	*	*	*	T_out
T <sub>amb</sub>	T_in	*	*	*	*	T_out

### Table 1.5Boundary Settings

remaining Boundary Settings as shown in Table 1.5. Click the Apply button after each entry, and click OK at the end of the process.

At this point, as mentioned in the discussion of the dual-pane window model, no conditions are specified for boundaries 2, 3, 4, and 5, because the COMSOL Multiphysics software default condition is to automatically establish continuity for interior boundaries. The numbers for boundaries 2, 3, 4, and 5 are grayed out to indicate that they are not available for setting. The default boundary settings can be overridden, if needed, by the advanced modeler.

Once all the Subdomain Settings and the Boundary Settings for this model have been either chosen or entered, the next step is to mesh the model. In this model, all the modeler needs to do is use the toolbar and select "Initialize Mesh." Figure 1.47 shows the initial mesh with 16 elements.

To improve the resolution, the mesh will be refined twice. All the modeler needs to do is use the menu bar and select Refine Mesh > Refine Mesh. The refined mesh of the triple-pane model now contains the 64 elements shown in Figure 1.48, rather than the original 16 elements shown in Figure 1.47.

**FIGURE 1.48** 1D triple-pane window with air gaps with refined mesh

For this model, the software will be allowed to automatically select the Solver and Solver Parameters. To solve this model, go to the menu bar and select Solve > Solve Problem. The solution is found almost immediately, in 0.346 second (the length of time to solution will vary depending on the platform). The solution is plotted using the default Postprocessing values and is shown in Figure 1.49.

FIGURE 1.50 1D triple-pane window solution plotted using °F and Color Bar

NOTE As noted earlier in the solutions of the single-pane and dual-pane models, the postprocessing parameters need to be altered to reveal the most information at a glance.

The plot presentation is changed as follows: Select Postprocessing > Plot Parameters from the menu bar. When the Plot Parameters window appears, select "Line" and then "°F (degF)" from the Unit pull-down bar. Select "Use expression to color lines." Click the Color Expression button. Select "°F (degF)" from the Unit pull-down bar. Click OK and then click OK again. The plot presentation will be rendered as shown in Figure 1.50.

### **Triple-Pane Analysis and Conclusions**

The 1D triple-pane window model reveals several fundamental factors about the physics of heat flow through the triple-pane window. The interior temperature T\_in was established at 70 °F, as in the single-pane and dual-pane models. The exterior temperature T\_out was established at 0 °F, as in the single-pane and dual-pane models. The temperature difference between the inner surface and the outer surface of the three different window types are compared as shown in Table 1.6.

The temperature difference between the air in the heated room (70 °F) and the interior surface of the triple-pane window (65.2 °F) is approximately 4.8 °F. This

	Single	Dual	Triple
$\Delta$ T (°F):	2	53	60
across all panes $\Delta$ T (°F):	34	8.5	4.8
inner pane surface to room ambient temp	perature		

Table 1.6 Comparison of Single-, Dual-, and Triple-Pane Windows

minimal temperature difference will result in little heat loss and little, if any, water vapor condensation (fogging).

Comparing the results for the three different window configuration models shows that there will be a large reduction in heat loss and annoyance factors (condensation) associated with a change from a single-pane window to a dual-pane window design. The incremental cost of such a design change would typically be less than 100%. However, adding the third pane to the window design reduces the heat loss by only a few percentage points and adds little to the cosmetic enhancement of the design (lack of fogging).

NOTE One of the basic reasons for modeling potential products is to evaluate their relative performance before the actual building of a first experimental physical model. Comparison of these three window models allows such a comparison to easily be made on a "first principles" basis, which will be discussed in the following section of this chapter. That approach is known as the "Model first, build second" approach to engineering design. When the model properly incorporates the fundamental materials properties and design factors, both the time and the cost to develop a fully functional prototype product are significantly reduced.

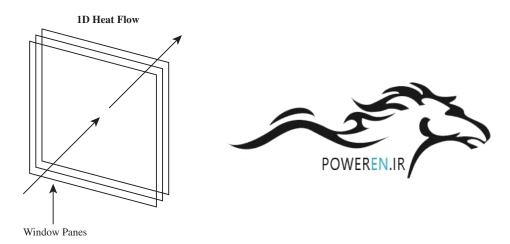
## First Principles Applied to Model Definition

*First principles analysis* is an analysis whose basis is intimately tied to the fundamental laws of nature. In the case of models described in this book, the modeler should be able to demonstrate both to himself or herself and to others that the calculated results derived from those models are consistent with the laws of physics and the observed properties of materials. Basically, the laws of physics require that what goes in (e.g., as mass, energy, charge) must come out (e.g., as mass, energy, charge) or must accumulate within the boundaries of the model. Note In the COMSOL Multiphysics software, the default interior boundary conditions are set to apply the conditions of continuity in the absence of sources (e.g., heat generation, charge generation, molecule generation) or sinks (e.g., heat loss, charge recombination, molecule loss).

The careful modeler will be able to determine by inspection that the appropriate factors have been considered in the development of the specifications for the various geometries, for the material properties of each subdomain, and for the boundary conditions. He or she must also be knowledgeable of the implicit assumptions and default specifications that are normally incorporated into the COMSOL Multiphysics software model, when a model is built using the default settings.

Consider, for example, the three window models developed earlier in this chapter. By choosing to develop those models in the simplest 1D geometrical mode, the implicit assumption was made that the heat flow occurred in only one direction. That direction was basically normal to the surface of the window and from the high temperature (inside temperature) to the low temperature (outside temperature), as shown by the heat flow indicator in Figure 1.51.

That assumption essentially eliminates the consideration of heat flow along other paths, such as through the window frame, through air leaks around the panes, and so forth. It also assumes that the materials are homogeneous and isotropic, and that there are no thin thermal barriers at the surfaces of the panes. None of these assumptions is typically true in the general case. However, by making such assumptions, it is possible to easily build a first approximation model.



**FIGURE 1.51** 1D triple-pane window with heat flow indicator

NOTE A first approximation model captures all of the essential features of the problem that needs to be solved, without dwelling excessively on minutiae. A good first approximation model will yield an answer that is sufficiently accurate to enable the modeler to determine whether he or she needs to invest the time and resources necessary to build a higher-dimensionality, significantly more-accurate model.

## Common Sources of Modeling Errors

There are four primary sources of modeling errors: insufficient model preparation time, insufficient attention to detail during the model preparation and creation phase, insufficient understanding of the physical and modeling principles required for the creation of an adequate model, and lack of a comprehensive understanding of what defines an adequate model in the modeler's context. The most common modeling errors are those that result from the modeler taking insufficient care in either the development of model details or the incorporation of conceptual errors and/or the generation of keying errors during data/parameter/formula entry.

Note One primary source of errors occurs during the process of naming variables. The modeler should be careful to *never give the same name to his or her variables as COMSOL gives to the default variables*. COMSOL Multiphysics software seeks a value for the designated variable everywhere within its operating domain. If two or more variables have the same designation, an error is created. Also, it is best to avoid human errors by using uniquely distinguishable characters in variable names (i.e., avoid using the lowercase "L," the number "1," and the uppercase "I," which in some fonts are relatively indistinguishable; similarly, avoid the uppercase "O" and the number zero "0"). Give your variables meaningful names (e.g., T\_in, T\_out, T\_hot). Also, variable names are case sensitive; that is, T\_in is not the same as T\_IN.

The first rule in model development is to define the nature of the problem to be solved and to specify in detail which aspects of the problem the model will address. The definition of the nature of the problem should include a hierarchical list of the magnitude of the relative contribution of physical properties vital to the functioning of the anticipated model and their relative degree of interaction.

Examples of typical physical properties that are probably coupled in any developed model are heat and geometrical expansion/contraction (liquid, gas, solid), current flow and heat generation/reduction, phase change and geometrical expansion/ contraction (liquid, gas, solid) and/or heat generation/reduction, and chemical reactions. Be sure to investigate your problem and build your model carefully.

Having built the hierarchical list, the modeler should then estimate the best physical, least-coupled, lowest-dimensionality modeling approach to achieve the most meaningful first approximation model.

## Exercises

- 1. Build, mesh, and solve the 1D single-pane window problem presented earlier in this chapter.
- 2. Build, mesh, and solve the 1D dual-pane window problem presented earlier in this chapter.
- 3. Build, mesh, and solve the 1D triple-pane window problem presented earlier in this chapter.
- 4. Add a fourth pane, and build, mesh, and solve the problem. Analyze, compare, and contrast the results with the results of Exercises 1, 2, and 3.



# **Materials and Databases**

# In This Chapter

Materials and Database Guidelines and Considerations COMSOL<sup>®</sup> Material Library Module: Searchable Materials Library MatWeb: Searchable Materials Properties Website PKS-MPD: Searchable Materials Properties Database

# Materials and Database Guidelines and Considerations

Materials selection and definition are the most important tasks performed by the modeler during the preliminary stages of model building preparation. The selection of appropriate materials is vital to the ultimate functionality of the device or process being modeled. Once the modeler has decided on a good first approximation to the device/process being modeled, the materials selection process begins.

NOTE A good first approximation is a problem statement that incorporates all the essential (first-order) physical properties and functionality of the device/process to be modeled.

Not all properties of all materials are or can be incorporated into the modeling process at the same time, because modeling resources (e.g., computer memory, computer speed, number of cores) are limited. It is important that the modeler start the modeling process by building a model that incorporates the most critical physical and functional aspects of the developmental problem under consideration.

To put the problem in perspective, a simple search of the Web on the term "materials properties database" yields approximately 48 million hits. Obviously, the modeler is not going to exhaustively explore all such links.

**NOTE** Exploration of any given subset of the 48 million links for the properties of specific materials will reveal several possible standard results: (1) those links do not have a value for the desired material property; (2) those links do have values for the desired material property, but in unconventional units that need to be converted and then

compared for relative accuracy and reliability; (3) some of those links do have values for the desired material property, in the desired units, that need to be compared for relative accuracy and reliability; (4) a link is found that has some of the desired properties for a particular material, but not all of the needed properties; and (5) once found, property values need to be hand-copied and hand-entered into the model. In any case, many hours can be spent trying to obtain and determine accurate values for specific properties of particular materials.

COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software and the associated add-on modules include basic materials libraries. The information contained in those basic materials libraries may easily be enhanced by the addition of other materials properties data through several different means. This chapter discusses three solutions to the obtaining and supplying of materials properties values directly to COMSOL Multiphysics software models. Each of these three solutions approaches the problem solution from a different viewpoint, with the same desired result—that is, supplying the modeler with the best materials properties values available to meet the modeler's needs.

### COMSOL® Material Library Module: Searchable Materials Library

The COMSOL Material Library<sup>1</sup> is a module that can be added through licensing to the basic COMSOL Multiphysics software package to expand the included basic library. The COMSOL Material Library Module has data on approximately 2500 materials including elements, minerals, soil, metal alloys, oxides, steels, thermal insulators, semiconductors, and optical materials, at the minimum. It is searchable by name, DIN,<sup>2</sup> and UNS<sup>3</sup> numbers. Each material can have a maximum of 27 defined properties. Each of those defined properties is available as a function of temperature.

There are two methods to gain access to the Material Library. The first method is through the Options menu. This path can be used to screen materials in advance of building a model. The second method is through the Load button on the Subdomain Settings page. This path incorporates the materials into the model library. Once the Material Library Module has been activated, the technique for using the library is the same. In this example, the Options menu route is used:

- 1. Activate the COMSOL Multiphysics application.
- Select COMSOL Multiphysics > Heat Transfer > Convection and Conduction > Steady-state analysis. See Figure 2.1.
- 3. Click OK.
- 4. Select Options > Materials/Coefficients Library. See Figure 2.2.

The modeler can determine if the items in the Material Library Module are available by viewing the list in the Materials selection window. The first entry in the

	MOG	lel Navigator			
New	Model Library	User Models	Open	Settings	
Space dimension:	2D	\$		<b>.</b> ,	
Application Modes		M	Heat	Transfer	
<ul> <li>COMSOL Multiphy</li> <li>Acoustics</li> </ul>	ysics				-
Convection at				NUS	2
<ul> <li>Electromagne</li> <li>Fluid Dynamic</li> </ul>					P
		Ĩ.		17/6	
🔻 🚞 Heat Transfer					
🔻 📄 Convectio	on and Conduction		Descriptio		
Convection			Heat trans	fer through convection n with heat flux, convec	tive,
Convection	on and Conduction ient analysis /-state analysis on		Heat trans conductio and tempo	fer through convection n with heat flux, convec erature boundary condit	tive,
<ul> <li>Convection</li> <li>Trans</li> <li>Steady</li> </ul>	on and Conduction ient analysis /-state analysis on		Heat trans conductio and tempo	fer through convection n with heat flux, convec	tive,
Convectio     Trans     Steady     Conductio     Structural Me	on and Conduction ient analysis /-state analysis on		Heat trans conductio and tempo	fer through convection n with heat flux, convec erature boundary condit	tive,
Convection Convection Trans Steady Conduction Conduction Conduction Properties PDE Modes	on and Conduction ient analysis y-state analysis on chanics		Heat trans conductio and tempo	fer through convection n with heat flux, convec erature boundary condit	tive,
Convection	on and Conduction ient analysis /-state analysis on		Heat trans conductio and tempo	fer through convection in with heat flux, conve erature boundary conc	ect

FIGURE 2.1 Model Navigator window

2/0/01	Materials/Coefficients Library
Materials Model (0) Material Library (2542) Basic Material Properties (31) Liquids and Gases (39) MEMS Material Properties (33) Heat Transfer Coefficients (8) Electric (AC/DC) Material Properties Piezoelectric Material Properties User Defined Materials (1) New Delete Copy Paste	- Material properties
Add Library	Enable individual settings
Search for: Name	(\$)
Search string:	÷)
( Search )	Hide undefined properties     Functions
Co To	(Mare Infa) (Plot

FIGURE 2.2 Materials/Coefficients Library search and/or selection window



FIGURE 2.3 Materials/Coefficients Library Materials selection window

Materials selection list is Model (0), which indicates that none of the materials in any of the libraries have been selected for use in the current model. The second entry in the Materials selection list is Material Library (2542); it indicates that 2542 materials are available to be selected for use in the current model. See Figure 2.3.

Suppose, for example, the modeler is interested in the properties of copper and copper alloys. He or she would follow these steps:

- 1. Enter "copper" in the Search string window.
- 2. Click the Search button. The search results show 87 possible materials are in the library. See Figure 2.4.

0.0	Materials/Coefficients Library
Materials  Cast Irons & Mold materials  Search results 1574	-Material properties-
Copper CY219/HY219/CY219 2	solid
Carbon fiber/copper Copper Alloys (84)	residual resistivity ratio of 100 🗘
Basic Material Properties (31)     Liquids and Gases (39)     MEMS Material Properties (33)     Heat Transfer Coefficients (8)     New Delete     Copy Paste     Add Library	Search results: 87 materials
Search	Enable individual settings
Search for: Name	•
Search string: copper	\$
Search	Hide undefined properties  Functions
Go To	(More Info Plot

FIGURE 2.4 Materials/Coefficients Library search results

laterials	-Material propertie	s		
Cast Irons & Mold materials	Name: UNS C10	100		
▼ Search results (87)	DIN number:		UNS num	ber: C10100
Copper CY219/HY219/CY219 2	Phase/Condition	-	solid	\$
Carbon fiber/copper	Orientation/Con	dition:	residual resistivity rat	tio of 100
Copper Alloys (84)	Elastic	Electric	Fluid Piezoelect	ric Thermal All
UNS C10200 UNS C10300	Quantity	Value	/Expression	Description
UNS C10400	C C01 C10 CS	C_sol	id_1(T[1/K])[J/(kg*K)]	Heat capacity at c Model parameter ( Model parameter ( Creep strength
New         Delete           Copy         Paste	CTE D	CTE(1	[1/K])[1/K]	Instantaneous coe A Diffusion coefficient
Add Library				
Search	Enable indi	vidual se	ettings	
Search for: Name	Phase/Conditio	n:		4 ¥
Search string: copper	Orientation/Co	ndition:		\$
Search	🗌 Hide undefir	ned prop	erties	Functions
Go To			More	Info Plot

FIGURE 2.5 Materials/Coefficients Library, UNS C10100 Material properties window

- Select Copper Alloys (84) > UNS C10100. The UNS C10100 is what is known as oxygen-free copper.<sup>4</sup> This high-quality copper is widely used in the electronics industry. The properties of UNS C10100 are shown in the Material properties display window. See Figure 2.5.
- 4. To see only the defined properties, check the Hide undefined properties check box. See Figures 2.6 and 2.7.

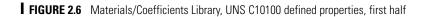
A similar process can be followed for other material choices.

## MatWeb: Searchable Materials Properties Website

MatWeb<sup>5</sup> is an online searchable subscription materials properties data source. MatWeb has 69,000 data sheets for materials that include plastics, metals, ceramics, semiconductors, fibers, and various other commercially available materials. See Figure 2.8.

MatWeb has three classes of access: Un-Registered (free limited feature access), Registered Member (free expanded feature access), and Premium Member (fee-based access to all features, plus selected data storage and modeling software formatted data 44

	and the second second second second		
Materials	Material properties		
▶ Cast Irons & Mold materials	Name: UNS C10100		
▼ Search results (87)	DIN number:	UNS number: C10100	
Copper	Phase/Condition:	solid	0
CY219/HY219/CY219 2 Carbon fiber/copper		(	
Copper Alloys (84)	Orientation/Condition	residual resistivity ratio of 100	4
UNS CLOLOG	Elastic Elect	ric Fluid Piezoelectric Thermal	All
UNS C10200			
UNS C10300		lue/Expression Description	
UNS C10400		solid_1(T[1/K])[J/(kg*K)] Heat capacity at c E(T[1/K])[1/K] Instantaneous co	
		[1/K])[Pa] Young's modulus	
		ha(T[1/K])[1/K]+(Tempr Thermal expansion	
New Delete	Sectors	(T[1/K])-dL(Tempref[1/K]) Linear expansion	and the second sec
Copy Paste	k k	solid residual resistivity Thermal conduct	I 🔻
( Add Library )			
Search	Enable individua	settings	
Search for: Name	Phase/Condition:		(†
	Orientation/Conditio		\$
Search string: copper	one manony conditio		
( Search )	Hide undefined p	operties Funct	ions
( Go To)		(More Info ) Pl	ot



Materials	Material properties		
► Cast Irons & Mold materials	Name: UNS C10100		
▼ Search results (87)	DIN number:	UNS numb	er: C10100
Copper U CY219/HY219/CY219 2	Phase/Condition:	solid	
Carbon fiber/copper	Orientation/Condition	n: residual resistivity rati	o of 100
▼ Copper Alloys (84)			
UNS C10100	Elastic Elec	tric Fluid Piezoelectr	ic Thermal All
UNS C10200 UNS C10300	Quantity V	alue/Expression	Description
UNS C10400		L(T[1/K])-dL(Tempref[1/K])	
		_solid_residual_resistivity appa(T[1/K])[Pa]	Initial bulk modulu
		u(T[1/K])[Pa]	Initial shear modul
New Delete	nu nu	u(T[1/K])	Poisson's ratio
	rho rh	no(T[1/K])[kg/m^3]	Density 🔻
Copy Paste			
( Add Library )	a second contract		
Search	Enable individua	al settings	
Search for: Name	Phase/Condition:		\$
	Orientation/Conditio		0
Search string: copper	Onentation/Condition	20.	*
( Search )	Hide undefined p	properties	Functions
Go To		More I	nfo Plot



### MatWeb, Your Source for Materials Information

What is MatWeb'? MatWeb's <u>searchable database of material properties</u> includes data sheets of thermoplastic and thermoset polymers such as ABS, nylon, polycarbonate, polyester, polyethylene and polypropylene; metals such as aluminum, cobalt, copper, lead, magnesium, nickel, steel, superalloys, titanium and zinc alloys; ceramics; plus semiconductors, fibers, and other engineering materials.

#### FIGURE 2.8 MatWeb site, home page

MatWeb Feature	Un-Registered	Registered	Premium
Material Data Sheets	1	1	1
- View Any of MatWeb's 69,000 Data Sheets For FREE			
Basic Search Engines	1	*	1
Koy Word or Phrase     Property, Metric or English Units     Material Type	- Manufacturer - Trade Name	Up to 1500 results	Up to 2000 results
Advanced Search Engine		4	1
- Combines Text, Category, Property, and Composition Se	earches	Search on up to 3 criteria at a time	Search on up to 10 criteria at a time
View Property Data in Search Results	4	v chicha at a tano	
Sort Search Results			4
<ul> <li>Sort by Material Name and Property Data in Search Res</li> <li>Ascending Value, Descending Value, or Alphabetical by</li> </ul>			Sort on any of first 3 numerical criteria
Saved Material Folders		1	1
- Organize Your Most-Referenced Materials - Add, Edit, and Delete Material Folders		Save up to 1 Material Folder	Save up to 10 Material Folders
Compare Folder Materials		1	J
- Side-By-Side Comparison of Material Property Data - Graph Property Value vs Material for any Property		3 Data Sheets per Folder	20 Data Sheets per Folder
Exclude Discontinued Materials		4	1
- Streamline Search Results by Omitting Discontinued Dat	ta Sheets		
Export Material Folder Data			1
Export Materials to CSV/Excel format.     Export Data Sheets to a SolidWorks/COSMOSWorks     Export Data Sheets to the ALGOR Library Format     Export Data Sheets to a NEiWorks Library     Export Data Sheets in the ANSYS Format     Export Data Sheets in the Comsol Format	Library		
- Export Data Sheets in the PlassoTech 3G.Author For	mat		
Basic Tools	1	1	1
<ul> <li>Unit Converter With Over 150 Units of Measure</li> <li>Basic Weight Calculator</li> </ul>			
Advanced Tools		1	1
- Hardness Converter     - Moment of Inertia Calculator     - Searchable Glossary     - Advanced Weight Calculator			

FIGURE 2.9 MatWeb membership level features comparison page

export). All features of the following example can be run (for free) as a Registered user, except for the export feature. To export the selected data, the modeler needs to acquire a Premium membership. See Figure 2.9.

The following example shows the results of a Premium membership search.

- 1. After login, select "Metal UNS Number" on the login home page. See Figure 2.10.
- 2. The Web page is shifted to the Metal Alloy UNS Number Search page. See Figure 2.11.



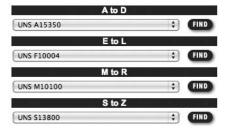


FIGURE 2.11 MatWeb Metal Alloy UNS Number Search selection page

- 3. Select "UNS C10100" from the A to D drop-down list. See Figure 2.12.
- 4. Click the FIND button to the right of the selected material number. The search has found 22 data sheets for UNS C10100 (oxygen-free electronic-grade copper); see Figure 2.13.
- 5. Using the Task pull-down list in the menu bar, create a folder named Copper.
- 6. Select item 1. See Figure 2.14.
- Select "Export to COMSOL" from the task list in the menu bar. The available properties values for UNS C10100 are exported as a text file to the modeler's computer. See Figure 2.15.

The exported file can be directly imported into COMSOL Multiphysics as follows:

- 1. Open COMSOL Multiphysics in the application mode of choice.
- 2. Using the menu bar, select Options > Materials/Coefficients Library.
- 3. Click the Add Library button.

There are many other features available to the Premium Member at the MatWeb website. Those features can be explored at the modeler's convenience.

Use Folder	Contains Other Tasks
My Folder	\$ 5/20 COMPARE MATERIALS task \$
Select	Material Name
01	Oxygen-free electronic Copper, UNS C10100
2	Oxygen-free Electronic Copper (OFE), UNS C10100, H00 Temper, flat products
3	Oxygen-free Electronic Copper (OFE), UNS C10100, H01 Temper, flat products
4	Oxygen-free Electronic Copper (OFE), UNS C10100, H02 Temper, flat products
5	Oxygen-free Electronic Copper (OFE), UNS C10100, H04 Temper, flat products
6	Oxygen-free Electronic Copper (OFE), UNS C10100, H04 Temper, plate (25 mm Thick)
07	Oxygen-free Electronic Copper (OFE), UNS C10100, H04 Temper, wire
8 🖯	Oxygen-free Electronic Copper (OFE). UNS C10100. H08 Temper. flat products (1 mm Thick)
9	Oxygen-free Electronic Copper (OFE), UNS C10100, H08 Temper, wire
🗆 10	Oxygen-free Electronic Copper (OFE), UNS C10100, H10 Temper, flat products
11	Oxygen-free Electronic Copper (OFE), UNS C10100, H55 (15%) temper, tubing
12	Oxygen-free Electronic Copper (OFE), UNS C10100, H80 (15%) temper, shapes
🗆 13	Oxygen-free Electronic Copper (OFE), UNS C10100, H80 (16%) Temper, rod
Ξ 14	Oxygen-free Electronic Copper (OFE), UNS C10100, H80 (35%), rod (25 mm Diameter)
15	Oxygen-free Electronic Copper (OFE), UNS C10100, H80 (40%) rod, tubing (25 mm o.d. and 1.65 wall thickness
🖂 16	Oxygen-free Electronic Copper (OFE). UNS C10100. M20 temper. flat products
17	Oxygen-free Electronic Copper (OFE), UNS C10100, M20 temper, rod and shapes
🖃 18	Oxygen-free Electronic Copper (OFE), UNS C10100, M30 temper, shapes
19	Oxygen-free Electronic Copper (OFE), UNS C10100, OS025 temper, flat products and tubing
20	Oxygen-free Electronic Copper (OFE), UNS C10100, OS050 temper, flat products, rod, tubing, and shapes
21	Oxygen-free Electronic Copper (OFE), UNS C10100, OS050 temper, wire
22	Cadi C10100 Oxygen Free Copper

**FIGURE 2.13** MatWeb search results for UNS C10100 (oxygen-free electronic-grade copper)

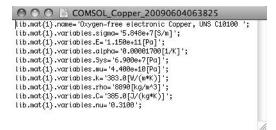


FIGURE 2.15 MatWeb properties of UNS C10100 (oxygen-free electronic-grade copper)

### PKS-MPD: Searchable Materials Properties Database

PKS-MPD (Pryor Knowledge Systems–Materials Properties Database)<sup>6,7</sup> is a searchable materials properties database with data on more than 4000 materials, including elements, minerals, soil, metals, metal alloys, oxides, steels, thermal insulators, semiconductors, optical materials, and biomaterials (tissue). Each material can have a maximum of 43 defined properties. Each of those defined properties is associated with the temperature of measurement and the frequency of measurement, as available. The collection of defined properties for each materials property datum is exportable in a format suitable for the COMSOL Multiphysics software.

The PKS-MPD selection page, on first use, requires that the modeler choose the version of COMSOL Multiphysics software in use to correctly format the export files (the COMSOL Multiphysics version selection choice remains as chosen until later changed). See Figure 2.16.

Using the same example material as previously, UNS C10100 (oxygen-free electronic-grade copper), the selection criteria can be entered by at least two different paths. To use the first path:

- 1. Click the Composition tab and select Copper (Cu) from the pull-down list. See Figure 2.17.
- 2. Click the Add to Search button. See Figure 2.18.

The Selection Criteria window shows that the search yields 440 possible coppercontaining candidate materials. Because oxygen-free electronic-grade copper is known to be very pure, the search can be narrowed by adding a specification of the compositional percentage of Cu to the search.

- 1. Select Copper (Cu) from the element pull-down list.
- 2. Check the Specify percentage range check box.
- 3. Enter Min. = 99.9 and Max. = 100 in the appropriate edit windows. See Figure 2.19.
- 4. Click the Add to Search button. See Figure 2.20.

			PKS-M	IPD			
Exit	View Properties	Print Materials	Export to Model SW	View Log	Clear	Instruc- tions	
<b>5</b> -1	and the second	operty Materia	al Type Comp	osition Other	Characteristics		
Select	a property from t	te list:					
							Add to Search
election C	riteria:			Ma	terials Selected		0
No. F	ield		Criteria			Count	Remove
-							

## **FIGURE 2.16** PKS-MPD main selection page

00						PKS-M	IPD					_	
U Exit	View Proper		Pri	int erials	Expo Mode	rt to el SW	Vie			llear	Instrution		
		Prop	perty	Materia	al Type	Compo	osition	Othe	r Chara	teristics	]		
				Select a	in eleme	nt from t	he list:						
				Coppe	er (Cu)			:					
						entage ra							
election	Criteria								Interial	Colorta	4		Add to Search
Selection	Criteria: Field					Criteria		м	laterial	s Selecte	d : Cou	int	
								M	laterial	s Selecte	1000 C	int	Search
								М	laterial	s Selecte	1000 C	unt	Search

**FIGURE 2.17** PKS-MPD Composition selection page for Copper (Cu)

50

10.200	)			PKS-	MPD			_
Exit	2	View Properties Pro		Export to Model SW erial Type Comp t an element from		Clear er Characteristics	Instruc- tions	
electi	ion Criteria	1:	S;	becify percentage r		Materials Selected :		Add to Search
No.	Field			Criteria		naterials selected .	Count	Remove
NO.				Criteria				E
1	Element	Copper (Cu)					440	

I FIGURE 2.18 PKS-MPD Composition selection added page for Copper (Cu)

0.0	_				PKS-MPD			_	
Exit		ew perties	Print Materials	Export t Model S	·w VI	lew og	Clear	Instruc- tions	
		Prop	perty Ma	terial Type	Composition	Other C	haracteristics	)	
			Sele	ct an element	from the list:				
			Co	opper (Cu)		:			
			1	Specify percent	age range				
			Pe	ercentage rang	e:				
			Po	ercentage rang . 99.	_	100			Add to Search
election	Criteria:				_		erials Selected	1:	
No.	Field		Min	. 99.	_			Count	Search 440 Remove
No.	Field	pper (Cu)	Min	. 99.	9 Max.			117.0	Search 440 Remove
No.	Field	pper (Cu)	Min	. 99.	9 Max.			Count	Search 440 Remove
No.	Field	pper (Cu)	Min	. 99.	9 Max.			Count	Search 440 Remove
No.	Field	pper (Cu)	Min	. 99.	9 Max.			Count	Search 440 Remove

**FIGURE 2.19** PKS-MPD Composition percentage range selection page for Copper (Cu)

C Exit	)	View Properties	Prir		Export		Viev		Clear	(Marcon Contraction Contractio	
			perty	Materia			Log		haracteristics	tions	
			1	_	fy percer tage ran	ge:		100			Add to Search
elect	ion Criteria	1:						Mate	rials Selected :		3
No.	Field				0	Criteria				Count	Remove
1	Element	Copper (Cu)								440	Ξ
2	Flement	Copper (Cu)	>=99.9	<= 100	%					3	

FIGURE 2.20 PKS-MPD Composition percentage range selection added page for Copper (Cu)

The Selection Criteria window shows that the search yields three possible copper candidate materials. Click the Print Materials button to view the candidate materials and optionally print a data sheet. See Figure 2.21.

The remaining materials candidates are Copper; Copper (UNS C10100); and Copper Alloy, pure copper, UNS C10200.

- 1. Select "Copper (UNS C10100)." See Figure 2.22.
- 2. Double-click the selection to view the properties data for the candidate material(s). See Figure 2.23.
- 3. Click the Accept button (far right, check-marked button).
- 4. Close the selection window.
- 5. Click OK on the Page Setup window.
- 6. Select Print Preview. See Figures 2.24 and 2.25.
- 7. Click either the Cancel button or the Print button.
- 8. Click the Export to Model SW button.

000	PKS-MPD	
elect materials for export or print by highlighting lose the window to continue processing.	. Double-click an item to see detailed data.	
MaterialName :	MaterialChemSymbol :	
Copper	Cu	
Copper (UNS C10100)	Cu+	
Copper Alloy, pure copper, UNS C10200	99.95Cu	

## **FIGURE 2.21** PKS-MPD Composition materials selection page for Copper (Cu)

900	PKS-MPD	
elect materials for export or print by highlighting. Close the window to continue processing.	. Double-click an item to see detailed data.	
MaterialName :	MaterialChemSymbol :	
Copper	Cu	
Copper (UNS C10100)	Cu+	
Copper Alloy, pure copper, UNS C10200	99.95Cu	

**FIGURE 2.23** PKS-MPD Materials selection Properties display page for Copper (UNS C10100)

- 9. Select "Copper (UNS C10100)." See Figure 2.26.
- 10. Double-click "Copper (UNS C10100)" to verify the candidate material selection choice. See Figure 2.27.
- 11. Click the Accept button (far right, check-marked button).
- 12. Close the selection window. See Figure 2.28.
- 13. Click Yes.
- 14. Enter Copper (UNS C10100) in the Material Library Name Request window (to provide a name for the entry in the export log).
- 15. Click OK. See Figure 2.29.

Note In the case where the material property is measured under different conditions (e.g., temperature, frequency), the modeler must choose which value he or she wishes to export for use with the material.

#### Material Properties

6/5/09

Property / Comments	Symbol/T0 Min	Value/T0 Max	UOM / TO UOM
Thermal expansion coeff. (alpha) At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	alpha 293.15	1.7700e-5 573.15	1/K K
Yield Strength At 20-300 degC (68-570 degF) for ceramic to metal scals.	Ys_pks 293.15	6.9000e+7 573.15	Pa K
Tensile Strength (Syt) At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	Syt 293.15	2.2000e+8 573.15	Ра К
Elongation modulus At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	293.15	45.0 573.15	% K
Young's modulus (E) At 20-300 degC (68-570 degF) for ceramic-to-metal seals.	E 293.15	1.1500e+11 573.15	Pa K
Density (rho) Density may depend considerably on previous treatment.	rho	8,960.0 	kg/m^3
Boiling Point	bpT_pks	2,868.15	к
leat Capacity (C)	С	380.0	J/(kg*K)
leat of fusion	lh_pks	2.1185e+5	J/kg
Thermal expansion coeff. (alpha)	alpha	1.6500e-5	1/K
Thermal conductivity (k) +/- 0.005 cal/cm^2/cm/s/K	k	393.9779 	W/(m*K)
Electrical Resistivity (res) At 20 degC (68 degF).	res 293.15	1.6730e-8 293.15	ohm-m K
Electrical Conductivity (sigma) At 20 degC (68 degF). Derived from electrical resistivity.	sigma 293.15	5.9773e+7 293.15	S/m K
Young's modulus (E)	E	1.1032e+11	Ра
Melting Point +/- 0.1 degC.	mpT_pks	1,356.15	к
Specific gravity Derived from density in g/cm^3.		8.96 	NONE
Young's modulus (E) At room temperature 20 degC (68 degF).	E 293.15	1.2800e+11 293.15	Pa K
Shear modulus (Gxy) At room temperature 20 degC (68 degF).	Gxy 293.15	4.6800e+10 293.15	Ра К
Poisson's Ratio (nu) At room temperature 20 degC (68 degF).	nu 293.15	3.0800e-1 293.15	NONE K
Yield Strength At room temperature 20 degC (68 deoF).	Ys_pks 293.15	3.3300e+7 293.15	Pa K

Page 1

**FIGURE 2.24** PKS-MPD Material Properties Print Preview Page 1 for Copper (UNS C10100)

# Material Properties

6/5/09

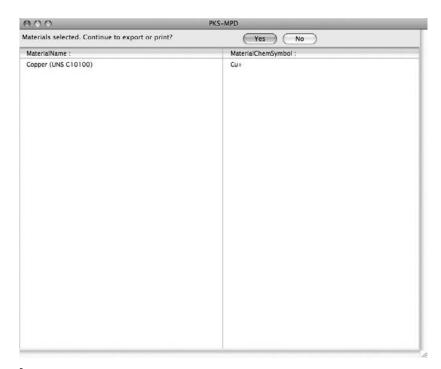
Property / Comments	Symbol/T0 Min	Value/T0 Max	UOM / TO UOM
Tensile Strength (Syt) At room temperature 20 degC (68 degF).	Syt 293.15	2.0900e+8 293.15	5 State
Elongation At room temperature 20 degC (68 degF).	293.15	33.3 293.15	
Element :	Perc	Percent Min: Percent M	
Copper (Cu)		99.99 99.99	
MaterialType :			
Metal			
Metal, Non-Ferrous			
Alloy			

**FIGURE 2.25** PKS-MPD Material Properties Print Preview Page 2 for Copper (UNS C10100)

I FIGURE 2.26 PKS-MPD Materials selection Properties display page for Copper (UNS C10100)

20	PI	KS-MPD		_	
erial 😥 😥	0 🕼	1			
		Flick the Com	position tab for a	2 of 3	
ID : 2502 Name : Copper (UNS C10100	))	chemical com Click the Type	ponents of the m is tab for a list of igned to the mat	aterial. the	
Formula : Cu+ Source: ASM VEM/D09/A06	/S0072383 (T1)	Select So	ource		
UserID : Designer	Last Changed : 6/3/0	9			
UserID : Designer	Last Changed : 6/3/0 Properties Co	9 omposition	Types		-
UserID : Designer			Types		th ()
UserID : Designer Material Name / Comments:			Types Value/T0 Max :	UOM / TO UO	() M:
	Properties Co	TO Min :		1/K	()) M:
Material Name / Comments: Copper (UNS C10100)	Properties Co	TO Min : eff, (alpha)	Value/10 Max : 1.7700e-5	1/K K Pa	<u>м:</u>
Material Name / Comments: Copper (UNS C10100) At 20-300 degC (68-570 degF Copper (UNS C10100)	Properties Co PropertyName Thermal expansion co	TO Min : 293.15 293.15	Value/10 Max : 1.7700e-5 573.15 6.9000e+7	1/K K Pa K Pa	()) M:
Material Name / Comments: Copper (UNS C10100) At 20-300 degC (68-570 degF Copper (UNS C10100) At 20-300 degC (68-570 degF Copper (UNS C10100)	PropertyName Thermal expansion co Yield Strength	T0 Min : eff. (alpha) 293.15 293.15 )	Value/10 Max : 1.7700e-5 573.15 6.9000e+7 573.15 2.2000e+8	1/K K Pa K Pa K %	() M:

# **FIGURE 2.27** PKS-MPD Materials selection Properties display page for Copper (UNS C10100)



**FIGURE 2.29** PKS-MPD Materials selection Tensile Strength (Syt) display page for Copper (UNS C10100)

16. Select the Tensile Strength (Syt) for Copper (UNS C10100) at room temperature.

17. Click Exit.

Continue the selection process for each property as displayed. When finished, click the Exit button for each property. The file is then exported as a text file with library management data leading. Figure 2.30 shows the material properties exported.

The exported file can be directly imported into COMSOL Multiphysics as follows:

- 1. Open COMSOL Multiphysics in the application mode of choice.
- 2. Using the menu bar, select Options > Materials/Coefficients Library.
- 3. Click the Add Library button. See Figure 2.31.
- 4. Select the newly exported Copper (UNS C10110) library. See Figure 2.32.

**FIGURE 2.30** PKS-MPD Materials selection properties for Copper (UNS C10100) exported

000	Materials/Coefficients Library	
Materials Model (0) Basic Material Properties (28) Liquids and Gases (18) MEMS Material Properties (33) Heat Transfer Coefficients (8) Electric (AC/DC) Material Propertie Piezoelectric Material Propertie User Defined Materials (1) FeNi Material Properties (5) Ductile Iron Material Properties Library 10 (8) PKS Test Library (1) PKS Test Library (1)	Material properties	
New Delete Copy Paste Add Library	Hide undefined properties	Functions Plot

**FIGURE 2.31** Materials/Coefficients Library edit page

000	_	Add I	ibrary	
	File:	Copper (UNS C1	0110)	
		A_Library		\$
Name		A	Date Modified	
🗎 Copper (U	NS C10110)		Thursday, Jun	e 4, 2009 4:11 PM
	File Forr	nat: Library D	Data Files (*.da	t; * 🗘

**FIGURE 2.32** Copper (UNS C10100) file selected as the library to be added

- 5. Click OK. The Copper (UNS C10100) (1) library is added as the last item on the Materials library list in the Materials window. See Figure 2.33.
- 6. Click OK.

The second method of Searching if the modeler knows the UNS number, is simply to enter that number.

0.0	Materials/Coefficients Library	
Materials Model (0) Basic Material Properties (28) Liquids and Gases (18) MEMS Material Properties (33) Heat Transfer Coefficients (8) Electric (AC/DC) Material Propertie User Defined Materials (1) FeNi Material Properties (5) Ductile Iron Material Propertie: Library 10 (8) PKS Test Library (1) PKS Test Library (1) Copper (UNS C10100) (1)	Material properties	
New Delete Copy Paste Add Library	Hide undefined properties	Functions

**FIGURE 2.33** Materials/Coefficients Library edit page

**FIGURE 2.34** PKS-MPD Materials selected Other Characteristics page

- 1. Click the Clear button.
- 2. Select "Other Characteristics."
- 3. Enter @UNS C10100@.
- 4. Click the Add to Search button. See Figure 2.34.
- 5. The search yields one candidate material. Click the Print Materials button. See Figure 2.35.

The rest of the instructions for printing, exporting, and adding materials properties to the COMSOL library are the same as given previously.

**FIGURE 2.35** PKS-MPD Materials selected Properties page

# References

- 1. http://www.comsol.com/products/material/
- 2. http://en.wikipedia.org/wiki/DIN
- 3. http://en.wikipedia.org/wiki/Unified\_numbering\_system
- 4. http://en.wikipedia.org/wiki/Oxygen-free\_copper
- 5. http://www.matweb.com/
- 6. http://www.pks-mpd.com/
- 7. Contact Pryor Knowledge Systems, Inc., at http://www.pks-mpd.com for a PKS-MPD sample database and an activation key.

# Exercises

- 1. Explore the processes of finding and exporting materials properties with the COMSOL Material Library module presented in this chapter.
- 2. Explore the processes of finding and exporting materials properties with MatWeb as presented in this chapter.
- 3. Explore the processes of finding and exporting materials properties with PKS-MPD as presented in this chapter.

# **3** 1D Modeling

# In This Chapter

1D Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers
1D Modeling Considerations Coordinate System
1D KdV Equation: Solitons and Optical Fibers COMSOL KdV Equation Model First Variation on the KdV Equation Model Second Variation on the KdV Equation Model
1D KdV Equation Models: Summary and Conclusions
1D Telegraph Equation COMSOL 1D Telegraph Equation Model First Variation on the Telegraph Equation Model
Second Variation on the Telegraph Equation Model
Second Variation on the Telegraph Equation Model
D Telegraph Equation Model
Second Variation on the Telegraph Equation Model
D Telegraph Equation Models: Summary and Conclusions

# ■ 1D Guidelines for New COMSOL® Multiphysics® Modelers

# **1D Modeling Considerations**

1D modeling is both the least difficult and potentially the most difficult type of model to build, irrespective of the modeling software utilized. The least difficult aspect of 1D model building arises from the fact that the geometry is simple: In a 1D model, the modeler can have only a single line or a sequence of line segments as the modeling space. However, the physics in a 1D model can range from reasonably easy (simple) to extremely difficult (complex).

**NOTE** COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software has two 1D modes: 1D (beginning-level through moderate-level modeling) and 1D Axisymmetric (advanced-level modeling). In keeping with the introductory focus of the material in this text, only 1D models (beginning-level through moderate-level models) will be presented. For information on the 1D Axisymmetric geometry, the associated physics, and the use of the same,

refer to the COMSOL manuals, the COMSOL website, and the general COMSOL Multiphysics software-related research literature.

The 1D model implicitly assumes that the energy flow, the materials properties, the environment, and any other conditions and variables that are of interest are homogeneous, isotropic, and constant, unless otherwise specified, throughout the entire domain of interest, both within the model and in the environs of the model. In other words, the properties assigned to the 1D model are representative of the properties of proximate nonmodeled regions. Bearing that in mind, the modeler needs to ensure that all modeling conditions and associated parameters have been properly considered, defined, or set to the appropriate values.

As mentioned earlier, it is always preferable for the modeler to be able to accurately anticipate the expected behavior (results) of the model. Calculated solution values that deviate widely from the expected values or from comparison values measured in experimentally derived realistic models are probably indicative of one or more modeling errors either in the original model design, in the earlier model analysis, or in the understanding of the underlying physics, or are simply due to human error.

## **Coordinate System**

In a 1D model, there are only two coordinates: space (x) and time (t). In a steady-state solution, parameters vary only as a function of space (x). In a transient solution model, parameters can vary both in space (x) and in time (t). The space coordinate (x) typically represents distance throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current).

NOTE To assist the reader to achieve a broader exposure to the applicability of physics discussed in this book and to demonstrate the power of 1D modeling techniques, modeling examples are presented that demonstrate techniques from two different, but similar, broadly applicable areas of physics. The examples presented explore wave propagation, in the broadest general sense.

# 1D KdV Equation: Solitons and Optical Fibers

The KdV equation<sup>1</sup> is a well-known example of a group of nonlinear partial differential equations<sup>2</sup> called exactly solvable.<sup>3</sup> That type of equation has solutions that can be specified with exactness and precision.

Note Nonlinear partial differential equations play an extremely important role in the description of physical systems.<sup>4</sup> Nonlinear partial differential equations are, by and large, inherently difficult to solve and require a unique approach for each equation type.

The KdV equation, solved in 1895 by Diederik Korteweg and Gustav de Vries, mathematically describes the propagation of a surface disturbance on a shallow canal. The effort to solve this wave propagation problem was undertaken based on observations by John Scott Russell in 1834,<sup>5</sup> among others. Subsequent activity in this mathematical area has led to soliton application in magnetics<sup>6</sup> and optics.<sup>7</sup> Work on soliton propagation problems is currently an active area of research.

The following numerical solution model (KdV equation) was originally developed by COMSOL for distribution with the Multiphysics software as an equationbased model. Here, we will build the model as presented in the COMSOL Model Library and then explore variations and expansions on the model.

It is important for the new modeler to personally try to build each model presented within the text. There is no substitute for the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and subsequently correct errors, adding to his or her experience and fund of modeling knowledge. Even the simplest model will expand the modeler's fund of knowledge.

The KdV equation (as written in standard notation) is

$$\partial_t u + \partial_x^3 u + 6u\partial_x u = 0 \tag{3.1}$$

In the COMSOL documentation, the formula is shown as

$$u_t + u_{xxx} = 6uu_x$$
 in  $\Omega = [-8, 8]$  (3.2)

The difference between the two equations is that (3.2) is the negative form of (3.1), which will be adjusted during postprocessing.

The boundary conditions are periodic, as shown here:

$$u(-8, t) = u(8, t)$$
 periodic (3.3)

The initial condition for this model is

$$u(x,0) = -6 \operatorname{sech}^2(x)$$
 (3.4)

Once the modeler builds and solves this model, it will be seen that the pulse immediately divides into two soliton pulses, with different width and propagation speeds.

#### **COMSOL KdV Equation Model**

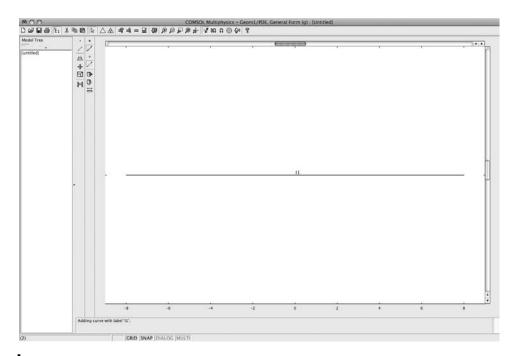
To start building the KdV\_Equation\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "1D" from the Space dimension pull-down list. Select COMSOL Multiphysics > PDE Modes > PDE, General

x limits	y-z limits
x min: -9	Auto
x max: 9	y min: -0.02
	y max: 0.02
	M Auto
	z min: -1
	z max: 1

FIGURE 3.1 1D Axes/Grid Settings window (x)

Form > Time-dependent analysis. Type u1 space u2 in the dependent variables edit field. Click OK. Using the menu bar, select Options > Axes/Grid Settings. Enter -9 tab 9 in the edit fields to define the *x* geometry. Click OK. See Figure 3.1.

Using the menu bar, select Draw > Specify Objects > Line. Enter -8 space 8 in the Line edit window. Click OK. See Figure 3.2.





. C	Source Destination	Source Vertices Destination Vertices
oundary selection	Expression	Constraint name
2		
	-	
	-	
Select by group	Vector element co	nstraint

FIGURE 3.3 Periodic Boundary Conditions window

#### **Periodic Boundary Condition Settings**

For the new modeler unfamiliar with periodic boundary conditions, their use allows the domain (x values) of the model to be extended essentially indefinitely. For example, the modeling workspace of a line has two ends that would form two abrupt terminations, if not somehow compensated for. The use of periodic boundary conditions forms the line into a circle, which is mathematically infinite (endless).

Having established the 1D geometry (line), the next step is to define the fundamental physics conditions. From the menu bar, select Physics > Periodic Conditions > Periodic Boundary Conditions. After the Periodic Boundary Conditions window appears, on the Source page, select "1" in the Boundary selection window. See Figure 3.3.

Enter u1 in the Expression edit window, and then press the Enter key. The constraint name "pconstr1" will appear in the Constraint name column. See Figure 3.4.

	Source	Destination	Source Vertices	Destination Vertice	es
Boundary selection	Expr	ession	Cons	straint name	
1	u1		pcon	strl	
2	-				
	-				
Select by group		ector element co	onstraint		

**FIGURE 3.4** Periodic Boundary Conditions window, Source page

000	Periodi	c Boundary Condi	tions		
Geometry: *Geor Boundary selection	Source Destination	Source Vertices          pconstr1         indaries as destination	Destination Verti	ces	
Select by grou	p				
		Help	Apply	Cancel	ОК

**FIGURE 3.5** Periodic Boundary Conditions window, Destination page, boundary 2

Click the Destination tab. Select "2" as the boundary, and enter u1 in the edit window. See Figure 3.5.

Click the Source Vertices tab. Select "1" as the vertex, and then click the >> button. See Figure 3.6.

Click the Destination Vertices tab. Select "2" as the vertex, and then click the >> button. See Figure 3.7.

Click the Source tab. Select "1" as the boundary, and then type u2 in the second Expression window. Press the Enter key. The label "pconstr2" will appear in the Constraint name column. See Figure 3.8.

Click the Destination tab. Select "2" as the boundary, and then enter u2 in the Expression edit window. See Figure 3.9.

000	_	Periodio	Boundary Cond	itions
	Source	Destination	Source Vertices	Destination Vertices
Constraint name: pco	onstr1			
Vertex selection:			The or relativ detern	rientation of the source domains ely the destination domains is nined by matching source and ation vertices in the order they are
			Help	Apply Cancel OK

**FIGURE 3.6** Periodic Boundary Conditions window, Source Vertices page, vertex 1

	Source	Destination	Source Vertices	Destination Vertices
onstraint name: p	constr1			
ertex selection:		Destinatio	n vertices:	
1		2		rientation of the source domains
2		5		ely the destination domains is nined by matching source and
	( <<		destin	ation vertices in the order they are
		-		•

**FIGURE 3.7** Periodic Boundary Conditions window, Destination Vertices page, vertex 2

oundary selection	Expression	Constraint name
	ul	pconstr1
	u2	pconstr2
Select by group	Vortes element er	
_ select by group	Vector element co	onstraint

**FIGURE 3.8** Periodic Boundary Conditions window, Source page, boundary 1, variable u2

0.0	Periodi	c Boundary Condi	tions		
Geometry: [ *Geom] Roundary selection 1 2 2	Source Destination Constraint name: ( Use selected bou Expression: u2	Source Vertices pconstr2 🛟	Destination Vertic	<u>es</u>	
Select by group					
		Help	Apply	Cancel	ОК

**FIGURE 3.9** Periodic Boundary Conditions window, Destination page

70

000	Periodi	Boundary Cond	itions
	Source Destination	Source Vertices	Destination Vertices
Constraint name: po	constr2		
Vertex selection:	Source ver		
1 2		relativ detern	rientation of the source domains rely the destination domains is nined by matching source and ation vertices in the order they are
		Help	Apply Cancel OK

**FIGURE 3.10** Periodic Boundary Conditions window, Source Vertices page

Click the Source Vertices tab. Select "1" as the vertex, and then click the >> button. See Figure 3.10.

Click the Destination Vertices tab. Select "2" as the vertex, and then click the >> button. See Figure 3.11. Click OK.

#### **Boundary Conditions Settings**

The next step is to set the boundary conditions. Using the menu bar, select Physics > Boundary Settings. Using Table 3.1 as a guide, on the Type page, select boundaries 1 and 2. Click the Neumann button, and then click the G tab. Verify or type 0 in each edit window. Click OK.

		renoure	Boundary Cond	litions
	Source	Destination	Source Vertices	Destination Vertices
Constraint name: pcor	istr2			
Vertex selection:		Destination		
1	>>	2		rientation of the source domains vely the destination domains is
2	C <<		deterr	mined by matching source and nation vertices in the order they are
			added	
		1		
			Help	Apply Cancel OK

**FIGURE 3.11** Periodic Boundary Conditions window, Destination Vertices page

Table 3.1	Boundary Settir	ıgs Window
-----------	-----------------	------------

Parameter	Boundary 1	Boundary 2	G(1)	G(2)
Туре	Neumann	Neumann		
Setting			0	0

Note: Boundary conditions settings specify the values that a solution to the problem being solved needs to take on at the boundary (edge). Two types of boundary conditions are used in this book: Dirichlet and Neumann. In the Dirichlet boundary condition,  $f(a) = n_1$  and  $f(b) = n_2$ , where *a*, *b* are the boundary points and  $n_1$ ,  $n_2$  are given numbers. In the Neumann boundary condition,  $df(a)/dx = n_3$  and  $df(b)/dx = n_4$ , where *a*, *b* are the boundary points and  $n_3$ ,  $n_4$  are given numbers. Mixed boundary conditions, which are a more advanced topic and will not be covered here, are also possible.

#### Subdomain Settings

The next step in building the KdV equation model is to set the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select subdomain 1, and enter the coefficient values under the correct tab as shown in Table 3.2.

Once the PDE coefficients have been entered, Click the Init tab. Enter the initial conditions shown in Table 3.3, and then click OK.

Table 3.2 Subdomain Settings window, PDE coefficients

PDE Coefficient	Value
<i>Γ</i> (1)	u2
Γ(2)	u1x
<i>F</i> (1)	6*u1*u1x
F(2)	u2
<i>d<sub>a</sub></i> (11)	1
<i>d</i> <sub>a</sub> (12)	0
<i>d</i> <sub>a</sub> (21)	0
<i>d</i> <sub>a</sub> (22)	0

Table 3.3	Initial C	conditions	Window
-----------	-----------	------------	--------

Initial Condition	Value
u1(t <sub>o</sub> )	-6*sech(x)^2
u2(t <sub>0</sub> )	-24*sech(x)^2*tanh(x)^2+12*sech(x)^2*(1-tanh(x)^2)

72

Global Sub	domain Boundary	ОК
laximum element size	0.1	Cano
laximum element size scaling factor:	1	App
lement growth rate:	1.3	Hel

**FIGURE 3.12** Free Mesh Parameters window

#### **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters. Type 0.1 in the Maximum element size edit window, as shown in Figure 3.12. The mesh consists of 160 elements. Click OK.

From the menu bar, select Mesh > Initialize Mesh. The result of the meshing operation is shown in Figure 3.13.

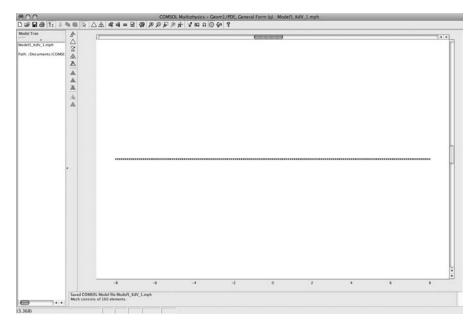


FIGURE 3.13 Meshed model

Analysis:	General	Time Stepping Advanced
Auto select solver	Time stepping Times:	linspace (0, 2, 81)
Stationary Time dependent Eigenvalue	Relative tolerance: Absolute tolerance:	0.01
Parametric Stationary segregated	Allow complex numbers	
Parametric segregated	Preconditioner:	ect (UMFPACK)
	Matrix symmetry: Au	iomatic

**FIGURE 3.14** Solver Parameters window

#### Solving the KdV Equation Model

First, using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver. Type linspace(0, 2, 81) in the Times edit window, as shown in Figure 3.14. This instruction divides the time-space into 81 equal intervals, in the period from 0 to 2 seconds.

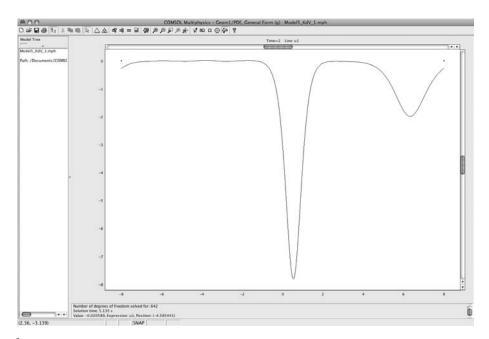
When the instruction linspace(a, b, c) is typed, it must be typed with *no* space between the last "e" of "linspace" and the open parenthesis (of the argument specification (a, b, c). If it is not typed exactly this way, the COMSOL Multiphysics software will indicate an error!

Click the Time Stepping tab. Type 2 in the Maximum BDF order edit window, as shown in Figure 3.15. Click OK.

Using the menu bar, select Solve > Solve Problem. The solution that is immediately seen is the negated (–) solution at the last time interval (t = 2 seconds), as shown in Figure 3.16.

	Solver	Parameter	s		_
analysis:		General	Time Step	ping Advanced	
Auto select solver	General Times to store in ou	itput:	Specifi	ied times	•
Stationary	Time steps taken by	solver.	Free		\$
Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated	Stop condition.	lion			1
ratametri, segi egareu	Manual tuning of step size				
	Initial time step: 0.0010 Maximum time step: 1.0				
Adaptive mesh refinement	Advanced				
	Maximum BDF orde	Maximum BDF order:		2	
	Minimum BDF order:		1		
	Singular mass matri	ix:		Maybe	\$
	Consistent initializa	tion of DAE	systems:	Backward Euler	\$
	Error estimation str	ategy:		Include algebraic	:

**FIGURE 3.15** Solver Parameters window, Time Stepping page



**FIGURE 3.16** Negated KdV model solution

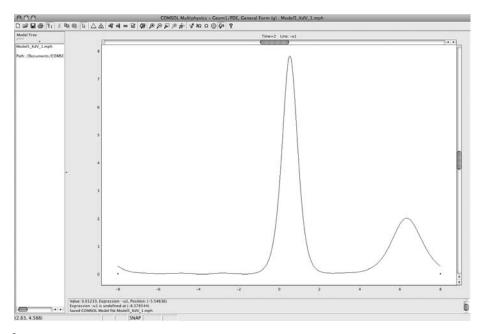


FIGURE 3.17 KdV equation model solution

#### Postprocessing

The positive solution can be viewed as follows: Select Postprocessing > Plot Parameters. Click the Line tab. Type -u1 in the Expression edit window. Click the Apply button. The positive results are shown in Figure 3.17.

Save the KdV Equation model as Model3\_KdV\_1.

The solution to the KdV equation can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears (see Figure 3.18), click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.19). Click the Start Animation button. Save the KdV equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3\_KdV\_1.avi that was supplied with this book.

Note Many modelers are better able to understand the dynamics of the solution when the solution is presented as an animation. It is available in addition to the presentation of the solution as a series of static plots.

The file extension that is created during the Save operation is platform dependent. If the platform is a Power Mac<sup>®</sup> computer, the extension for an animation will be different (.mov) than that for a Mac<sup>®</sup> computer with Intel<sup>®</sup> processor or a PC (.avi). Either can be played using a free QuickTime<sup>®</sup> player (http://www.apple.com/quicktime).

76

0.0	F	Plot Parameter	'S	
(	General	Line Max/Min	n Animate	
🗹 Line plot				
🗹 Height data				
Predefined quantities:	[		÷	
Expression:	-u1		Smooth	
Unit:			•	
Line color				
O Use expression to	color lines	Colo	r Expression	
<ul> <li>Uniform color</li> </ul>		Color		
	(	Help	Apply Cancel	ОСок

**FIGURE 3.18** KdV model solution Plot Parameters window, Line page

	PIOL P	arameters	_
	General Line	Max/Min Animate	
Movie settings		Solutions to use	
File type:	AVI 🛟	Select via: Stored output tim	ies 🗘
Width (in pixels):	640	0	0
Height (in pixels):	480	0.025 0.05	U.
Frames per second:	10	0.075	
	Advanced	0.1 0.125	
		0.15 0.175	
Static / Eigenfunction		0.2 0.225	A V
Cycle type:	Full harmonic	Times:	
Number of frames:	11		
Reverse direction			
		(Start A	nimatior
	He	Ip Apply Cancel	$\square$

**FIGURE 3.19** KdV model solution, Animate page

# **First Variation on the KdV Equation Model**

The previous solution to the KdV equation results in two soliton pulses propagating in the same medium at the same time. Next, we will explore how the model behaves when the initial conditions are modified. In this case, the argument is made smaller.

Information transmission relies on the measurement of a difference. In Morse code (a time differentiation method), the difference is between a long pulse, a short pulse, and no pulse. No pulse signifies no message. Thus, even if a message was sent, if it was not received (detected), then the recipient of the non-message classifies the message traffic as zero. To receive a message, the received signal must be of adequate amplitude (analog), of adequate duration (time), and in the expected frequency band of the receiver. The signal amplitude must be sufficiently greater than the detection threshold to allow information to be collected. The signal-to-noise ratio determines the minimum detectable signal.<sup>8</sup>

The stable, long-distance, light pulses used to convey information through optical fibers are known as temporal solitons.<sup>9</sup> To achieve detectability, the fiber is designed to compensate for dispersion (frequency spreading) and power loss.

First, save a copy of the just-created KdV equation model as Model3\_KdV\_2. You can then modify the KdV equation model without being concerned about damaging the original model.

If Model3\_KdV\_2 is already open on your desktop, skip to the "Scalar Expressions" section. If Model3\_KdV\_2 is not already open on your desktop, using the menu bar, select File > Open. When the Open Model window is displayed as in Figure 3.20, select "Model3\_KdV\_2." Click OK.

#### Scalar Expressions

Using the menu bar, select Options > Expressions > Scalar Expressions. When the Scalar Expressions window opens, type  $x_a$  in the Name column and x/1.33 in the Expression column, as shown in Figure 3.21. Click OK.

NOTE The scalar expression that was just created will be used as the new argument for the initial conditions of the KdV equation model.

Having created the new scalar variable x\_a, the next step is to modify the Initial Conditions expression(s).

#### **Changing the Subdomain Settings**

The next step in modifying the KdV equation model is to change the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window

00	M	odel Navigator	
	New Model Library	User Models Open	Settings
	5_Mode	ls	
		Name 🔺	Date Modified
		Model5_KdV_1	Tuesday, July 1, 200
10		Model5_KdV_2 Model5_KdV_3	Wednesday, July 2, 2 Wednesday, July 2, 2
Description:			
	File Format: CO	MSOL Multiphysics Mod	

FIGURE 3.20 Open Model window

appears, select subdomain 1. Verify that the coefficient value under each of the indicated tabs is as shown in Table 3.4.

Once the PDE coefficients have been verified, click the Init tab. Either modify the existing equations or enter the initial conditions shown in Table 3.5, and then click OK.

0.00	Sc	alar Expression	ns
Name	Expression	Unit	Description
x_a	x/1.33	m	
			¥
-			) 4   4

**FIGURE 3.21** Scalar Expressions window

PDE Coefficient	Value
<i>Γ</i> (1)	u2
<i>Γ</i> (2)	u1x
<i>F</i> (1)	6*u1*u1x
F(2)	u2
<i>d</i> <sub>a</sub> (11)	1
<i>d</i> <sub>a</sub> (12)	0
<i>d</i> <sub>a</sub> (21)	0
<i>d</i> <sub>a</sub> (22)	0

 Table 3.4
 Subdomain Settings Window, PDE Coefficients

Because the new KdV equation model is a revised copy of the original KdV equation model, the new model will need to be reset. Using the menu bar, select File > Reset Model.

**NOTE** The Reset Model command clears the copied model of previous meshes and solutions.

#### **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters. Verify or type 0.1 in the Maximum element size edit window, as shown in Figure 3.22. Click OK.

From the menu bar, select Mesh > Initialize Mesh. The result of the meshing operation is shown in Figure 3.23.

#### Solving the First Revised KdV Equation Model

Using the menu bar, select Solve > Solver Parameters. Verify that the COMSOL Multiphysics software automatically selected the Time dependent solver. Verify or type linspace(0, 2, 81) in the Times edit window, as shown in Figure 3.24. This instruction divides the time-space into 81 equal intervals, in the period from 0 to 2 seconds.

Initial Condition	Value
u1(t <sub>o</sub> )	-6*sech(x_a)^2
u2(t <sub>0</sub> )	-24*sech(x_a)^2*tanh(x_a)^2+12*sech(x_a)^2*
	(1-tanh(x_a)^2)

#### Table 3.5 Initial Conditions Window

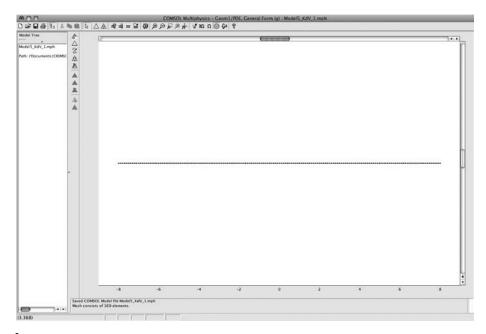
0.0	Free Mesh Parameters	
	Global Subdomain Boundary	ОК
Maximum element si	ze: 0.1	Cancel
Maximum element si	ze scaling factor: 1	Apply
Element growth rate	1.3	Help
		1
		1
Reset to Defaults	Remesh Mesh Selected	

**FIGURE 3.22** Free Mesh Parameters window

Click the Time Stepping tab. Verify or type 2 in the Maximum BDF order edit window, as shown in Figure 3.25. Click OK.

Using the menu bar, select Solve > Solve Problem.

In this variation, the solution that is immediately seen is not the negated (-) solution at the last time interval. Instead, the solution shown is the positive solution





900	Solve	er Paramet	ers		_
Analysis:		General	Time Stepping	Advanced	
Auto select solver Solver: Stationary Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated Adaptive mesh refinement	Time stepping Times: Relative tolerance Absolute tolerance Allow comple Linear system so Linear system so Preconditioner:	ce: ex numbers lver	0.0	ipace (0, 2, 81) 1 010 010 010 010	
	Matrix symmetry	Aut	omatic	(	Settings
		C	Help Ap	ply Cance	ОК

**FIGURE 3.24** Solver Parameters window

	Solver Paramete	rs		_	
Analysis:	General	Time Step	ping Advanced	]	
Auto select solver	General Times to store in output:	Specifi	ed times		
Stationary	Time steps taken by solver:	Free		•	
Time dependent Eigenvalue Parametric Stationary segregated	Use stop condition Stop condition:				
Parametric segregated	Manual tuning of step size	0.0010			
	Maximum time step:	1.0			
Adaptive mesh refinement	Advanced				
	Maximum BDF order:		2		
	Minimum BDF order:		1		
	Singular mass matrix: Consistent initialization of DAE systems:		Maybe	\$	
			Backward Euler	\$	
	Error estimation strategy:		Include algebrai	c 🛟	

**FIGURE 3.25** Solver Parameters window, Time Stepping page

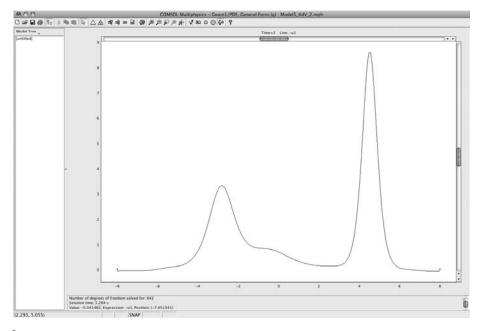


FIGURE 3.26 First revised KdV model solution

of the KdV Equation, because the sign inversion (–) was adjusted in postprocessing of the copied model. The results of the changed argument solution are shown in Figure 3.26.

Save this KdV Equation model as Model3\_KdV\_2 to retain the current solution.

#### **Postprocessing Animation**

This solution to the KdV equation can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.27). Click the Start Animation button. Save this KdV equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3\_KdV\_2.avi that was supplied with this book.

The reduction of the argument for the initial conditions results in the splitting of the initial, single soliton pulse into three separate soliton pulses that propagate through the medium (e.g., optical fiber) at three different velocities and arrive at the receiver at different times.

Adoption of either the first solution or the second solution in an information transmission system would cause serious message distortion or interference problems

	General Line	Max/Min Animate	
Movie settings		Solutions to use	
File type:	AVI 🛟	Select via: Stored	output times 🛟
Width (in pixels):	640	0	2
Height (in pixels):	480	0.025 0.05	U
Frames per second:	10	0.075	
	Advanced	0.125	
Static / Eigenfunction	animation	0.15 0.175	
Cycle type:	Full harmonic	0.2 0.225	
	11	Times:	
Reverse direction			
			Start Animatio

**FIGURE 3.27** KdV model solution, Animate page

at the receiving site. These solutions would cause the same nature of interference as multiple-path propagation in atmospheric transmission (e.g., the same signal arriving several times at the same receiver in a slightly delayed mode).

# **Second Variation on the KdV Equation Model**

The first revised solution to the KdV equation results in three soliton pulses propagating in the same medium at the same time. Next, we will explore how the model behaves when the initial conditions are again modified. In this case, the argument of the initial conditions will be increased in size.

**Note** Remember—information transmission relies on the measurement of a difference. Each pulse is one bit of information. No pulse signifies no message. Thus, even if a message was sent, if it was not received (detected), then the recipient of the non-message classifies the message traffic as zero.

To receive the correct message, the signal must be of adequate amplitude (analog), of adequate duration (time), in the expected frequency band of the receiver, and must correlate exactly with the message sent. The signal amplitude must be sufficiently greater than the detection threshold to allow information to be collected and must not contain spurious, random pulses.

84

000	Model Navigator
	New Model Library User Models Open Settings
	5_Models
Description: Preview	Name       Date Modified         Model5_KdV_1       Tuesday, July 1, 200         Model5_KdV_2       Wednesday, July 2, 2         Model5_KdV_3       Wednesday, July 2, 2
-	File Format: COMSOL Multiphysics Mod 🗘

FIGURE 3.28 Open Model window

First, save a copy of the just-created first variation on the KdV equation model as Model3\_KdV\_3. You can then modify the KdV equation model without being concerned about damaging the just-built model.

If Model3\_KdV\_3 is already open on your desktop, skip to the "Scalar Expressions" section. If Model3\_KdV\_3 is not already open on your desktop, using the menu bar, select File > Open. When the Open Model window is displayed as in Figure 3.28, select "Model3\_KdV\_3." Click OK.

#### Scalar Expressions

Using the menu bar, select Options > Expressions > Scalar Expressions. When the Scalar Expressions window opens, verify or type  $x_a$  in the Name column and pi/2\*x in the Expression column, as shown in Figure 3.29. Click OK.

NOTE The scalar expression that was just created will be used as the new larger argument for the initial conditions of the second variation on the KdV equation model.

Having created the new Scalar Variable x\_a, the next step is to modify the Initial Conditions expression(s).

Name	Expression	Unit	Description
x_a	pi/2*x	m	
c	Hel	p Apply	Cancel OK

FIGURE 3.29 KdV Scalar Expressions window

## **Changing the Subdomain Settings**

The next step in modifying this version of the KdV equation model is to change the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select subdomain 1. Verify that the coefficient value assigned to each of the indicated tabs is as shown in Table 3.6.

Once the PDE coefficients have been verified, click the Init tab. Verify the existing equations or type the initial conditions found in Table 3.7, and then click OK.

Table 3.6	Subdomain Settings Window, PDE Coefficients
-----------	---

PDE Coefficient	Value
<i>Γ</i> (1)	u2
<i>Γ</i> (2)	u1x
<i>F</i> (1)	6*u1*u1x
F(2)	u2
<i>d</i> <sub>a</sub> (11)	1
<i>d</i> <sub>a</sub> (12)	0
<i>d</i> <sub>a</sub> (21)	0
<i>d</i> <sub>a</sub> (22)	0

#### Table 3.7 Initial Conditions Window

Initial Condition	Value
u1(t <sub>o</sub> )	-6*sech(x_a)^2
u2(t <sub>0</sub> )	-24*sech(x_a)^2*tanh(x_a)^2+12*sech(x_a)^2* (1-tanh(x_a)^2)

0.0	Fr	ee Mesh Parameters	
	Global Subo	Iomain Boundary	ОК
Maximum eleme	ent size:	0.1	Cance
Maximum eleme	ent size scaling factor:	1	Apply
Element growth	rate:	1.3	Help
Reset to Default	s Remesh	Mesh Selected	

**FIGURE 3.30** Free Mesh Parameters window

Because the new KdV equation model is a revised copy of the original KdV equation model, this model will need to be reset. Using the menu bar, select File > Reset Model.

**NOTE** The Reset Model command clears the copied model of previous meshes and solutions.

#### **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters. Verify or type 0.1 in the Maximum element size edit window, as shown in Figure 3.30. Click OK.

From the menu bar, select Mesh > Initialize Mesh. The result of the meshing operation is shown in Figure 3.31.

#### Solving the Second Revised KdV Equation Model

Using the menu bar, select Solve > Solver Parameters. Verify that the COMSOL Multiphysics software automatically selected the Time dependent solver. Verify or type linspace(0, 2, 81) in the Times edit window, as shown in Figure 3.32. This instruction divides the time-space into 81 equal intervals, in the period from 0 to 2 seconds.

Click the Time Stepping tab. Verify or type 2 in the Maximum BDF order edit window, as shown in Figure 3.33. Click OK.

Using the menu bar, select Solve > Solve Problem.

The solution that is immediately seen is not the negated (–) solution at the last time interval. Instead, the solution shown is the positive solution of the KdV equation,

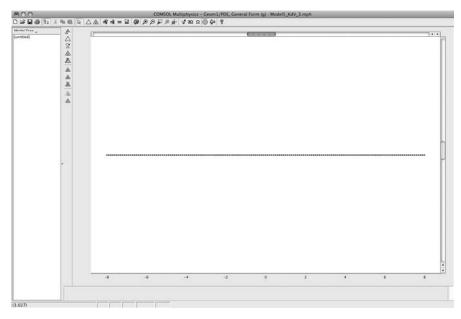


FIGURE 3.31 Remeshed model

000	Solver Para	ameters	
Nnalysis:	Gene	eral Lime Stepping	Advanced
Auto select solver solver: Stationary	Time stepping Times: Relative tolerance.	linsp 0.01	ace(0, 2, 81)
Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated	Absolute tolerance:	0.00	10
	Linear system solver Linear system solver:	Direct (UMFPACK)	Settings)
	Matrix symmetry:	Automatic	
		(Help) (Appl	y Cancel OK

FIGURE 3.32 Solver Parameters window

000	Solver Para	umeters		
Analysis:	Gene	eral Time Step	ping Advanced	-
Auto select solver Solver.	General Times to store in output	t: Specifi	ied times	•
Stationary	Time steps taken by sol	ver: Free		\$
Time dependent Eigenvalue Parametric Stationary segregated	Stop condition			
Parametric segregated	Manual tuning of st	ep size		
	Initial time step: 0.0010 Maximum time step: 1.0			
Adaptive mesh refinement	Advanced			
	Maximum BDF order:		2	
	Minimum BDF order:		1	-
	Singular mass matrix:		Maybe	\$
	Consistent initialization	of DAE systems:	Backward Euler	\$
	Error estimation strateg	IV:	Include algebraic	\$

FIGURE 3.33 Solver Parameters window, Time Stepping page

because the sign inversion (–) was adjusted in postprocessing of the previous model. The results of the changed argument solution are as shown in Figure 3.34.

Save this KdV equation model as Model3\_KdV\_3 to retain the current solution.

#### **Postprocessing Animation**

This solution to the KdV equation can also be viewed as an animation. To view this solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.35). Click the Start Animation button. Save this KdV equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3\_KdV\_3.avi that was supplied with this book.

**NOTE** The result of the argument change for the second variation on the KdV equation model initial conditions is the generation of a single soliton that propagates through the medium (e.g., optical fiber) at one velocity. This soliton pulse will reliably convey information to the receiving station.

One factor that this model does not address is the loss of energy (attenuation) as a function of distance. It is a more advanced topic that will not be covered in this book.

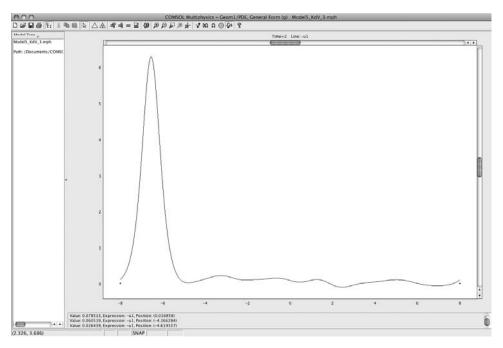
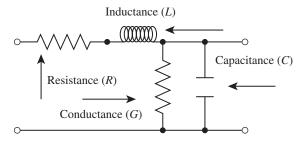


FIGURE 3.34 Second variation on the KdV equation model solution

0.0	Plot Para	imeters	-
	General Line I	fax/Min Animate	
Movie settings		Solutions to use	
File type:	AVI 🗘	Select via: Stored output tin	nes 🗘
Width (in pixels):	640	0	
Height (in pixels):	480	0.025 0.05	
Frames per second:	10	0.075 0.1	
	(Advanced)	0.125	
Static / Eigenfunction	animation	0.15 0.175	
Cycle type:	Full harmonic	0.2 0.225	
Number of frames:	11	Times:	
Reverse direction			
		Start A	nimatio
	Help	Apply Cancel	C

I FIGURE 3.35 KdV model solution, Animate page



**FIGURE 3.36** Telegraph equation electrical component model

#### **1D KdV Equation Models: Summary and Conclusions**

The KdV equation is a powerful tool that can be used to model soliton wave propagation in diverse media (e.g., physical waves in liquids, electromagnetic waves in transparent media). It is easily and simply modeled with a 1D PDE mode model.

# 1D Telegraph Equation

The telegraph equation<sup>10</sup> was developed by Oliver Heaviside<sup>11</sup> and first published about 1885.<sup>12</sup> The telegraph equation is based on a lumped constant, four-terminal electrical component model, as shown in Figure 3.36.

In this schematic model of the telegraph wires (and other transmission lines), there are four fundamental components: resistance (R) per unit of length (e.g., foot, meter), inductance (L) per unit of length (e.g., foot, meter), conductance (G) per unit of length (e.g., foot, meter), and capacitance (C) per unit of length (e.g., foot, meter). The differential equations for voltage (V) and current (I) have the same form, as shown in equations 3.5 and 3.6.

Equation 3.5 shows the partial differential equation for voltage (*V*):

$$\frac{\partial^2}{\partial x^2}V = LC\frac{\partial^2}{\partial t^2}V + (RC + GL)\frac{\partial}{\partial t}V + GRV$$
(3.5)

Equation 3.6 shows the partial differential equation for current (*I*):

$$\frac{\partial^2}{\partial x^2}I = LC\frac{\partial^2}{\partial t^2}I + (RC + GL)\frac{\partial}{\partial t}I + GRI$$
(3.6)

Equations 3.5 and 3.6 are similar in form to equation 3.7, as shown here for the COMSOL Multiphysics telegraph equation model:

$$u_{tt} + (\alpha + \beta)u_t + \alpha\beta u = c^2 u_{xx}$$
(3.7)

where  $\alpha$  and  $\beta$  are positive constants, *c* is the transport velocity, and *u* is the voltage.

Equation 3.5 can be restated in subscript notation:

$$u_{xx} = LC u_{tt} + (RC + GL)u_t + GRu$$
(3.8)

Rearranging the terms of equation 3.7 gives the following equation:

$$u_{xx} = \frac{1}{c^2} u_{tt} + \frac{1}{c^2} (\alpha + \beta) u_t + \frac{1}{c^2} \alpha \beta u$$
(3.9)

Comparing equations 3.8 and 3.9 yields

$$LC = \frac{1}{c^2} \tag{3.10}$$

and

$$\alpha + \beta = \frac{(RC + GL)}{LC}$$
(3.11)

and

$$\alpha\beta = \frac{GR}{LC} \tag{3.12}$$

Solving for  $\alpha$  and  $\beta$ :

$$\alpha = \frac{CGL + C^2R - \sqrt{-4CGLR + (-CGL - C^2R)^2}}{2L}$$

$$\beta = \frac{CGL + C^2R + \sqrt{-4CGLR + (-CGL - C^2R)^2}}{2L}$$
(3.13)

or

$$\alpha = \frac{CGL + C^{2}R + \sqrt{-4CGLR + (-CGL - C^{2}R)^{2}}}{2L}$$

$$\beta = \frac{CGL + C^2R - \sqrt{-4CGLR + (-CGL - C^2R)^2}}{2L}$$

In the event that

$$R = G = 0 \tag{3.14}$$

the transmission line is considered lossless and the telegraph equation becomes

$$u_{xx} = LC u_{tt} \tag{3.15}$$

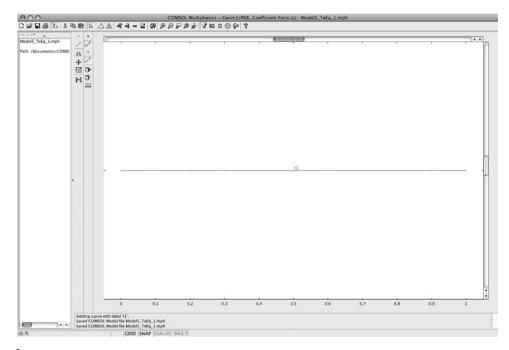
# **COMSOL 1D Telegraph Equation Model**

#### **Model Navigator**

To start building the telegraph equation model, activate the COMSOL Multiphysics software. In the Model Navigator, select "1D" from the Space dimension pull-down list. Select COMSOL Multiphysics > PDE Modes > PDE, Coefficient Form > Time-dependent analysis, wave type. Verify that Lagrange-Quadratic elements have been selected in the Element pull-down list. Click OK.

#### **1D Geometry**

Once the COMSOL Multiphysics 1D workspace window has appeared, using the menu bar, select Draw > Specify Objects > Line. Type 0 space 1 in the Coordinates edit window of the Line window. Click OK. Using the toolbar, click the Zoom Extents button. The 1D geometry will appear as shown in Figure 3.37.



**FIGURE 3.37** Telegraph equation geometry

## Table 3.8 Constants Window

Expression
1
0.25
0.25

# Table 3.9 Boundary Settings Window

Parameter	Boundary 1	Boundary 2	q	g
Туре	Neumann	Neumann		
Setting			0	0

# Constants

Using the menu bar, select Options > Constants. Type the constants in the Constants edit window, as indicated in Table 3.8, and then click OK.

# **Boundary Conditions**

Using the menu bar, select Physics > Boundary Settings. After the Boundary Settings window appears, select both boundaries 1 and 2. Enter or verify the settings indicated in Table 3.9, and then click OK.

### Subdomain Settings

The next step in building the telegraph equation model is to set the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select subdomain 1 and enter the coefficient values under the correct tab as shown in Table 3.10. Verify and then leave the other coefficient settings at their 0 value, as shown in Figure 3.38.

### Table 3.10 Subdomain Settings Window, PDE Coefficients

PDE Coefficient	Value
С	C*C
а	alpha*beta
f	–(alpha+beta)*ut
e <sub>a</sub>	1
d <sub>a</sub>	0

$a^{2}u/\partial t^{2} + d_{a}\partial u/\partial t + \nabla \cdot (-c\nabla$			ement Weak Color/Style
Subdomain selection	PDE coefficier		ement weak color/style
1 (default)	Coefficient	Value/Expression	Description
	c	c*c	Diffusion coefficient
	a	alpha*beta	Absorption coefficient
	f	-(alpha+beta)*ut	Source term
	e <sub>a</sub>	1	Mass coefficient
	d <sub>a</sub>	0	Damping/Mass coefficient
Group: default	α	0	Conservative flux convection coeff
Select by group	β	0	Convection coefficient
Active in this domain	Y	0	Conservative flux source term

**FIGURE 3.38** PDE window, Coefficients page

Once the PDE coefficients have been entered, click the Init tab. Enter the initial conditions found in Table 3.11, and then click OK. See Figure 3.39.

### **Mesh Generation**

Using the toolbar, select Initialize Mesh > Refine Mesh once. The final mesh, with 30 elements, is shown in Figure 3.40.

# Solving the Telegraph Equation Model

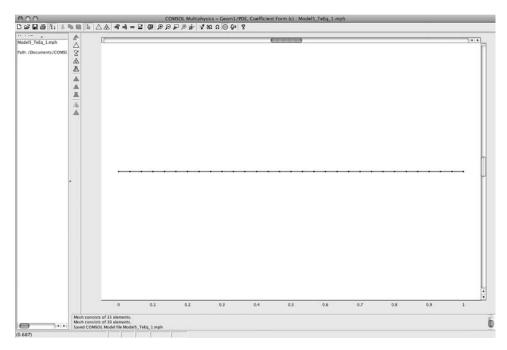
Using the menu bar, select Solve > Solver Parameters. Once the Solver Parameters window appears, click the Time Stepping tab. Place a check mark in the Manual tuning of step size check box. Type 0.002 in the Initial time step edit field, as shown in Figure 3.41. Click OK.

Table 3.11	Initial Cond	ditions Windo	w
------------	--------------	---------------	---

Value
exp(-3*(x/0.2-1)^2)
0

Subdomains Groups	$c\nabla u - \alpha u + \gamma) + au + \beta \cdot \nabla u = f$
bdomain selection	Initial value u(t <sub>0</sub> ) kxp(-3*(x/0.2-1)^2)
	$ut(t_0)$ 0
oup: default	
Active in this domain	

**FIGURE 3.39** PDE window, Init page



**FIGURE 3.40** PDE, telegraph equation model mesh

Analysis: Ceneral Time Stepping Advanced Ceneral Times to store in output: Specified times Solver: Solver: Stationary Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated Parametric segregated Parametric segregated Parametric segregated Parametric segregated Adaptive mesh refinement Adaptive mesh refinement Adaptive mesh refinement Adaptive mesh refinement Adaptive mesh refinement Ceneral Times to store in output: Specified times Use stop condition Stop condition Stop condition: Manual tuning of step size Initial time step: 0.002 Maximum time step: 1.0 Advanced Maximum BDF order: 2 Minimum BDF order: 1	000	Solver Parameters
✓ Auto select solver         Solver:         Stationary         Time dependent         Eigenvalue         Parametric         Stationary segregated         Parametric segregated         Adaptive mesh refinement         Adaptive mesh refinement         Adaptive mesh refinement	3	General Time Stepping Advanced
Time dependent         Eigenvalue         Parametric         Stationary segregated         Parametric segregated         Image: Manual tuning of step size         Initial time step:       0.002         Maximum time step:       1.0         Adaptive mesh refinement       Advanced         Maximum BDF order:       2	Auto select solver	Times to store in output: Specified times
Adaptive mesh refinement Adaptive mesh refinement Adaptive mesh refinement Adaptive mesh refinement Advanced Maximum BDF order: 2	Fime dependent Eigenvalue Parametric Stationary segregated	Use stop condition
Advanced Maximum BDF order: 2	arametric segregateo	Initial time step: 0.002
	Adaptive mesh refinement	Maximum BDF order: 2
Singular mass matrix: Consistent initialization of DAE systems: Error estimation strategy: Include algebraic +		Singular mass matrix: Maybe 🗘

**FIGURE 3.41** Solver Parameters window, Time Stepping page

NOTE The 0.002 time step is selected, in this case, to yield adequate solution resolution without requiring extensive resource consumption (computer/modeler time).

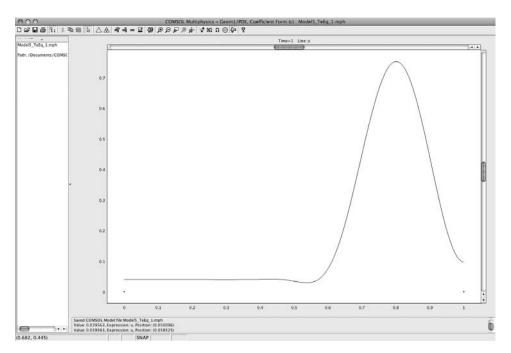
Using the menu bar, select Solve > Solve Problem. The solution for the final time interval is as shown in Figure 3.42.

#### Postprocessing

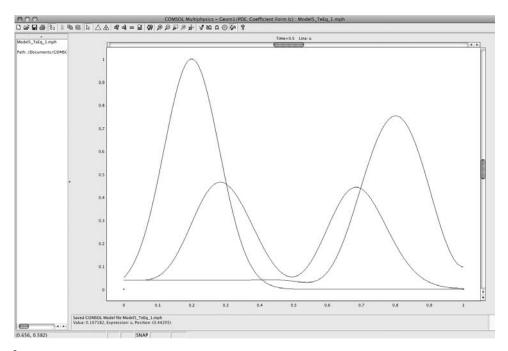
Using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the General tab. Place a check mark in the Keep current plot check box. Select "Solution at time: 0." Click the Apply button. Select "Solution at time: 0.5." Click the Apply button, and then click OK. Figure 3.43 shows the resulting plot of the pulse amplitude as it propagates from left to right.

#### **Postprocessing Animation**

This solution to the telegraph equation can also be viewed as an animation. To view this solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page,



**FIGURE 3.42** Telegraph equation model solution



**FIGURE 3.43** Telegraph equation pulse amplitude plot

0	Plot Pa	rameters	
	General Line	Max/Min Anima	ate
Movie settings		Solutions to us	e
File type:	AVI	Select via:	Stored output times 💲
Width (in pixels):	640	0	0
Height (in pixels):	480	0.1 0.2	
Frames per second:	10	0.3 0.4	
	(Advanced)	0.5	
Static / Eigenfunction	animation	0.6 0.7	
Cycle type:	Full harmonic	0.8 0.9	Y
Number of frames:	11	Times:	
Reverse direction		1	
_			
			Start Animation
	Help	Apply	Cancel 0

**FIGURE 3.44** Telegraph Equation Plot Parameters window, Animate page

select all the solutions in the Stored output times window (see Figure 3.44). Click the Start Animation button. Save this telegraph equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3\_TE\_1.avi that was supplied with this book.

Select File > Save as. Type Model3\_TeEq\_1 in the Save As edit window.

#### First Variation on the Telegraph Equation Model

The previous solution to the telegraph equation shows a pulse propagating from left to right. Let us now explore how the model behaves when the initial conditions are modified. In this case, the argument is made smaller, reflecting the behavior of a lower-loss transmission line.

Note Information transmission relies on the measurement of differences, as stated earlier. To receive a message, the signal must be of detectable amplitude (analog), of detectable duration (time), and in the pass-band (correct frequency or frequency spread) of the receiver. The signal amplitude must be sufficiently greater than the detection threshold and above the noise level (on the average) to allow information to be collected.

First, save a new copy of the just-created telegraph equation model Model3\_ TeEq\_1 as Model3\_TeEq\_2. You can then modify the telegraph equation model without being concerned about damaging the original model.

#### Table 3.12 Constants Window

Name	Expression
С	1
alpha	0.005
beta	0.005

Name	Expression	Value	Description
	1	1	
lpha	0.005	0.005	
oeta	0.005	0.005	
			-
			) 4 • (

FIGURE 3.45 Constants window

Using the menu bar, select Options > Constants. After the Constants window appears, type the expressions indicated in Table 3.12 (also see Figure 3.45), and then click OK.

## **Boundary Conditions**

Using the menu bar, select Physics > Boundary Settings. After the Boundary Settings window appears, select both boundaries 1 and 2. Enter or verify the settings indicated in Table 3.13 (also see Figure 3.46). Click OK.

### Subdomain Settings

The next step in building the revised telegraph equation model is to set the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window

Table 3.13 Boundary Settings Window

Parameter	Boundary 1	Boundary 2	q	g
Туре	Neumann	Neumann		
Setting			0	0

Equation n·(c∇u + αu - γ) + qu = g		
Boundaries Group	)5 ]	Coefficients Weak Color
Boundary selection	Neun     Dirici	conditions nann boundary condition hlet boundary condition ent Value/Expression
Group:	g h r	
Interior boundaries		

**FIGURE 3.46** Boundary Settings window

appears, select subdomain 1. Enter or verify the coefficient values under the correct tab as shown in Table 3.14. Leave the other coefficient settings at their 0 value, as shown in Figure 3.47.

Once the PDE coefficients have been entered or verified, click the Init tab. Type or verify the initial conditions found in Table 3.15 in the edit windows, as shown in Figure 3.48. Click OK.

#### **Model Reset**

Select File > Reset Model > Yes.

# Table 3.14 Subdomain Settings Window, PDE Coefficients

PDE Coefficient	Value
С	C*C
а	alpha*beta
f	–(alpha+beta)*ut
e <sub>a</sub>	1
d <sub>a</sub>	0

Subdomains Groups	600	efficients Init Ele	ement Weak Color/Style
ubdomain selection	PDE coefficier		enent weak colorjatije
1 (default)	Coefficient	Value/Expression	Description
	c	¢*¢	Diffusion coefficient
	a	alpha*beta	Absorption coefficient
	f	-(alpha+beta)*ut	Source term
	e <sub>a</sub>	1	Mass coefficient
	da	0	Damping/Mass coefficient
iroup: default	α	0	Conservative flux convection coeff
	β	0	Convection coefficient
Select by group	Y	0	Conservative flux source term

**FIGURE 3.47** PDE, Subdomain Settings window, PDE Coefficients

#### Table 3.15 Initial Conditions Window

Initial Condition	Value
u1(t <sub>o</sub> )	exp(-3*(x/0.2-1)^2)
u2(t <sub>0</sub> )	0

### **Mesh Generation**

Using the toolbar, select Initialize Mesh > Refine Mesh once. The final 30-element mesh is shown in Figure 3.49.

### Solving the Telegraph Equation Model

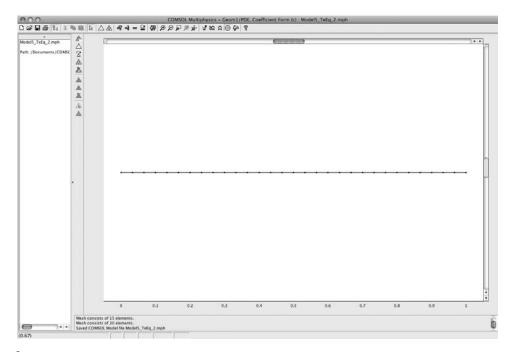
Using the menu bar, select Solve > Solver Parameters. Once the Solver Parameters window appears, click the Time Stepping tab. Place a check mark in the Manual tuning of step size check box. Type 0.002 in the Initial time step edit field, as shown in Figure 3.50. Click OK.

NOTE As mentioned in an earlier note, the 0.002 time step is selected to yield adequate solution resolution without requiring extensive resource consumption (computer/modeler time).

Using the menu bar, select Solve > Solve Problem.

	Subdomain Settings - PDE, Coefficient Form (c)
quation $\partial^2 u/\partial t^2 + d_a \partial u/\partial t + \nabla \cdot (-c \nabla$	$(u - \alpha u + \gamma) + au + \beta \cdot \nabla u = f$
Subdomains Groups	Coefficients Init Element Weak Color/Style
Subdomain selection	Initial value
1 (default)	$u(t_0) = exp(-3*(x/0.2-1)^2)$
Group: default 🛟	
Group: default	

**FIGURE 3.48** PDE, Coefficient window, Init page



**FIGURE 3.49** Telegraph equation model mesh

Analysis:	General	Time Step	ping	Advanced	)
Auto select solver	General Times to store in output:	Specifi	ed time	s	\$
Stationary	Time steps taken by solver:	Free			\$
Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated	Use stop condition Stop condition:	[			]
	Manual tuning of step siz				-
	Initial time step:	0.002			
	Maximum time step:	1.0			
Adaptive mesh refinement	Advanced Maximum BDF order: Minimum BDF order:		2		
	Singular mass matrix:		May	be	\$
	Consistent initialization of D	AE systems:	Back	ward Euler	\$
	Error estimation strategy:		Incl	ude algebraic	\$

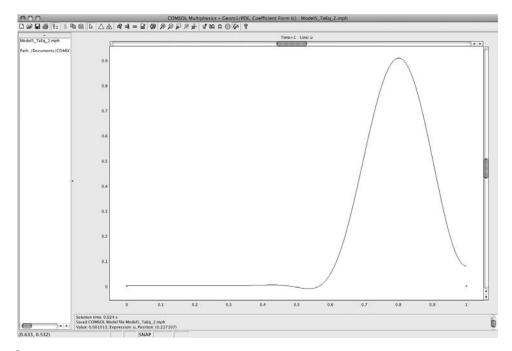
**FIGURE 3.50** Solver Parameters window, Time Stepping page

# Postprocessing

Using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the General tab. Place a check mark in the Keep current plot check box. Select "Solution at time: 0." Click the Apply button. Select "Solution at time: 0.5." Click the Apply button, and then click OK. Figure 3.51 shows the resulting plot of the pulse amplitude as it propagates from left to right. Note that the final pulse amplitude is 0.9 as compared to 0.7 for the original model.

# **Postprocessing Animation**

This solution to the telegraph equation can also be viewed as an animation. To view this solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.52). Click the Start Animation button. Save this telegraph equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3\_TE\_2.avi that was supplied with this book.



**FIGURE 3.51** Telegraph equation pulse amplitude plot, low-loss line

	General Line	Max/Min	Animate
Movie settings		Solutions	to use
File type:	AVI 🛟	Select via	: Stored output times 🖨
Width (in pixels):	640	0	-
Height (in pixels):	480	0.1 0.2	
Frames per second:	10	0.3 0.4	
	(Advanced)	0.5	
Static / Eigenfunction	animation	0.6 0.7	
Cycle type:	Full harmonic	0.8 0.9	Y
	11	Times:	
Reverse direction			
			Start Animation

**FIGURE 3.52** Telegraph equation Plot Parameters window, Animate page

#### Table 3.16 Constants Window

Expression	
1	
5	
5	

Select File > Save as. Type Model3\_TeEq\_2 in the Save As edit window. Click the Yes button to replace the earlier file.

# Second Variation on the Telegraph Equation Model

The previous solution to the telegraph equation shows a pulse propagating from left to right. Let us now explore how the model behaves when the initial conditions are modified. In this case, the argument is made larger, reflecting the behavior of a higher-loss transmission line.

As stated earlier, information transmission relies on the measurement of differences. To receive a message, the signal must be of detectable amplitude (analog), of detectable duration (time), and in the pass-band (correct frequency or frequency spread) of the receiver. The signal amplitude must be sufficiently greater than the detection threshold and above the noise level (on the average) to allow information to be collected.

First, save a new copy of the just-created telegraph equation model Model3\_ TeEq\_2 as Model3\_TeEq\_3. You can then modify the telegraph equation model without being concerned about damaging the just-built model.

Using the menu bar, select Options > Constants. After the Constants window appears, type the expressions indicated in Table 3.16 (also see Figure 3.53), and then click OK.

Name	Expression	Value	Description
c	1	1	
alpha	5	5	
beta	5	5	
i 🖬	He	Ip Apply	Cancel OK

FIGURE 3.53 Constants window

Table 3.17	Boundary	Settings	Window
------------	----------	----------	--------

Parameter	Boundary 1	Boundary 2	q	g
Type	Neumann	Neumann	0	0
Setting			0	0

#### **Boundary Conditions**

Using the menu bar, select Physics > Boundary Settings. After the Boundary Settings window appears, select both boundaries 1 and 2. Enter or verify the settings indicated in Table 3.17, as shown in Figure 3.54. Click OK.

#### **Subdomain Settings**

The next step in building the revised telegraph equation model is to set the Subdomain Settings. Select Physics > Subdomain Settings. Once the Subdomain Settings window appears, select subdomain 1. Enter or verify the coefficient values under the correct tab as shown in Table 3.18. Leave the other coefficient settings at their 0 value, as shown in Figure 3.55.

Once the PDE coefficients have been entered or verified, click the Init tab. Type or verify the initial conditions found in Table 3.19 in the edit windows, as shown in Figure 3.56. Click OK.

Boundaries Groups		Coefficients Weak Color
Boundary selection	Neum     Dirich	conditions nann boundary condition nlet boundary condition nt Value/Expression
	q	0
	g	0
Group:	h	1
Select by group	r	0



PDE Coefficient	Value
С	C*C
а	alpha*beta
f	–(alpha+beta)*ut
e <sub>a</sub>	1
d <sub>a</sub>	0

# Table 3.18 Subdomain Settings Window, PDE Coefficients

Subdomains Groups			ement Weak Color/Style
ubdomain selection	-PDE coefficient	Value/Expression	Description
	c	c*c	Diffusion coefficient
	a	alpha*beta	Absorption coefficient
	f	-(alpha+beta)*ut	Source term
	e <sub>a</sub>	1	Mass coefficient
	da	0	Damping/Mass coefficient
	α	0	Conservative flux convection coeff
Group: default	β	0	Convection coefficient
Select by group	Y	0	Conservative flux source term

**FIGURE 3.55** PDE, Subdomain Settings window, PDE Coefficients

# Table 3.19Initial Conditions Window

Value
exp(-3*(x/0.2-1)^2)
0

Subdomains Groups	Coefficients Init Element Weak Color/Style
domain selection	Initial value
default)	u(t <sub>0</sub> ) exp(-3*(x/0.2-1)^2)
	ut(t <sub>0</sub> ) 0
	17 M
roup: default 🛟	
Select by group	

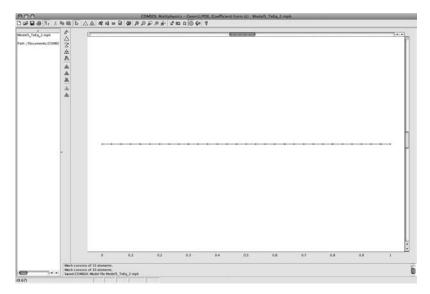
**FIGURE 3.56** PDE, Coefficient window, Init page

# **Model Reset**

Select File > Reset Model > Yes.

### **Mesh Generation**

Using the tool bar, select Initialize Mesh > Refine Mesh once. The final 30-element mesh is shown in Figure 3.57.





000	Solver Parameters
Analysis:	General Time Stepping Advanced
Auto select solver Solver: Stationary Time dependent	General Times to store in output: Specified times  Time steps taken by solver: Free
Eigenvalue Parametric Stationary segregated Parametric segregated	Use stop condition
	Manual tuning of step size       Initial time step:     0.002       Maximum time step:     1.0
Adaptive mesh refinement	Advanced         Maximum BDF order:       2         Minimum BDF order:       1         Singular mass matrix:       Maybe         Consistent initialization of DAE systems:       Backward Euler         Error estimation strategy:       Include algebraic
	Help Apply Cancel OK

**FIGURE 3.58** Solver Parameters window, Time Stepping page

### Solving the Telegraph Equation Model

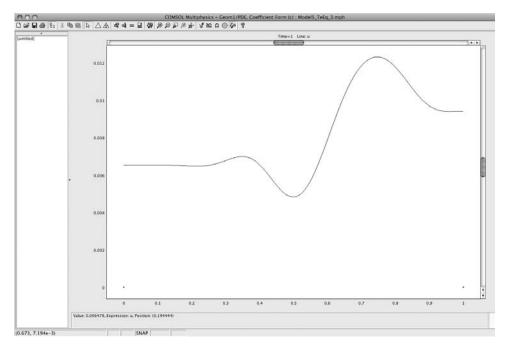
Using the menu bar, select Solve > Solver Parameters. Once the Solver Parameters window appears, click the Time Stepping tab. Place a check mark in the Manual tuning of step size check box. Enter or verify 0.002 in the Initial time step edit field, as shown in Figure 3.58. Click OK.

NOTE As mentioned in an earlier note, the 0.002 time step is selected to yield adequate solution resolution without requiring extensive resource consumption (computer/modeler time).

Using the menu bar, select Solve > Solve Problem. The solution for the final time interval is as shown in Figure 3.59.

#### Postprocessing

Using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the General tab. Place a check mark in the Keep



**FIGURE 3.59** Telegraph equation solution, final time interval

current plot check box. Select "Solution at time: 0." Click the Apply button. Select "Solution at time: 0.5." Click the Apply button, and then click OK. Figure 3.60 shows the resulting plot of the pulse amplitude as it propagates from left to right.

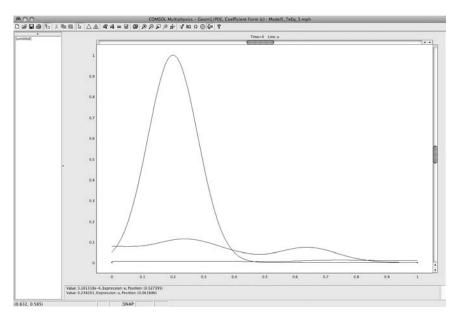
# **Postprocessing Animation**

This solution to the telegraph equation can also be viewed as an animation. To view this solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 3.61). Click the Start Animation button. Save this telegraph equation model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie3\_TE\_3.avi that was supplied with this book.

Select File > Save as. Type Model3\_TeEq\_3 in the Save As edit window. Click the Yes button to replace the earlier file.

# **1D Telegraph Equation Models: Summary and Conclusions**

The telegraph equation is a powerful tool that can be used to model wave propagation in diverse transmission lines. It can be used to thoroughly characterize the propagation conditions of coaxial lines, twin pair lines, microstrip lines, and more. The telegraph equation is easily and simply modeled with a 1D PDE mode model.



**FIGURE 3.60** Telegraph equation pulse amplitude plot, high-loss line

0	Plot P	arameters	
	General Line	Max/Min	Animate
Movie settings		Solution	is to use
File type:	AVI 🛟	Select v	ia: Stored output times 🛟
Width (in pixels):	640	0	
Height (in pixels):	480	0.1	
Frames per second:	10	0.3	
	Advanced	0.5	
Static / Eigenfunction	animation	0.6	
Cycle type:	Full harmonic	0.8 0.9	
Number of frames:	11	Times:	
Reverse direction			
			Start Animation
	_		
	He		pply Cancel C

**FIGURE 3.61** Telegraph equation Plot Parameters window, Animate page

# References

- 1. http://en.wikipedia.org/wiki/KdV
- 2. http://en.wikipedia.org/wiki/List\_of\_nonlinear\_partial\_differential\_equations
- 3. http://en.wikipedia.org/wiki/Exactly\_solvable
- 4. http://en.wikipedia.org/wiki/Nonlinear\_system
- 5. http://en.wikipedia.org/wiki/John\_Scott\_Russell
- 6. http://en.wikipedia.org/wiki/Soliton
- 7. http://en.wikipedia.org/wiki/Soliton\_%28optics%29
- 8. http://en.wikipedia.org/wiki/Signal\_to\_noise
- 9. http://en.wikipedia.org/wiki/Soliton\_%28optics%29
- 10. http://en.wikipedia.org/wiki/Telegraph\_equation
- 11. http://en.wikipedia.org/wiki/Oliver\_Heaviside
- 12. http://en.wikipedia.org/wiki/Transmission\_line

# Exercises

- 1. Build, mesh, and solve the 1D KdV equation problem presented in this chapter.
- 2. Build, mesh, and solve the first variation of the KdV equation problem presented in this chapter.
- 3. Build, mesh, and solve the second variation of the KdV equation problem presented in this chapter.
- 4. Build, mesh, and solve the telegraph equation problem presented in this chapter.
- 5. Build, mesh, and solve the first variation of the telegraph equation problem presented in this chapter.
- 6. Build, mesh, and solve the second variation of the telegraph equation problem presented in this chapter.
- 7. Explore other variations of the arguments in the KdV equation model.
- 8. Explore other variations of the arguments in the telegraph equation model.
- 9. Explore the role that characteristic impedance plays in transmission lines.

# **4** 2D Modeling

# In This Chapter

2D Guidelines for New COMSOL® Multiphysics® Modelers
2D Modeling Considerations
Coordinate System
2D Electrochemical Polishing (Electropolishing) Theory
COMSOL 2D Electrochemical Polishing Model
First Variation on the 2D Electrochemical Polishing Model
Second Variation on the 2D Electrochemical Polishing Model
2D Electrochemical Polishing Models: Summary and Conclusions
2D Hall Effect Model Considerations
2D Hall Effect Model
First Variation on the 2D Hall Effect Model
Second Variation on the 2D Hall Effect Model
2D Hall Effect Models: Summary and Conclusions

# **2D** Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers

# **2D Modeling Considerations**

2D modeling can be less difficult than 1D modeling, having fewer implicit assumptions, and yet potentially can still be a challenging type of model to build, depending on the underlying physics involved, irrespective of the modeling software utilized. The least difficult aspect of 2D model building arises from the fact that the geometry is relatively simple: In a 2D model, the modeler has only a single plane as the modeling space. However, the physics in a 2D model can range from relatively easy to extremely complex.

**NOTE** COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software has two 2D modeling modes: 2D (beginning-level through advanced-level 2D modeling) and 2D Axisymmetric (advanced-level 2D modeling). In keeping with the introductory focus of the material in this text, both model types, their associated physics, and the related methodology for

use of the models, are introduced in Chapters 4 and 5. Significantly more advanced 2D modeling techniques exist than are presented in these two chapters. Examples of some of those more challenging techniques are reserved for introduction in Chapters 6 and 7. For further expansion of the 2D modeling horizons, refer to the COMSOL manuals, the COMSOL Website, and the general COMSOL Multiphysics software-related research literature.

The 2D model implicitly assumes, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables that are of interest are homogeneous, isotropic, and constant, unless otherwise specified, throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model. Bearing that in mind, the modeler needs to ensure that all of the modeling conditions and associated parameters (default settings) in each new model created have been properly considered, defined, or set to the appropriate values.

The modeler also needs to seriously consider the steps that will be required in properly establishing the correct postprocessing and visualization settings to extract the desired information from the modeling solution. The default parameter settings on any given model will probably not present exactly the information that the modeler needs or desires, although it will probably come close. It is the responsibility of the modeler to determine exactly which of the myriad of postprocessing and visualization choices available in the COMSOL Multiphysics software to employ.

It is always preferable for the modeler to be able to accurately anticipate the expected behavior (results) of the model and the way in which those results should be presented. Never assume that the default values that are initially present when the model is first created will suit the needs of a new model. Always verify that the values employed in the model are the correct values needed for that model. Calculated solution values that significantly deviate from the expected values or from comparison values measured in experimentally derived realistic models are probably indicative of one or more modeling errors either in the original model design, in the earlier model analysis, or in the understanding of the underlying physics, or are simply due to human error.

# **Coordinate System**

In 2D models, there are three coordinates: space (x), space (y), and time (t). In a steadystate solution to a 2D model, parameters can vary only as a function of position in the space (x) and space (y) coordinates. Such a 2D model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (x) and space (y), and in time (t). The transient solution model is essentially a sequential collection of steady-state (quasi-static) solutions. The space coordinates (x) and (y) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ( $x_{\min} < = x < = x_{\max}$ ) and ( $y_{\min} < = y < = y_{\max}$ ). The time coordinate (t) represents the range of values ( $t_{\min} < = t < = t_{\max}$ ) from the beginning of observation period ( $t_{\min}$ ) to the end of observation period ( $t_{\max}$ ).

To assist the reader to achieve a broader exposure to the applicability of the physics discussed here and to demonstrate the power of the basic COMSOL 2D modeling techniques, the modeling examples in this chapter illustrate techniques from two substantially different, but important and widely applied technologies currently employed in applied engineering and physics. The first example presented, electropolishing, explores the modeling of a processing methodology utilized in the fabrication and finishing of many metallic objects that require a smooth surface (e.g., microscope samples, precision metal parts, medical equipment and tools, large and small metal drums, thin analytical samples, vacuum chambers). The second example, the Hall effect (a magnetic sensor technology), explores the behavior of currents (electrons or holes) flowing in a semiconducting material (e.g., Si, Ge) under the influence of an external magnetic field.

# 2D Electrochemical Polishing (Electropolishing) Theory

Electrochemical polishing<sup>1</sup> (also known as electropolishing<sup>2</sup>) is a well-known process in the metal finishing industry. It allows the finished surface smoothness of a conducting material to be cleanly controlled to a high degree of precision, using relatively simple processing equipment. The electrochemical polishing technique eliminates the abrasive residue typically present on the polished surface from a mechanical polishing process; it also eliminates the need for complex, mechanical polishing machinery.

The science of electricity, and consequently that of electrochemistry, started with the work of William Gilbert through his study of magnetism. Gilbert first published his studies in 1600.<sup>3</sup> Charles-Augustin de Coulomb,<sup>4</sup> Joseph Priestley,<sup>5</sup> Georg Ohm,<sup>6</sup> and others made additional independent contributions that furthered the basic understanding of the nature of electricity and electrochemistry. Those contributions led to the discovery and disclosure by Michael Faraday<sup>7</sup> of his two laws of electrochemistry in 1832.

The numerical solution model for electrochemical polishing was originally developed by COMSOL for distribution with the Multiphysics software as a COMSOL Multiphysics electromagnetics model. This model introduces two important basic concepts, the first in applied physics and the second in applied modeling: (1) electropolishing and (2) the moving mesh (ALE = arbitrary Lagrangian–Eulerian<sup>8</sup>). The electrochemical polishing model built in this chapter is substantially the same as presented in the COMSOL Model Library. In this chapter, following development of the first model, variations and expansions on the basic electrochemical polishing model are explored.

Note It is important for the new modeler to personally build each model presented within this text. There is no substitute in the path to an understanding of the modeling process for the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and subsequently correct errors, adding to his or her experience and fund of modeling knowledge. Even building the simplest model will expand the modeler's fund of knowledge.

Polishing (smoothing) of a material surface, via either mechanical or electrochemical means, results from the reduction of asperities (bumps) to achieve a nominally smooth surface (uniform thickness  $\pm \Delta$  thickness). In a mechanical polishing technique, the reduction of asperities occurs through the use of finer (smaller) and finer grit (abrasive) sizes. The mechanical polishing of many surfaces is difficult, if not impossible, owing to the complexity and/or physical size of such surfaces. Figure 4.1 shows a simple asperity, as will be modeled in this section of the chapter.

The surface of the electrode, using this method, is polished by the differential removal of material from local asperities in selected areas, accomplished through the immersion of the nominally rough electrode in an electrolyte and the application of a current (electron bombardment). A first-order approximation to the experimentally observed material removal process is that the rate (velocity) of material removal (U) from the electrode surface is proportional to the amplitude of the current and direction of the current **J**, relative to the local surface normal vector **n** (see Figure 4.2):

$$U = -K^* \mathbf{J} \cdot \mathbf{n} = -K^* J_n \tag{4.1}$$

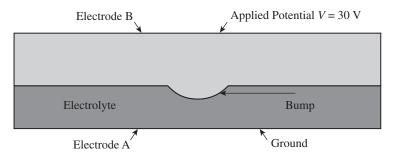


FIGURE 4.1 2D An asperity (bump) on an electrode

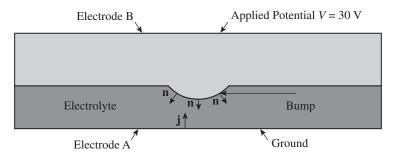


FIGURE 4.2 Surface normal vector **n** and the current vector **J** 

**NOTE** The electropolishing technique, to a first approximation, is the inverse of electroplating. As a result, the rate of removal of material (velocity = U) from the nominally rough surface of the positive electrode is proportional to the normal current density at the positive electrode surface, as shown in equation 4.1.

The exact value of the proportionality constant (K) in physical applications (e.g., research experiments, processing) is determined by the electrode material, the electrolyte, the temperature, and other factors, and, to some extent, will be explored in later examples in this chapter.

For this model, the proportionality constant is chosen to be

$$K = 1.0 \times 10^{-11} \,\mathrm{m}^3/(\mathrm{A*s}) \tag{4.2}$$

where m = metersA = amperess = seconds

Obviously, because material is removed from the positive electrode during the electropolishing process, the spacing between the upper and lower electrodes will increase. The time rate of change of the model geometry (electrode spacing) needs to be accommodated somewhere within the model. The Moving Mesh (ALE = arbitrary Lagrangian–Eulerian) Application Mode accommodates that time rate of change, resulting from the normal current  $(J_n)$  flowing in the electrolyte during the quasi-static use of the Conductive Media DC Application Mode.

The Moving Mesh Application Mode allows the modeler to create models in which the physics of the process introduces and controls geometric changes in the model. However, the modeler must know and work carefully within the limits of the modeling system. The Moving Mesh Application Mode is a powerful tool. However, the calculated mesh parameters can drift, as the mesh is deformed and ultimately lead to 1

00	Model Navigator	
New	Model Library User Models	Open Settings
Electrosta	nd Diffusion tics re Media DC tics tatics Electromagnetics is	Add Remove Geom1 (2D) Frame (ale) Moving Mesh (ALE) (ale) Conductive Media DC (dc) Dependent variables: V Application Mode Propertie Add Geometry
Dependent variables:	V2	Add Frame
Application mode name:	dc2	Ruling application mode: Moving Mesh (ALE) (ale)
Element:	Lagrange – Quadra 🛊	Multiphysics

FIGURE 4.3 2D Electropolishing\_1 Model Navigator setup

nonphysical, nonconvergent results. Avoidance of such nonphysical results requires the modeler to understand the basic physics of the modeled problem and to choose the meshing method that yields the best overall results.

# **COMSOL 2D Electrochemical Polishing Model**

To start building the Electropolishing\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Deformed Mesh > Moving Mesh (ALE) > Transient analysis. Click the Multiphysics button, and then click the Add button.

From the Application Modes list, select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. Click the Add button. See Figure 4.3. Click OK.

Name	Expression	Description
К	1.0e-11[m^3/(A*s)]	Coefficient of proportionality

Table 4.1	Constants Edit	t Window
-----------	----------------	----------

Name	Expression	Value	Description
К	1.0e-11 [m^3/(A*s)]	(1e-11)[m <sup>3</sup> /(s·A)]	Coefficient of proportionality
c		Help	Apply Cancel OK

**FIGURE 4.4** 2D Electropolishing\_1 model Constants edit window

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.1; see Figure 4.4. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 2.8 and a height of 0.4. Select "Base: Corner" and set X equal to -1.4 and Y equal to 0 in the Rectangle edit window. See Figure 4.5.

Click the Apply button, and then click OK. See Figure 4.6.

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.3. Select "Base: Center" and set X equal to 0 and Y equal to 0.6 in the Circle edit window. See Figure 4.7.

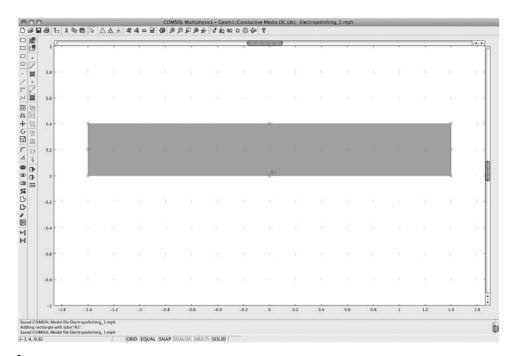
Click the Apply button, and then click OK. See Figure 4.8.

Select both the rectangle and the circle by clicking on the rectangle and Shiftclicking on the circle. See Figure 4.9.

Click the Difference button on the Draw toolbar to remove the overlapping portion of the circle from the rectangle. The upper surface of the electrolyte rectangle

Size		Rotat	ion angle-	
Width:	2.8	α:	0	(degrees)
Height:	0.4			
Position				
Base:	Corner	Style:	Solid	+
X:	-1.4	Name:	R1	
Y:	0			

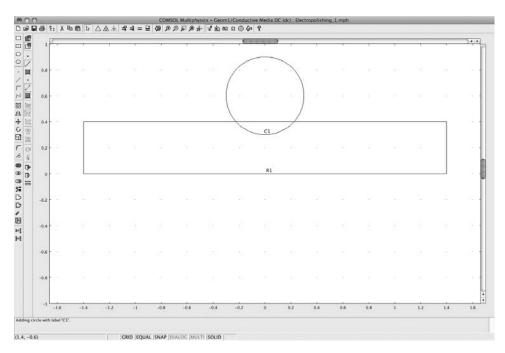
**FIGURE 4.5** 2D Electropolishing\_1 model Rectangle edit window



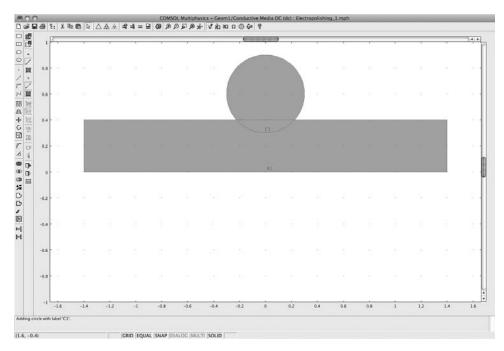
**FIGURE 4.6** 2D Electropolishing\_1 model electrolyte rectangle

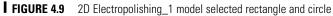
Size		Rotation angle
Radius:	0.3	α: 0 (degrees
Positior	1	7
Base:	Center	Style: Solid
X:	0	Name: C1
Y:	0.6	

**FIGURE 4.7** 2D Electropolishing\_1 model Circle edit window



**FIGURE 4.8** 2D Electropolishing\_1 model rectangle and circle





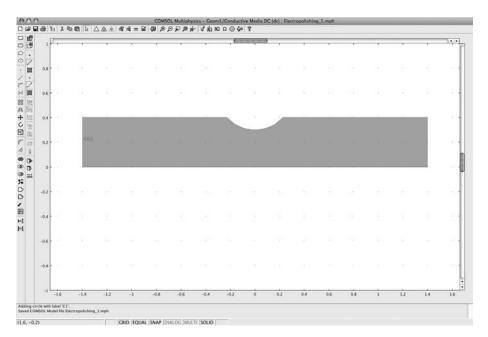


FIGURE 4.10 2D Electropolishing\_1 model electrode with asperity

(CO1) is the lower surface of the electrode, with the asperity, that will be electropolished. See Figure 4.10.

The model geometry, as presently scaled, is 1.4 meters in length and 0.4 meter in height. Electropolishing is typically applied as the final finishing (smoothing) step in a precision fabrication process (e.g., metallographic samples, vacuum chambers). Thus the model geometry will need to be reduced in scale to emulate reality.

Click on the text "CO1." Next, click the Scale button on the Draw toolbar. Enter 1e-3 in both the X and Y Scale factor edit windows. See Figure 4.11. Click OK.

Click the Zoom Extents button on the menu bar. See Figure 4.12.

Sca	ale factor	<u> </u>
X:	1e-3	ОК
Y:	1e-3	Cancel
Sca	ale base point	Help
X:	0	
Y:	0	

FIGURE 4.11 2D Electropolishing\_1 model Scale edit window

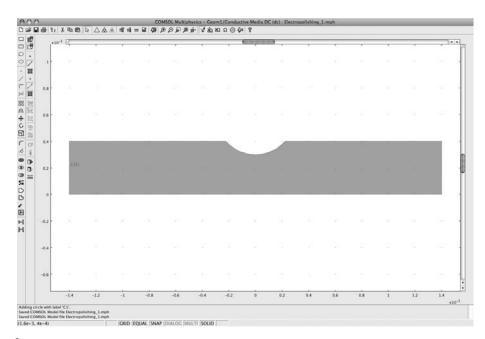


FIGURE 4.12 2D Electropolishing\_1 model scaled electrolyte/electrode geometry

# Physics Subdomain Settings: Conductive Media DC

Having established the 2D geometry for the electrochemical polishing model (a rectangle with a negative asperity on the upper surface), the next step is to define the fundamental physics conditions. Using the menu bar, select Multiphysics > Conductive Media DC. Next, using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain).

Enter 10 in the Electric conductivity ( $\sigma$ ) edit window. See Figure 4.13. Click OK.

#### Physics Boundary Settings: Conductive Media DC

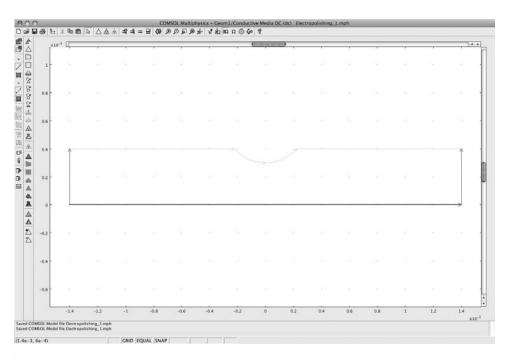
Using the menu bar, Select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 4.2, and then click OK. See Figure 4.14.

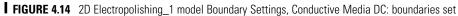
#### Table 4.2 Subdomain Settings, Conductive Media DC Window

Boundary	<b>Boundary Condition</b>	Value/Expression
1, 5	Electric insulation	_
3, 4, 6 ,7	Electric potential	30
2	Ground	—

Subdomains Groups		Physics Init E	lement C	olor
bdomain selection	-Material properties and Library material:		oad)	
	Quantity J <sup>e</sup> Q <sub>j</sub> d Conductivity relation: σ	Value/Expression 0 0 0 0 1 Conductivity 10	Unit A/m <sup>2</sup> A/m <sup>3</sup> m	Description External current density Current source Thickness Electric conductivity
oup: 🔹				

**FIGURE 4.13** Subdomain Settings window





Boundary	Coordinate	Boundary Condition	Value/Expression
1, 5	Global	Mesh velocity	vx = 0
3, 4, 6, 7	Tangent and normal	Mesh velocity	$vn = -K*nJ_dc$
	Deformed mesh		
2	Global	Mesh displacement	dx = 0, dy = 0

## Table 4.3 Boundary Settings, Moving Mesh (ALE) Window

# Physics Boundary Settings: Moving Mesh (ALE)

Using the menu bar, select Multiphysics > Moving Mesh (ALE). Next, using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 4.3, and then click OK. See Figure 4.15.

# **Mesh Generation**

On the menu bar, click the Initialize Mesh button once.

Click the Refine Mesh button once. This results in a mesh of approximately 700 elements.

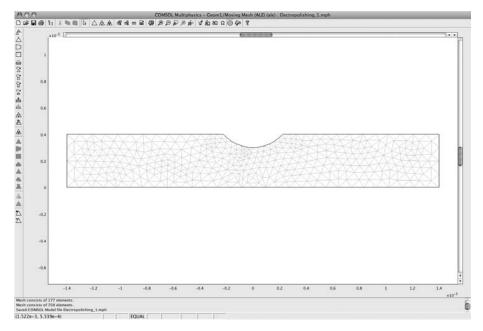
Click OK. See Figure 4.16.

# Solving the 2D Electrochemical Polishing Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver. Enter 0:1:10 (typical values) in the Times edit window, as shown in Figure 4.17. This instruction causes the Solver to divide the modeling time-space into 10 equal intervals, over the period from 0 to 10 seconds. Click the Apply button, and then click OK.

Boundaries Groups		Mesh	Weak Constr. Color/Style	
Boundary selection	Mesh displacement			
1 2	Coordinate system: Tangent and normal coord. sys. in deformed mesh			
3	Quantity	Value/Expression	Unit	Description
4 5	O Mesh displacer	nent		
6	🗌 dn	0	m	Mesh displacement, n direction
7	🗌 dt	0	m	Mesh displacement, t direction
Group:	Mesh velocity			
Select by group	🗹 vn	-K*nJ_dc	m/s	Mesh velocity, n direction
Interior boundaries	🗌 vt	0	m/s	Mesh velocity, t direction

FIGURE 4.15 2D Electropolishing\_1 model Boundary Settings, Moving Mesh (ALE): boundaries 3, 4, 6, 7 win-



**FIGURE 4.16** 2D Electropolishing\_1 model mesh

Analysis:	Ge	neral Time St	epping A	dvanced
Transient 🗘	Time stepping		0:1:10	
Stationary Time dependent Elgenvalue Parametric	Relative tolerance: Absolute tolerance Allow complex nu	mbare	0.01	
Stationary segregated Parametric segregated Adaptive mesh refinement	Linear system solver	Direct (UMFPA	SCK)	\$
	Matrix symmetry:	Automatic		
		Help	Apply	Cancel O

**FIGURE 4.17** 2D Electropolishing\_1 model Solver Parameters window

**NOTE** The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, always change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can try a variation on the model parameter space to seek improved results.

Using the menu bar, select Solve > Solve Problem.

### Postprocessing

Select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked.

From the Predefined quantities drop-down list, select "Conductive Media DC (dc) > Total current density, norm." See Figure 4.18.

Conservation Conservation		
General Surface	Contour Boundar	y Arrow Principal
Surface plot		
	Surface Data Height Data	L
	Constant 1	
Predefined quantities:	Total current density, norm	Range
Expression:	normj_dc	Smooth
Unit:	A/m <sup>2</sup>	
onit.	(A/m	
oloring and fill		
Coloring: Interpo	lated 🗘 Fill style:	Filled
	rin style.	
urface color		N 10
Colormap:	jet 🗘 Colors: 10	24 🗹 Color scale
O Uniform color: (	Color	
O Uniform color: (	Color	
O Uniform color: (	Color	
O Uniform color: (	Color	
O Uniform color: (	Color	
O Uniform color: (	Color	,
O Uniform color: (	Color	
O Uniform color: (	Color	
O Uniform color: (	Color	
Uniform color: (	Color	
O Uniform color: (	Color	
O Uniform color: (	Color	
O Uniform color: (	Color	
O Uniform color: (	Color	

FIGURE 4.18 2D Electropolishing\_1 model Plot Parameters window, Surface page

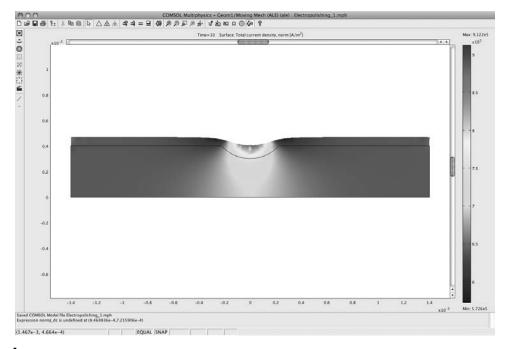


FIGURE 4.19 2D Electropolishing\_1 model Surface plot window

Click OK. See Figure 4.19.

In Figure 4.19, the model calculation shows that the maximum current density is approximately 0.92e5 A/m<sup>2</sup>, in the region of the asperity. Figure 4.19 also shows that the normal current density ( $J_n$ ) concentrated in the region of the asperity is approximately 1.5 times the normal current density elsewhere on the electrode surface. As a result, the removal rate of the electrode material will be approximately 1.5 times as high.

To see the change in the position of the electrode surface and the relative removal of material from the asperity, first select Postprocessing > Plot Parameters. Next, click the Surface tab, and verify that the Surface plot check box is checked.

From the Predefined quantities drop-down list, select "Moving Mesh (ALE) (ale) > y-displacement." See Figure 4.20.

Click OK. Figure 4.21 shows the displacement of the electrode surface in the *y*-direction (dy\_ale) after 10 seconds of electropolishing.

In Figure 4.21, the model calculation shows that the maximum displacement of the electrode surface in the y-direction (dy\_ale) after 10 seconds of electropolishing is approximately 1.08e-4 m (0.108 mm), in the region of the asperity.

General	_	Plot	Parameters		
General	Surface	Contour	Boundary	Arrow	Principal
Surface plo	t				
	(	Surface Data	Height Data		
Predefined qu	uantities:	y-displaceme	nt 🗘	Range.	
Expression:		dy_ale		Smoot	h
Unit:		m	\$	)	
Caladian and D					
Coloring and fi Coloring:	ll Interpol	ated 🛟	Fill style:	Filled	•
Surface color	Interpor		rin style.	Thicu	•
Colormap	e (	jet 🗘	Colors: 1024	Color	scale
O Uniform c	olor:	Color			
O Uniform c	olor:	Color			
O Uniform c	olor:	Color			
O Uniform c	olor:	Color			
O Uniform c	olor:	Color			
Uniform c	olor:	Color			
Uniform c	olor: (	Color			
O Uniform c	olor: (	Color			
O Uniform c	olor: (	Color			

**FIGURE 4.20** 2D Electropolishing\_1 model Surface plot window: Moving Mesh (ALE) (ale), y displacement (dy\_ale)

**NOTE** The result of the modeling calculation is approximately 1.1e-4 m. Calculating the estimated result on a "first principles" basis:

$$d = |U|\Delta t = K|J_n|\Delta t = \left(10^{-11}\frac{\mathrm{m}^3}{\mathrm{A}^*\mathrm{s}}\right) * \left(10^6\frac{\mathrm{A}}{\mathrm{m}^2}\right) * (10^1\mathrm{s}) = 10^{-4}\mathrm{m} \qquad (4.3)$$

This agrees well with the results of the model.

## **Postprocessing Animation**

This solution to the 2D electrochemical polishing model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing >

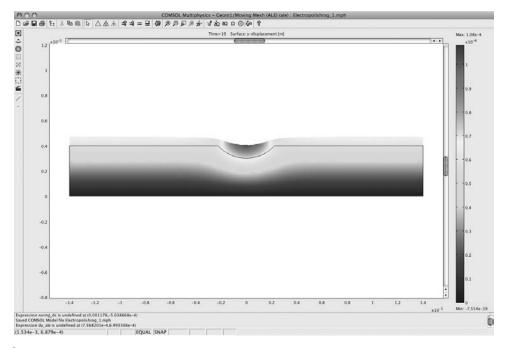


FIGURE 4.21 2D Electropolishing\_1 model Surface plot window: Moving Mesh (ALE) (ale), y displacement (dy\_ale)

Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.22). Click the Start Animation button. Save this 2D electrochemical polishing model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4\_EP\_1.avi that was supplied with this book.

# First Variation on the 2D Electrochemical Polishing Model

Note: This model will explore the effect of the mesh element type (triangle, quadrilateral [quad]) on the ultimate values determined by the calculated solution. Both the model geometry and the model mesh play major roles in the ease of solving any particular problem.

To start building the Electropolishing\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Deformed Mesh > Moving Mesh (ALE) > Transient analysis. Click the Multiphysics button, and then click the Add button.

Movie settings   File type:   AVI   Width (in pixels):   640   Height (in pixels):   480   Frames per second:   10   Advanced   Static / Eigenfunction animation   Cycle type:   Full harmonic   Number of frames:   11   Reverse direction   Use camera settings from main window	File type: AVI Width (in pixels): 640 Height (in pixels): 480 Frames per second: 10 Advanced Static / Eigenfunction animation Cycle type: Full harmonic  Number of frames: 11 Reverse direction	File type: AVI Width (in pixels): 640 Height (in pixels): 480 Frames per second: 10 Advanced Static / Eigenfunction animation Cycle type: Full harmonic  Number of frames: 11 Reverse direction	Principal S1	reamline Particle	Tracing Max/Min	Deform Anima
Cycle type: Full harmonic C Times: Times: Times:	Cycle type: Full harmonic C Times: Times: Times:	Cycle type: Full harmonic C Times: Times: Times:	File type: Width (in pixels): Height (in pixels): Frames per second:	640 480 10 Advanced	Select via: Stor 0 1 2 3 4 5 6 7	ed output times 🔹
Use camera settings from main window	Use camera settings from main window	Use camera settings from main window	Curles	Full harmonic		Y
			Number of frames:	(1.)	Times:	
			Number of frames:	n		

**FIGURE 4.22** 2D Electropolishing\_1 model animation Plot Parameters window

Using the Application Modes list, select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. Click the Add button. See Figure 4.23.

Click OK. Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.4; see Figure 4.24.

Table 4.4	Constants Edit Window
-----------	-----------------------

Name	Expression	Description
К	1.0e-11[m^3/(A*s)]	Coefficient of proportionality

	Mode	l Navigator		
New		Jser Models	Open S Multiphysics	ettings
Electrosta	nd Diffusion tics ve Media DC tics tatics Electromagnetics		Add Geom1 ( Fram Dependent v Applica	Remove 2D) te (ale) Moving Mesh (ALE) (ale) conductive Media DC (dc) arriables: V tion Mode Propertie dd Geometry
Dependent variables: Application mode name:	V2 dc2	Ri	uling applica Moving Me	Add Frame tion mode: sh (ALE) (ale)

**FIGURE 4.23** 2D Electropolishing\_2 Model Navigator setup

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 2.8 and a height of 0.4. Select "Base: Corner" and set X equal to -1.4 and Y equal to 0 in the Rectangle edit window. See Figure 4.25.

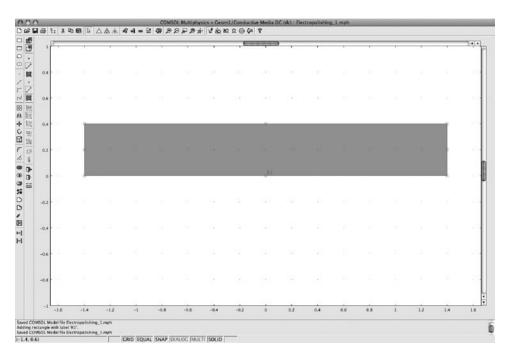
Click the Apply button, and then click OK. See Figure 4.26.

000	O Constants					
Name	Expression	Value	Description			
K	1.0e-11 [m^3/(A*s)]	(1e-11)[m <sup>3</sup> /(s·A)]	Coefficient of proportionality			
C		Help	Apply Cancel OK			

**FIGURE 4.24** 2D Electropolishing\_2 model Constants edit window

) 0 0	)	Rectangle
Size		Rotation angle
Width:	2.8	α: 0 (degrees
Height:	0.4	
Position		
Base:	Corner	Style: Solid
<b>X</b> :	-1.4	Name: R1
Y:	0	

**FIGURE 4.25** 2D Electropolishing\_2 model Rectangle edit window



**FIGURE 4.26** 2D Electropolishing\_2 model electrolyte rectangle

Size		Rotation angle
Radius:	0.3	α: 0 (degree
Position		
Base:	Center	Style: Solid
x:	0	Name: C1
y:	0.6	

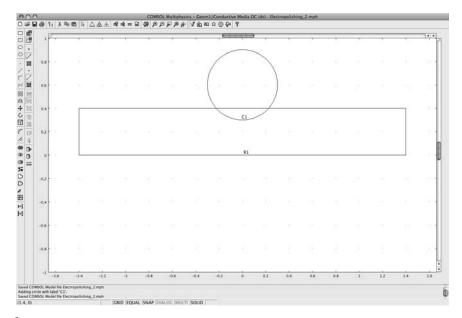
**FIGURE 4.27** 2D Electropolishing\_12 model Circle edit window

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.3. select "Base: Center" and set X equal to 0 and Y equal to 0.6 in the Circle edit window. See Figure 4.27.

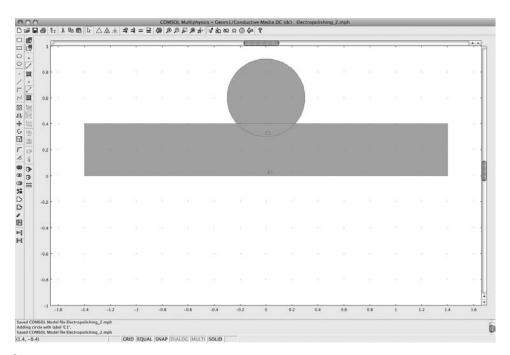
Click OK. See Figure 4.28.

Select both the rectangle and the circle by clicking on the rectangle and Shiftclicking on the circle. See Figure 4.29.

Click the Difference button on the Draw toolbar to remove the overlapping portion of the circle from the rectangle. The upper surface of the electrolyte rectangle (CO1) is the lower surface of the electrode, with the asperity, that will be electropolished. See Figure 4.30.







**FIGURE 4.29** 2D Electropolishing\_12 model selected rectangle and circle

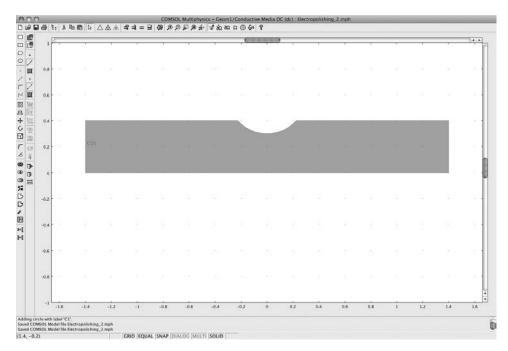


FIGURE 4.30 2D Electropolishing\_12 model electrode with asperity

Sca	ale factor	6 04
X:	1e-3	ОК
Y:	1e-3	Cancel
Sca	ale base point	Help
X:	0	
Y:		

FIGURE 4.31 2D Electropolishing\_12 model Scale edit window

The model geometry, as presently scaled, is 1.4 meters in length and 0.4 meter in height. Electropolishing is typically applied as the final finishing (smoothing) step in a precision fabrication process (e.g., metallographic samples, vacuum chambers). Thus the model geometry will need to be reduced in scale to emulate reality.

Click on the text "CO1." Click the Scale button on the Draw toolbar. Enter 1e-3 in both the X and Y text boxes in the Scale factor edit windows. See Figure 4.31.

Click OK, and then click the Zoom Extents button on the menu bar. See Figure 4.32.

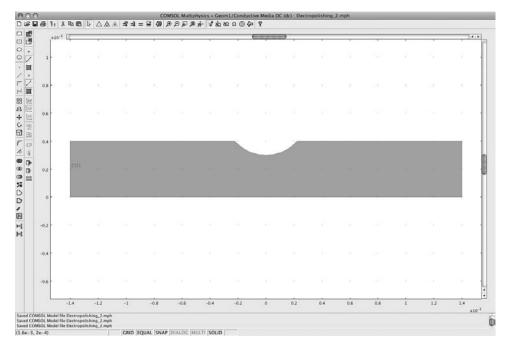


FIGURE 4.32 2D Electropolishing\_12 model scaled electrolyte/electrode geometry

Subdomains Groups		Physics Init E	lement 0	olor ]
ubdomain selection	Material properties and	I sources		
	l ibrary material:	: (	oad)	
	Quantity	Value/Expression	Unit	Description
	J <sup>e</sup>	0 0	A/m <sup>2</sup>	External current density
	Qj	0	A/m <sup>3</sup>	Current source
	d	1	m	Thickness
	Conductivity relation:	Conductivity	\$	
	σ	10	5/m	Electric conductivity
roup:				
Select by group				
Active in this domain				

**FIGURE 4.33** Subdomain Settings window

# Physics Subdomain Settings: Conductive Media DC

Having established the 2D geometry for the electrochemical polishing model (a rectangle with a negative asperity on the upper surface), the next step is to define the fundamental physics conditions. Using the menu bar, select Multiphysics > Conductive Media DC. Next, using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). Enter 10 in the Electric conductivity ( $\sigma$ ) edit window. See Figure 4.33. Click OK.

# Physics Boundary Settings: Conductive Media DC

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 4.5. See Figures 4.34, 4.35, and 4.36.

## Table 4.5 Boundary Settings, Conductive Media DC Window

Boundary	Boundary Condition	Value/Expression
1, 5	Electric insulation	_
3, 4, 6, 7	Electric potential	30
2	Ground	—

J = 0	
Boundaries Groups Boundary selection	Conditions       Color/Style         Boundary sources and constraints       Library material:         Library material:

FIGURE 4.34 Boundary Settings (1, 5), Conductive Media DC: boundaries set

Click OK. See Figure 4.37.

#### Physics Boundary Settings: Moving Mesh (ALE)

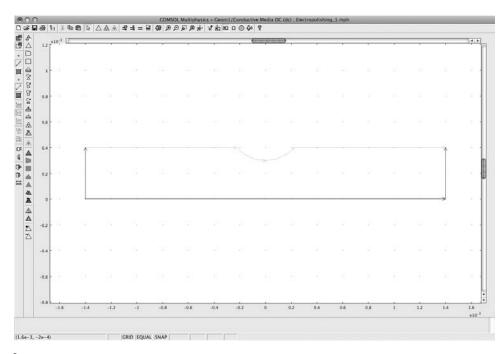
Using the menu bar, select Multiphysics > Moving Mesh (ALE). Next, using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or

00	Boundary Settings - Conductive Media DC (dc)
quation / = V <sub>0</sub>	
Boundaries Groups	Conditions Color/Style
Boundary selection	Boundary sources and constraints Library material: Library material: Load Boundary condition: Electric potential Quantity Value/Expression Unit Description V <sub>0</sub> 30 V Electric potential
Interior boundaries	(Help Apply Cancel O

**FIGURE 4.35** Boundary Settings (3, 4, 6, 7), Conductive Media DC: boundaries set

Boundaries Groups	Conditions Color/Style
80undary selection 1 2 3 4 5 6 7	Boundary sources and constraints Library material:
Group: 👘 💠	

FIGURE 4.36 Boundary Settings (2), Conductive Media DC: boundaries set



**FIGURE 4.37** Boundary Settings (1, 5 = blue; 3, 4, 6, 7 = green; 2 = red), Conductive Media DC: boundaries set

Boundary	Coordinate	Boundary Condition	Value/Expression
1, 5	Global	Mesh velocity	vx = 0
3, 4, 6, 7	Tangent and normal Deformed mesh	Mesh velocity	$vn = -K*nJ_dc$
2	Global	Mesh displacement	dx = 0, dy = 0

#### Table 4.6 Boundary Settings, Moving Mesh (ALE) Window

enter the given boundary condition and value in the edit windows as indicated in Table 4.6. See Figure 4.38, 4.39, and 4.40.

Click OK. Figure 4.41 shows the Boundary Settings, Moving Mesh (ALE) options organized by color.

#### **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 1 in the Subdomain selection window. Enter 4e-5 in the Maximum element size edit window. Select "Quad" from the Method drop-down list. See Figure 4.42.

NOTE The model default mesh, in the COMSOL Multiphysics software, is the triangular mesh. The triangular mesh is simpler and generates fewer parameters to calculate. However, the quad mesh may be a better mathematical fit to the model for which a solution is sought. The modeler needs to decide the most appropriate choice for the model under consideration.

Boundaries Groups		Mesh	Weak Constr. Color/Style	
Boundary selection	Mesh displacement			
2	Coordinate system:	Global coordinate s	ystem 🗘	
3	Quantity	Value/Expression	Unit	Description
4	O Mesh displacer	nent		
6	🗌 dx	0	m	Mesh displacement, x direction
7	🗌 dv	0	m	Mesh displacement, y direction
Group:	Mesh velocity			
Select by group	Vx 💟	0	m/s	Mesh velocity, x direction
Interior boundaries	□ w	0	m/s	Mesh velocity, y direction

**FIGURE 4.38** Boundary Settings (1, 5), Moving Mesh (ALE): Boundaries window

Boundaries Groups	(=	Mesh	Weak Constr. Color/Style	
Boundary selection	Mesh displacement			
1	Coordinate system:	Tangent and norm	al coord. sys. in deformed mesh 🛟	
3	Quantity	Value/Expression	Unit	Description
4 5	O Mesh displacer	nent		
6	🗌 dn	0	m	Mesh displacement, n direction
7	🖂 di	0	m	Mesh displacement, t direction
Group:	Mesh velocity			
Select by group	M vn	-K*nJ_dc	m/s	Mesh velocity, n direction
Interior boundaries	🗌 vt	0	m/s	Mesh velocity, t direction

FIGURE 4.39 Boundary Settings (3, 4, 6, 7), Moving Mesh (ALE): Boundaries window

For a simple 2D model, the maximum element size value for a particular subdivision can be estimated by dividing the lesser (A < B) dimension by 10 and then testing how the calculated model satisfies the goals of the modeler.

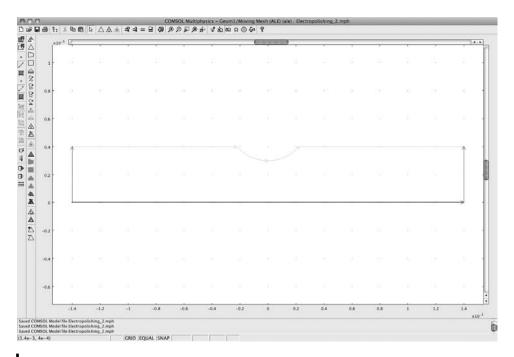
Click the Remesh button, and then click OK. See Figure 4.43. This mesh contains approximately 754 elements.

#### Solving the First Variation on the 2D Electrochemical Polishing Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver. Enter 0:0.5:10 in the Times edit window, as shown in Figure 4.44. This instruction causes the Solver to divide the modeling time-space into 20 equal intervals, over the period from 0 to 10 seconds. Click OK.

Boundaries Groups		Mes	h Weak Constr. Color/Style	
Boundary selection	Mesh displacement Coordinate system:	Global coordinate	system 🗘	3
3	Quantity Mesh displacer	Value/Expression	Unit	Description
6	<b>⊠</b> dx	0	m	Mesh displacement, x direction
7	🗹 dy	0	m	Mesh displacement, y direction
Group:	O Mesh velocity			
Select by group	VX	0	m/s	Mesh velocity, x direction
Interior boundaries	vy	0	m/s	Mesh velocity, y direction

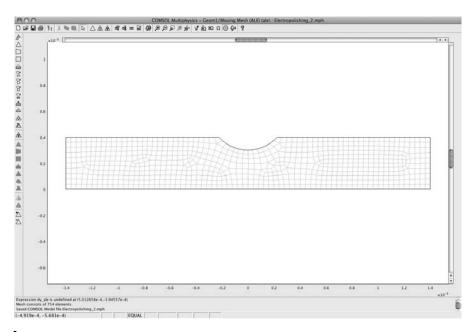
FIGURE 4.40 Boundary Settings (2), Moving Mesh (ALE): Boundaries window



**FIGURE 4.41** Boundary Settings, Moving Mesh (ALE): boundaries organized by color [green = tangent and normal coordinate system in deformed mesh; blue = global (vx = 0); red = global (dx = 0, dy = 0)]

000		Free Mesh Parameters	
	Global	Subdomain Boundary Point Advanced	ОК
	ct by group t Remaining	Subdomain mesh parameters Maximum element size: 4e-5 Element growth rate: Method: Quad \$	Cancel Apply Help
Reset to D	efaults	Remesh Mesh Selected	

**I FIGURE 4.42** 2D electrochemical polishing model Free Mesh Parameters window



**FIGURE 4.43** 2D electrochemical polishing model free mesh (quad)

Analysis:	General	Time Stepping Advanced
Transient		0:0.5:10
Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated Adaptive mesh refinement	Absolute tolerance:	0.0010
	Linear system solver: Dir Preconditioner:	ect (UMFPACK)
	Matrix symmetry:	tomatic
	(	ielp (Apply) (Cancel ) (OK

**FIGURE 4.44** 2D electrochemical polishing model Solver Parameters window

Using the menu bar, select Solve > Solve Problem.

In the process of solving this model, using the Moving Mesh Application Mode (ALE), the modeler may occasionally see a warning about an "inverted mesh element." If the solver continues on to the solution, ignore the warning. Such warnings are normal when using the deformed mesh.

If the model does not continue to a solution and the solver displays numerous warnings, then either there is an error in the model or the modeler needs to use the advanced technique called remesh (not discussed in this book).

#### Postprocessing

Select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked. From the Predefined quantities drop-down list, select Conductive Media DC (dc) > Total current density, norm. See Figure 4.45. Click OK.

	Surface	Contour	Boundary	Arrow Principal
Surface	plot			
		Surface Data	Height Data	
Predefine	d quantities:	Total current o	lensity, norm 🛟	Range
Expressio	n:	normj_dc		Smooth
Unit:		A/m <sup>2</sup>	\$	
Coloring an	d fill			
Coloring:	Interpol	ated 🛟	Fill style:	illed 🛟
Surface colo O Colorr	map:	jet 🔹	Colors: 1024	✓ Color scale

FIGURE 4.45 2D electrochemical polishing model Plot Parameters window

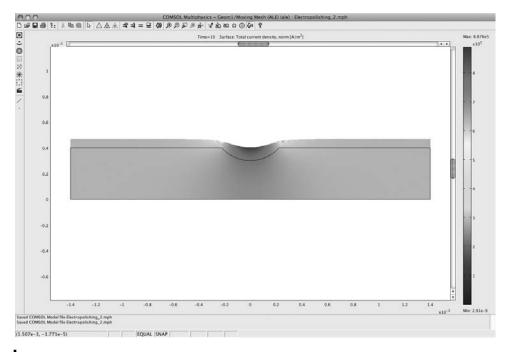


FIGURE 4.46 2D electrochemical polishing model Surface plot window, total normal current density

In Figure 4.46, the model calculation shows that the maximum current density is approximately 8.87e5 A/m<sup>2</sup>, in the region of the asperity (very close to the calculated value in the original model). Figure 4.46 also shows that the normal current density  $(J_n)$  concentrated in the region of the asperity is approximately 1.5 times the normal current density elsewhere on the electrode surface. As a result, the removal rate of the electrode material will be approximately 1.5 times as high.

To see the change in the position of the electrode surface and the relative removal of material from the asperity, select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked. From the Predefined quantities drop-down list, select Moving Mesh (ALE) (ale) > y-displacement. See Figure 4.47.

Click OK. Figure 4.48 shows the displacement of the electrode surface in the *y*-direction (dy\_ale) after 10 seconds of electropolishing.

0		Plot Parameters	
General	Surface	Contour Boundary	Arrow Principal
Surface	plot		
	(	Surface Data Height Data	
	quantities:	y-displacement	(Range)
Expressio	n:	dy_ale	Smooth
Unit:		m 🗘	
Coloring an	d fill		
Coloring:	Interpola	ted 🗘 Fill style: Fil	lled 🗘
Surface colo	r		
Colorn	nap:	et 🗘 Colors: 1024	Color scale
	n color:	Color	
0 0	Contraction C		
			-
		POWEREN.IR	
		Help Apply	Cancel OK
		Current Current	

**FIGURE 4.47** 2D electrochemical polishing model surface Plot Parameters window: Moving Mesh (ALE) (ale), y direction (dy\_ale)

#### **Postprocessing Animation**

This solution to the 2D electrochemical polishing model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.49). Click the Start Animation button. Save this 2D electrochemical polishing model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4\_EP\_2.avi that was supplied with this book.

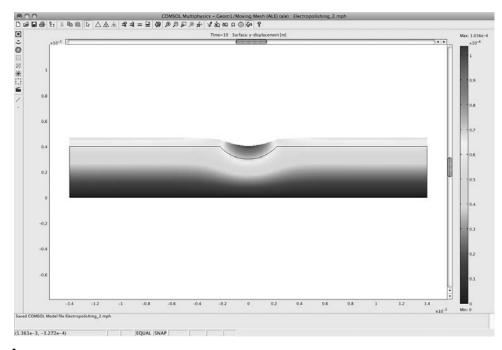


FIGURE 4.48 2D electrochemical polishing model Surface plot window: Moving Mesh (ALE) (ale), y direction (dy\_ale)

# Second Variation on the 2D Electrochemical Polishing Model

Note This model will explore the effect of a change in the shape and number of asperities. In this model, the quad mesh element type will be used, based on the excellent values calculated in the previous modeling solution. Bear in mind that both the model geometry and the model mesh play major roles in the ease of solving any particular problem.

To start building the Electropolishing\_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Deformed Mesh > Moving Mesh (ALE) > Transient analysis. Click the Multiphysics button, and then click the Add button.

Using the Application Modes list, select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. Click the Add button. See Figure 4.50.

Click OK. Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.7; also see Figure 4.51.

Principal Str	eamline Particle Tr	acing Max/Min	Deform Anima
Movie settings File type: Width (in pixels): Height (in pixels): Frames per second: Static / Eigenfunctior Cycle type:	AVI \$	Solutions to use	d output times 🛟
Reverse direction	gs from main window		

I FIGURE 4.49 2D electrochemical polishing model animation Plot Parameters window

# Table 4.7 Constants Edit Window

Name	Expression	Description
К	1.0e-11[m^3/(A*s)]	Coefficient of proportionality

New	Model Library User Mode	els Open Settings
Space dimension:	2D 🛟	Multiphysics
Application Modes	i i	Add Remove
🔻 🚞 COMSOL Multiphy	sics	Geom1 (2D)
Acoustics		🔻 📺 Frame (ale)
Convection a		Moving Mesh (ALE) (ale)
V 💼 Electromagne		Conductive Media DC (dc
Conductiv	ve Media DC	
Magnetos		
	Electromagnetics	
▶ 🚞 Fluid Dynamie		Dependent variables: V
▶ 🚞 Heat Transfer		Application Mode Propertie
Structural Me	chanics 🔹	Add Geometry
PDE Modes	1	
Dependent variables:	V2	Add Frame
		Ruling application mode:
Application mode name:	dc2	Moving Mesh (ALE) (ale)
Element:	Lagrange - Quadra 🗘	Multiphysics

**FIGURE 4.50** 2D Electropolishing\_3 Model Navigator setup

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 2.8 and a height of 0.4. Select "Base: Corner" and set X equal to -1.4 and Y equal to 0 in the Rectangle edit window. See Figure 4.52.

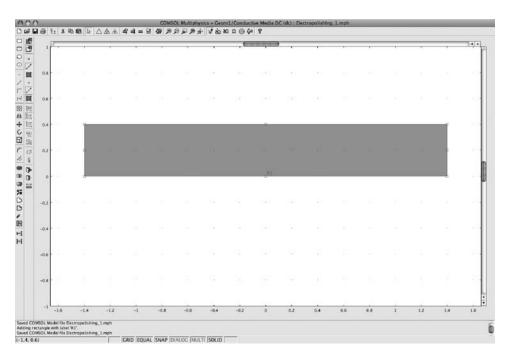
Click the Apply button, and then click OK. See Figure 4.53.

000		Constar	nts
Name	Expression	Value	Description
K	1.0e-11 [m^3/(A*s)]	(1e-11)[m <sup>3</sup> /(s·A)]	Coefficient of proportionality
¢ 8		Help	Apply Cancel OK

**FIGURE 4.51** 2D Electropolishing\_3 model Constants edit window

000		Recta	ingle		
Size			Rotat	tion angle-	
Width:	2.8		α:	0	(degrees)
Height:	0.4				
Position			L 		
Base:	Corner	\$	Style:	Solid	\$
<b>X</b> :	-1.4		Name:	R1	
Y:	0				
	Help	Apply	$) \subset$	Cancel	ОК

**FIGURE 4.52** 2D Electropolishing\_3 model Rectangle edit window



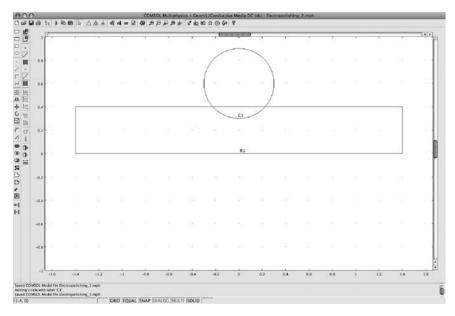
**FIGURE 4.53** 2D Electropolishing\_3 model electrolyte rectangle

Size		Rotation angle
Radius:	0.3	α: 0 (degrees)
Positior	1	
Base:	Center	Style: Solid 🗘
x:	0	Name: C1
	0.6	

**FIGURE 4.54** 2D Electropolishing\_3 model Circle edit window

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.3. Select "Base: Center" and set x equal to 0 and y equal to 0.6 in the Circle edit window. See Figure 4.54.

Click OK. See Figure 4.55.



**FIGURE 4.55** 2D Electropolishing\_3 model rectangle and circle

Size		Rotat	ion angle	
A-semiaxes:	0.3	α:	0	(degrees)
B-semiaxes:	0.1			
Position				
Base:	Center	\$ Style:	Solid	\$
	-1.0	Name:	E1	
X:	-1.0			

FIGURE 4.56 2D Electropolishing\_3 model Ellipse edit window

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter 0.3 in the A-semiaxes edit window and 0.1 in the B-semiaxes edit window. Select "Base: Center" and set X equal to -1.0 and Y equal to 0.4 in the X and Y edit windows. See Figure 4.56.

Click OK. See Figure 4.57.

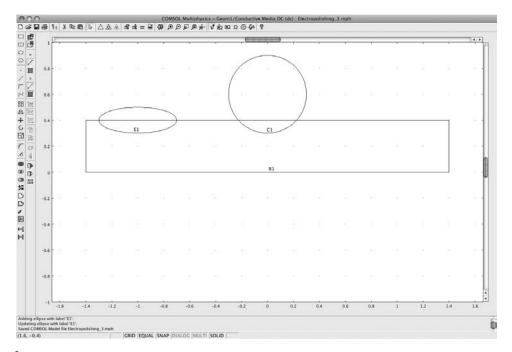


FIGURE 4.57 2D Electropolishing\_3 model (C1, E1, R1)

0	O Pa	aste
Dis	placements	
<b>X</b> :	2.0	
Υ:	0	Cancel

**FIGURE 4.58** 2D Electropolishing\_3 model Paste edit window

Select the text "E1." Using the menu bar, select Edit > Copy. Using the menu bar, select Edit > Paste. Enter 2.0 in the X: Displacements edit window. See Figure 4.58.

Click OK. See Figure 4.59.

Using the menu bar, select Draw > Create Composite Object. Enter R1-C1-E1-E2 in the Set formula edit window. See Figure 4.60.

NOTE To obtain the desired difference response, the modeler needs to key in the requested R1-C1-E1-E2 information in the edit window, rather than clicking on items in the Object selection window.

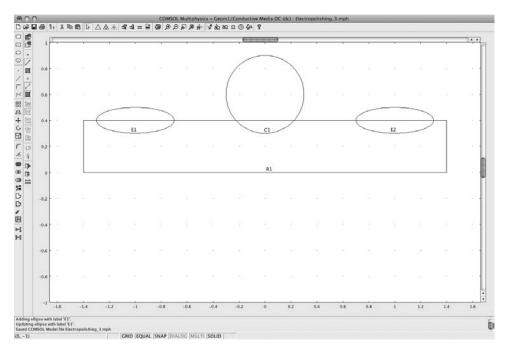
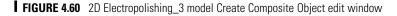


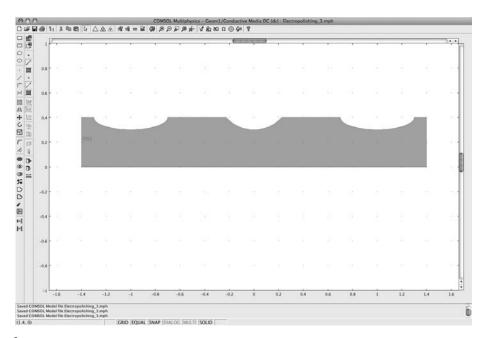
FIGURE 4.59 2D Electropolishing\_3 model (C1, E1, E2, R1)

000 0	reate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: R1 C1	Set formula: R1-C1-E1-E2
E1 E2	Repair 1.0E-4



Click OK. See Figure 4.61.

The model geometry, as presently scaled, is 1.4 meters in length and 0.4 meter in height. Electropolishing is typically applied as the final finishing (smoothing) step in a precision fabrication process (e.g., metallographic samples, vacuum chambers). Thus the model geometry will need to be reduced in scale to emulate reality.



**FIGURE 4.61** 2D Electropolishing\_3 model electrode with asperities

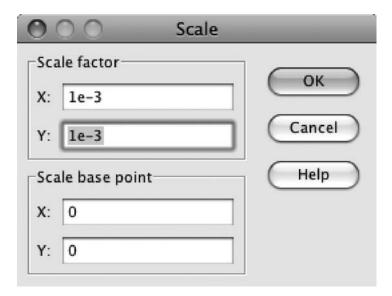
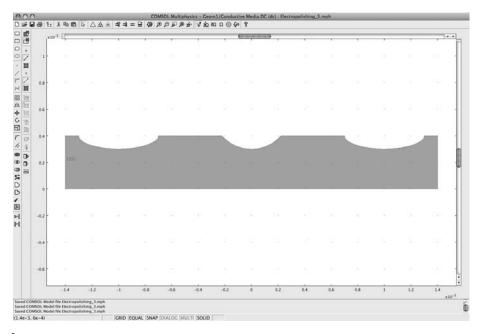


FIGURE 4.62 2D Electropolishing\_3 model Scale edit window

Click on the text "CO1." Click the Scale button on the Draw toolbar. Enter 1e-3 in both the X and Y edit windows. See Figure 4.62.

Click OK, and then click the Zoom Extents button on the menu bar. See Figure 4.63.



**FIGURE 4.63** 2D Electropolishing\_3 model scaled electrolyte/electrode geometry

Subdomains Groups			lement C	olor
1	Material properties and Library material:		.oad )	
Group: •	Quantity J° Qj d Conductivity relation: σ	Value/Expression 0 0 0 0 0 1 Conductivity 10	Unit A/m <sup>2</sup> A/m <sup>3</sup> m	Description External current density Current source Thickness Electric conductivity

FIGURE 4.64 Subdomain Settings window

### **Physics Subdomain Settings: Conductive Media DC**

Having established the 2D geometry for the electrochemical polishing model (a rectangle with negative asperities on the upper surface), the next step is to define the fundamental physics conditions. Using the menu bar, select Multiphysics > Conductive Media DC. Next, using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). Enter 10 in the Electric conductivity ( $\sigma$ ) edit window. See Figure 4.64. Click OK.

### Physics Boundary Settings: Conductive Media DC

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 4.8. See Figures 4.65, 4.66, and 4.67.

Click OK. See Figure 4.68.

#### Table 4.8 Boundary Settings, Conductive Media DC Window

Boundary	<b>Boundary Condition</b>	Value/Expression
1, 7	Electric insulation	—
3–6, 8–13	Electric potential	30
2	Ground	—

Boundaries Groups	Conditions     Color/Style       Boundary sources and constraints     Library material:       Library material:
8	

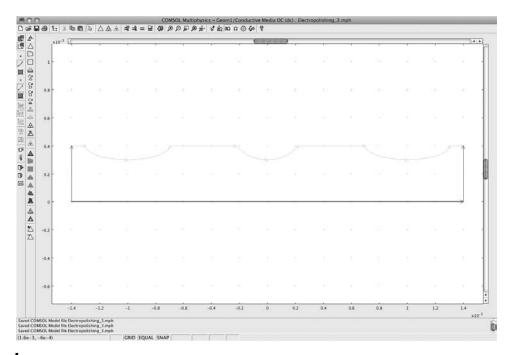
**FIGURE 4.65** Boundary Settings (1, 7), Conductive Media DC: boundaries set

Boundaries Groups	Conditions       Color/Style         Boundary sources and constraints       Library material:         Library material:
-------------------	---

I FIGURE 4.66 Boundary Settings (3–6, 8–13), Conductive Media DC: boundaries set

000	Boundary Settings - Conductive Media DC (dc)
Equation V = 0	
Boundaries Group:	Conditions Color/Style  Boundary sources and constraints Library material:  Boundary condition:  Ground  Cround  Croun
Interior boundar	Help Apply Cancel OK

FIGURE 4.67 Boundary Settings (2), Conductive Media DC: boundary set



**FIGURE 4.68** Boundary Settings (1, 7 = blue; 3–6, 8–13 = green; 2 = red), Conductive Media DC: boundaries set

Boundary	Coordinate	Boundary Condition	Value/Expression
1, 7	Global	Mesh velocity	vx = 0
3–6, 8–13	Tangent and normal	Mesh velocity	$vn = -K*nJ_dc$
	Deformed mesh		
2	Global	Mesh displacement	dx = 0, dy = 0

Table 4.9 Boundary Settings, Moving Mesh (ALE) Window

## Boundary Settings: Moving Mesh (ALE)

Using the menu bar, select Multiphysics > Moving Mesh (ALE). Next, using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value in the edit windows as indicated in Table 4.9. See Figures 4.69, 4.70, and 4.71.

Click OK. See Figure 4.72.

## **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 1 in the Subdomain selection window. Enter 4e-5 in the Maximum element size edit window. Select "Quad" from the Method drop-down list. See Figure 4.73.

Boundaries Groups		Mesh	Weak Constr. Color/Style	
Boundary selection	Mesh displacement Coordinate system:			
2		Global coordinate : Value/Expression	Unit	
4	Quantity Mesh displacer	and the second second second second	Unit	Description
6	🗌 dx	0	m	Mesh displacement, x directio
8 7	🗌 dy	0	m	Mesh displacement, y direction
Group:	Mesh velocity			
Select by group	VX 🕅	0	m/s	Mesh velocity, x direction
Interior boundaries	🗆 vy	0	m/s	Mesh velocity, y direction

**FIGURE 4.69** Boundary Settings (1, 7), Moving Mesh (ALE): boundaries set

oundary selection	Mesh displacement			
î î	Coordinate system:	Tangent and norm	al coord. sys. in deformed mesh 🛟	
	Quantity	Value/Expression	Unit	Description
24	O Mesh displacer	nent		
0 1	🗌 dn	0	m	Mesh displacement, n direction
2 4	🗌 di	0	m	Mesh displacement, t direction
oup:	Mesh velocity			
Select by group	🗹 vn	-K*nJ_dc	m/s	Mesh velocity, n direction
Interior boundaries	🗌 vt	0	m/s	Mesh velocity, t direction

FIGURE 4.70 Boundary Settings (3–6, 8–13), Moving Mesh (ALE): boundaries set

Click the Remesh button, and then click OK. See Figure 4.74. This mesh contains approximately 675 elements.

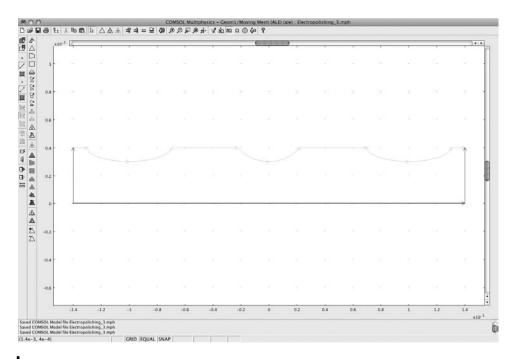
#### Solving the Second Variation on the 2D Electrochemical Polishing Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver. Enter 0:0.5:10 in the Times edit window, as shown in Figure 4.75. This instruction causes the Solver to divide the modeling time-space into 20 equal intervals, over the period from 0 to 10 seconds. Click OK.

Using the menu bar, select Solve > Solve Problem.

Boundary selection	Mesh displacement			
1	Coordinate system:	Global coordinate	system 🛟	1
3	Quantity	Value/Expression	Unit	Description
4 5	Mesh displacer	nent		
6	🗹 dx	0	m	Mesh displacement, x direction
8 4	🗹 dy	0	m	Mesh displacement, y direction
iroup:	Mesh velocity			
Select by group	U vx	0	m/s	Mesh velocity, x direction
Interior boundaries	🗌 vy	0	m/s	Mesh velocity, y direction

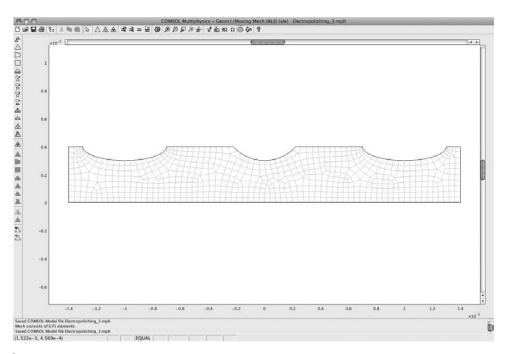
FIGURE 4.71 Boundary Settings (2), Moving Mesh (ALE): boundary set



**FIGURE 4.72** Boundary Settings, Moving Mesh (ALE): boundaries organized by color (1, 7 = blue; 3–6, 8–13 = green; 2 = red)

Maximum element size: 4e-5 Element growth rate:	Global	Subdomain Boundary Point Ad	vanced
(Select Remaining)	1	Maximum element size: 4e-5 Element growth rate: Method: Quad	
Select Meshed	Select Remaining		

**FIGURE 4.73** 2D Electrochemical polishing model Free Mesh Parameters window



**FIGURE 4.74** 2D Electrochemical polishing model free mesh (quad)

Analysis:	Solver Paramet	
Transient	General	Time Stepping Advanced
Auto select solver	Time stepping Times:	0:0.5:10
tationary ime dependent igenvalue	Relative tolerance: Absolute tolerance:	0.01
Parametric Stationary segregated Parametric segregated	Linear system solver	ect (UMFPACK)
Adaptive mesh refinemen	Preconditioner:	
		Settings
	Matrix symmetry:	omatic
	CH	elp Apply Cancel OK

**FIGURE 4.75** 2D Electrochemical polishing model Solver Parameters window

In the process of solving this model, using the Moving Mesh Application Mode (ALE), the modeler may occasionally see a warning about an "inverted mesh element." If the solver continues on to a solution, ignore the warning. Such warnings are normal when using the deformed mesh.

If the model does not continue to a solution and the solver displays numerous warnings, then either there is an error in the model or the modeler needs to use the advanced technique called remesh (not discussed in this book).

#### Postprocessing

Select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked. From the Predefined quantities drop-down list, select Conductive Media DC (dc) > Total current density, norm. See Figure 4.76. Click OK.

Conoral		-	Parameters		
General	Surface	Contour	Boundary	Arrow	Principal
Surface	plot				
		Surface Data	Height Data		
Predefined	d quantities:	Total current d	ensity, norm 🛟	Range.	
Expressio	n:	normJ_dc		Smoot	'n
Unit:		A/m <sup>2</sup>	\$	)	
Coloring an			-		
Coloring:	Interpo	lated 🗘	Fill style:	Filled	
Surface colo				7.5	
	6	7		1.4	
<ul> <li>Colorn</li> </ul>	nap:	jet 🗘	Colors: 1024	Color :	scale
O Uniform		Color	Colors: 1024	Color:	scale
-			Colors: 1024	Color:	scale
-			Colors: 1024	Color	scale
-			Colors: 1024	Color :	scale
-			Colors: 1024	Color	scale
-			Colors: 1024	Color :	scale
-			Colors: 1024	Color :	scale
-			Colors: 1024	Color :	scale
-			Colors: 1024	Color :	scale
-			Colors: 1024	Color	scale
-			Colors: 1024	Color	scale

FIGURE 4.76 2D electrochemical polishing model surface Plot Parameters window, total normal current density

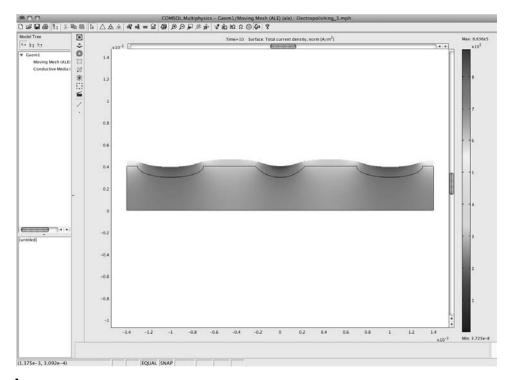


FIGURE 4.77 2D electrochemical polishing model Surface plot window, total normal current density

In Figure 4.77, the model calculation shows that the maximum current density is approximately 8.84e5 A/m<sup>2</sup>, in the region of the asperities. Figure 4.77 also shows that the normal current density ( $J_n$ ) concentrated in the region of the asperities is approximately 1.5 times the normal current density elsewhere on the electrode surface. As a result, the removal rate of the electrode material will be approximately 1.5 times as high.

To see the change in the position of the electrode surface and the relative removal of material from the asperities, select Postprocessing > Plot Parameters. Click the Surface tab, and verify that the Surface plot check box is checked. From the Predefined quantities drop-down list, select Moving Mesh (ALE) (ale) > y-displacement. See Figure 4.78.

Click OK. Figure 4.79 shows the displacement of the electrode surface in the *y*-direction (dy\_ale) after 10 seconds of electropolishing.

#### **Postprocessing Animation**

This solution to the 2D electrochemical polishing model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, Click the Animate tab.

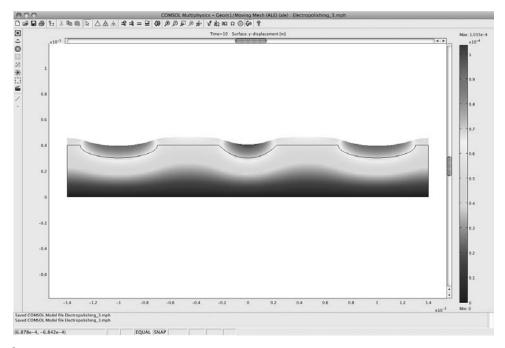
Surface plot		
	Surface Data Height Data	
Predefined quantities:	y-displacement Range	
Expression:	dy_ale Smooth	
Unit:	m	
Coloring: Interpo	olated 🗘 Fill style: Filled 🛟	
Surface color Colormap: ( Uniform color:	jet Colors: 1024 Color scale	
O Colormap: (		
• Colormap:		

**FIGURE 4.78** 2D electrochemical polishing model surface Plot Parameters window: Moving Mesh (ALE) (ale), *y*-direction (dy\_ale)

On the Animate page, select all the solutions in the Stored output times window (see Figure 4.80). Click the Start Animation button. Save this 2D electrochemical polishing model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4\_EP\_3.avi that was supplied with this book.

# 2D Electrochemical Polishing Models: Summary and Conclusions

The models presented in this section have introduced the following new concepts: two-dimensional modeling (2D), deformed mesh—Moving Mesh (ALE), transient analysis, Conductive Media DC, vector dot product current (K\*nJ\_dc), triangular mesh, free mesh parameters, subdomain mesh, maximum element size, and quadrilateral



**FIGURE 4.79** 2D electrochemical polishing model Surface plot window: Moving Mesh (ALE) (ale), *y*-direction (dy\_ale)

mesh (quad). The 2D electrochemical polishing model is a powerful tool that can be used to model surface smoothing for diverse projects (e.g., microscope samples, precision metal parts, medical equipment and tools, large and small metal drums, thin analytical samples, vacuum chambers). A comparison of the calculated results for the three electropolishing models is shown in Table 4.10.

The differences between the calculations for the tested models are in the range of a few percentage points. It is left to the modeler to explore other differences between the models by varying the parameters, as suggested in the exercises at the end of this chapter.

Model	Asperities	Mesh	Peak J <sub>n</sub>	$\Delta J_n(\%)$	dy	∆ <i>dy</i> (%)
EP_1	1	Triangular	9.12e5	—	1.08e-4	_
EP_2	1	Quad	8.87e5	~2.7	1.04e-4	~3.7
EP_3	3	Quad	8.83e5	~3.2	1.04e-4	~3.7

Principal	Streamline	Particle Tracing	Max/Min	Deform	Anima
Movie settings		So	lutions to use		
File type.	AVI	\$ Se	lect via. Stor	ed output tim	es 🛊
Width (in pixels):	640	0			õ
Height (in pixels)	480	0.1			
Frames per seco	nd: 10	1.			Ψ
	Adva	nced 2.			
Static / Eigenfund	rtion animation	3			
Cycle type:	Full harm	onic 🗘 4	5		+
			nes:		
Number of frame					
Reverse direc	tion				
Reverse direc	tion				

**FIGURE 4.80** 2D electrochemical polishing model animation Plot Parameters window

# 2D Hall Effect Model Considerations

In 1827, Georg Ohm published<sup>9</sup> his now fundamental and famous Ohm's law:

$$I = \frac{V}{R} \tag{4.4}$$

where

$$I =$$
current in amperes

V = potential difference in volts

R = resistance in ohms

See Figure 4.81.

As useful as Ohm's law is, it is basically phenomenological. To more fully understand conduction in homogeneous, isotropic solid materials, the calculations

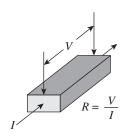


FIGURE 4.81 Ohm's law

need to be expanded until they reflect the behavior (motion) of the fundamental charged particles (electrons, holes).

Note In solid materials (e.g., metals, semiconductors), there are three potential mobile carriers of charge: electrons (-), holes (+), and ions (charge sign can be either + or -, depending on the type of ion). Ions in a solid typically have a very low mobility (pinned in position) and, therefore, contribute little to the observed current flow in most solids. Ion flow will not be considered here.

In metals, due to the underlying physical and electronic structure, electrons are the only carrier. In semiconductors (e.g., Si, Ge, GaAs, InP), either electrons or holes (the absence of an electron) can exist as the primary carrier types. The density of each carrier type (electrons, holes) is determined by the electronic structure of the host material (e.g., Si, Ge, SiGe) and the density and distribution of any foreign impurity atoms (e.g., As, P, N, Al) within the host solid material. For further information on the nature of solids and the behavior of impurity atoms in a host matrix, see works by Kittel<sup>10</sup> and Sze.<sup>11</sup>

The resistance of a homogeneous, isotropic solid material R is defined as follows:

$$R = \frac{\rho L}{A} \tag{4.5}$$

where

 $\rho$  = resistivity in ohm-meters ( $\Omega$ -m)

L =length of sample in meters (m)

A =cross-sectional area of sample in meters squared (m<sup>2</sup>)

See Figure 4.82.

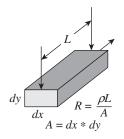


FIGURE 4.82 Resistance

The resistivity of a homogeneous, isotropic solid material is defined as follows:<sup>12</sup>

$$\sigma \equiv \frac{1}{\rho} = n_{\rm e} |\mathbf{e}| \boldsymbol{\mu}_{\rm e} + n_{\rm h} |\mathbf{e}| \boldsymbol{\mu}_{\rm h}$$
(4.6)

where

 $\rho$  = resistivity ohm-meters ( $\Omega$ -m)

 $\sigma$  = conductivity in siemens per meter (S/m)

 $n_{\rm e}$  = electron density in electrons per cubic meter (N<sub>e</sub>/m<sup>3</sup>)

 $n_{\rm h}$  = hole density in holes per cubic meter (N<sub>h</sub>/m<sup>3</sup>)

|e| = absolute value of the charge on an electron (hole) in coulombs (C)

 $\mu_{\rm e}$  = electron mobility in meters squared per volt-second (m<sup>2</sup>/(V\*s))

 $\mu_{\rm h}$  = hole mobility in meters squared per volt-second (m<sup>2</sup>/(V\*s))

The Hall effect<sup>13</sup> was discovered by Edwin Hall in 1879<sup>14</sup> through measurements on the behavior of currents in thin gold foils, in the presence of a magnetic field. The magnetic field introduced into the current flow region of the solid in the Hall effect measurements effectively adds an anisotropic term into the conductivity of a nominally homogeneous, isotropic solid material. The anisotropic conductivity is caused by the magnetic field through the Lorentz force.<sup>15</sup> The Lorentz force produces a proportional, differential voltage/charge accumulation between two surfaces or edges of a conducting material orthogonal to the current flow.

The Lorentz force is

$$\mathbf{F} = q \left( \mathbf{E} + \left( \mathbf{v} \times \mathbf{B} \right) \right) \tag{4.7}$$

where

 $\mathbf{F}$  = force vector on the charged particle (electron and/or hole)

q = charge on the particle (electron and/or hole)

 $\mathbf{E} = \text{electric field vector}$ 

 $\mathbf{v}$  = instantaneous velocity vector of the particle

 $\mathbf{B}$  = magnetic field vector

The Hall voltage<sup>16</sup> is

$$V_{\rm H} = \frac{R_{\rm H} * I * B}{t} \tag{4.8}$$

where  $V_{\rm H}$  = Hall voltage

 $R_{\rm H}$  = Hall coefficient

- I = current
- B = magnetic field
- t = thickness of sample

The Hall coefficient  $(R_{\rm H})$  is

$$R_{\rm H} = -\frac{r}{n_{\rm e} \rm e} \tag{4.9}$$

where

 $R_{\rm H}$  = Hall coefficient  $r = 1 \le \times \le 2$   $n_{\rm e}$  = density of electrons e = charge on the electron

Note: In the Hall effect models presented here, it is assumed that r = 1. That assumption is a valid first approximation. For applied development models, the modeler will need to determine experimentally the best approximation for the value of r for the particular material and physical conditions being modeled.

For example, in the case that the charge carrier is a "hole," the minus sign (-) in the equation for the Hall coefficient changes to a plus sign (+). In the case of mixed electron/hole flow,  $R_{\rm H}$  can become zero.

The differential voltage/charge accumulation—Hall voltage ( $V_{\rm H}$ )—that results from the Lorentz force interaction between any currents (electron and/or hole) flowing through that conducting material and the local magnetic field is shown in Figure 4.83.

As can be seen from the introductory material, depending on the characteristics of the material being modeled, the calculation of the Hall effect can be very complex. The Hall coefficient ( $R_{\rm H}$ ) varies for different materials and has a predominant functional dependence that involves temperature, carrier type, carrier concentration, carrier mobility, carrier lifetime, and carrier velocity. In a dual-carrier system, such as semiconducting materials (electrons and holes), under the proper conditions,  $R_{\rm H}$  can become equal to zero. Semiconductor sensors, however, are among the most sensitive magnetic field Hall sensors currently manufactured.

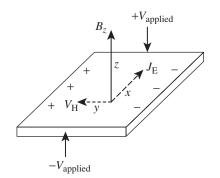


FIGURE 4.83 Hall effect sensor geometry, electron flow

Hall effect sensors are widely available in a large number of geometric configurations. They are typically applied in sensing fluid flow, rotating or linear motion, proximity, current, pressure, and orientation. In the 2D models presented in the remainder of this chapter, several simplifying assumptions will be made that allow the basic physics principles to be demonstrated without excessive complexity.

Owing to the underlying complexity of the Hall effect, the models in this section of Chapter 4 require the use of either the AC/DC Module or the MEMS Module, in addition to the basic COMSOL Multiphysics software. In the first model, only a single carrier conduction system (electrons) will be employed. For ease of modeling, it will be assumed that the system is quasi-static. This model introduces the COMSOL modeling concepts of point constraints and floating contacts.<sup>17</sup>

# 2D Hall Effect Model

To start building the Hall\_Effect\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Conductive Media DC. Click the Multiphysics button, and then click the Add button. See Figure 4.84.

Model Library User Models	
•	Open Settings
2D 🗘	Multiphysics
rsics e Media DC tics tics, Generalized tatics , Electric , Magnetic , Electromagnetic	Add Remove
ninery	Add Geometry
V2	Add Frame
emdc2	Ruling application mode: Conductive Media DC (emdc)
Lagrange - Quadra 🛊	Multiphysics
	e Media DC tics tics, Generalized tatics Electric Magnetic Electromagnetic inery nal Interaction

FIGURE 4.84 Multiphysics Model Navigator window

Properties		
Default element type:	Lagrange – Quadratic	ę
Weak constraints:	On	¢
Constraint type:	Non-ideal	\$

**FIGURE 4.85** Application Mode Properties window

Click the Application Mode Properties button. Select "On" from the Weak constraints pull-down list. Select "Non-ideal" from the Constraint type pull-down list. See Figure 4.85. Click OK.

### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.11; also see Figure 4.86. Click OK.

# **2D Hall Effect Geometry**

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.8e-2 and height of 6e-3. Select "Base: Corner" x equal to -9e-3 and y equal to -3e-3 in the Rectangle edit window. See Figure 4.87.

Click OK, and then click the Zoom Extents button. See Figure 4.88.

Vame	Expression	Description
igma0	1.04e3[S/m]	Silicon conductivity
lh	1.25e-4[m^3/C]	Hall coefficient
z	0.1[T]	Magnetic field
oeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	Conductivity anisotropy 2
D	5.0[V]	Applied voltage
_Si	1.0e-3[m]	Silicon thickness
peff1	sigma0*Rh*Bz	Conductivity anisotropy 1
11	coeff0	Conductivity matrix term 11
12	coeff0*coeff1	Conductivity matrix term 12
:21	-coeff0*coeff1	Conductivity matrix term 21
22	coeff0	Conductivity matrix term 22

Name	Expression	Value	Description
sigma0	1.04e3[S/m]	1040[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	$(1.25e-4)[m^3/(s \cdot A)]$	Hall coefficient
Bz	0.1[T]	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	1039.82427[S/m]	Conductivity anisotropy 2
V0	5.0[V]	5[V]	Applied voltage
t_Si	1.0e-3[m]	0.001[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	0.013[1]	Conductivity anisotropy 1
s11	coeff0	1039.82427[S/m]	Conductivity matrix term 11
s12	coeff0*coeff1	13.517716[S/m]	Conductivity matrix term 12
s21	-coeff0*coeff1	-13.517716[S/m]	Conductivity matrix term 21
s22	coeff0	1039.82427[S/m]	Conductivity matrix term 22

FIGURE 4.86 2D Hall\_Effect\_1 model Constants edit window

In this model, points will be added to the boundary of the rectangle to define the location of the edges of the floating contacts. Using the menu bar, select Draw > Specify Objects > Point. In the Draw > Specify Objects > Point edit window, individually create each of the points shown in Table 4.12 by selecting the window, entering the data, and then clicking OK. The final rectangle with all four points is shown in Figure 4.89.

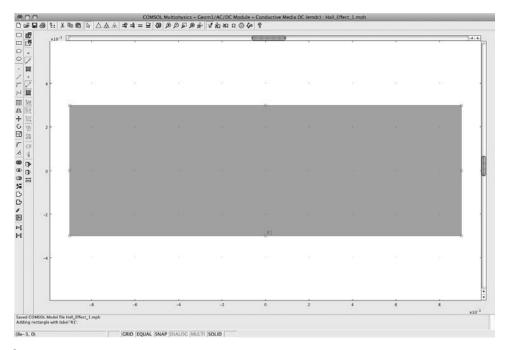
NOTE The points are added to the boundary of the rectangle so that the edges of the floating contacts are precisely defined.

#### 2D Hall Effect Subdomain Settings

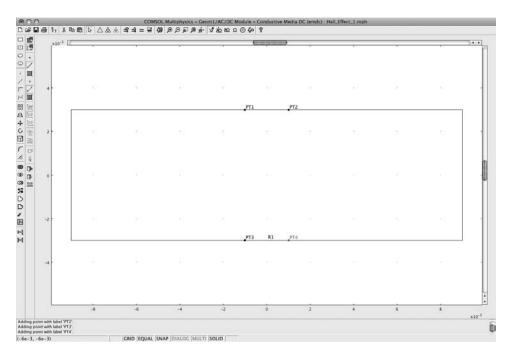
Using the menu bar, select Physics > Subdomain Settings > Subdomain 1 (the only choice). Enter t\_Si in the d (Thickness) edit window. Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Click in the Electric conductivity

Size		Rotat	ion angle-	
Width:	1.8e-2	α:	0	(degrees)
Height:	6e-3			
Position				
Base:	Corner	\$ Style:	Solid	¢
x:	-9e-3	Name:	R1	
y:	-3e-3			

**FIGURE 4.87** 2D Hall\_Effect\_1 model Rectangle edit window



**FIGURE 4.88** 2D Hall\_Effect\_1 model rectangle geometry



**FIGURE 4.89** 2D Hall\_Effect\_1 model rectangle geometry with points

Point Number	x Location	y Location
1	-1e-3	3e-3
2	1e-3	3e-3
3	-1e-3	-3e-3
4	1e-3	-3e-3

### Table 4.12Points Edit Window

### Table 4.13 Matrix Elements Edit Window

Matrix Element Number	Value
11	s11
12	s12
21	s21
22	s22

edit window. Select "Anisotropic-full" from the Conductivity type pull-down list. Enter the matrix elements as shown in Table 4.13; see Figure 4.90.

NOTE These matrix elements define the anisotropic coupling of the magnetic field and the current flowing in the silicon sample.

Close the Conductivity Matrix edit window by clicking on the Subdomain Settings window. After the Conductivity Matrix window closes, the matrix elements will be as shown in the Conductivity edit window on the Subdomain Settings page in Figure 4.91. Click OK.

s11	s12	
s21	s22	
Anisotropic - f		

**FIGURE 4.90** 2D Hall\_Effect\_1 model conductivity matrix elements

# Table 4.14 Boundary Settings

Boundary Number	Condition	Group Index	Source Current/Potential	Figure
1	Ground	_	_	4.92
2, 3, 6, 7	Electric insulation	—	—	4.93
4	Floating potential	2	0	4.94
5	Floating potential	1	0	4.95
8	Electric potential	_	VO	4.96

# **2D Hall Effect Boundary Settings**

Using the menu bar, select Physics > Boundary Settings. Enter the boundary settings as shown in Table 4.14. See Figures 4.92, 4.93, 4.94, 4.95, and 4.96.

**NOTE** The addition of the group index designation decouples the two floating contacts from each other. Failure to insert a different group index number for each floating contact couples (mathematically short-circuits) the contacts together.

Click the Weak Constr. tab. Verify that the Use weak constraints check box is checked. See Figure 4.97.

Subdomains Groups	Physic		lnit Ele	ment Color
ubdomain selection	Material properties and Library material:		oad	
iroup:	Quantity J <sup>e</sup> Q <sub>j</sub> d Conductivity relation: σ	Value/Expression          0       0         0       0         t_Si       Conductivity         \$11\$ \$21\$ \$12\$ \$22	Unit A/m <sup>2</sup> A/m <sup>3</sup> m	Description External current density Current source Thickness Electric conductivity

**FIGURE 4.91** 2D Hall\_Effect\_1 model Subdomain Settings

= 0	
Boundaries Groups Boundary selection	Conditions       Port       Weak Constr.       Color/Style         Boundary sources and constraints       Library material: <ul> <li>Load</li> <li>Boundary condition:</li> <li>Ground</li> <li>Cround</li> <li>Cround</li> <li>Cround</li> <li>Cround</li> <li>Cround</li> <li>Cround</li> <li>Cround</li> <li>Cround</li> <li>Cround</li> <li>Cond</li> <li>Cround</li> <l< th=""></l<></ul>

**FIGURE 4.92** 2D Hall\_Effect\_1 model Boundary Settings (1)

Boundaries Groups	Conditions Port Weak Constr. Color/Style
Boundary selection	Boundary sources and constraints Library material:

**FIGURE 4.93** 2D Hall\_Effect\_1 model Boundary Settings (2, 3, 6, 7)

Boundaries Groups	Condition	ns Port Weak Cons	tr. Color/Style
bundary selection	Boundary sources and con Library material: Boundary condition: Quantity	Load.	Unit Description           1         Group index           A         Source current
Select by group			

**FIGURE 4.94** 2D Hall\_Effect\_1 model Boundary Settings (4)

Boundaries Groups	Cond	itions Port Weak Con	str. Color/Style
Boundary selection	Boundary sources and Library material: Boundary condit Quantity	¢ Load	Unit Description          1       Group index         A       Source current

**FIGURE 4.95** 2D Hall\_Effect\_1 model Boundary Settings (5)

Boundaries Groups	Co	nditions Port Weak Con	str. Color/Style
oundary selection	Boundary sources a Library material: Boundary con Quantity V <sub>0</sub>	toad	Unit Description V Electric potential

**FIGURE 4.96** 2D Hall\_Effect\_1 model Boundary Settings (8)

Boundaries Groups		Conditions Port Weak Constr. Colo	r/Style
Boundary selection	Predefined e Coefficient wcshape wcgporder	Ik constraints	<b>Description</b> Shape functions Integration order
Group: 🔅 💠	wcinit	0	Initial value

I FIGURE 4.97 2D Hall\_Effect\_1 model Boundary Settings, Weak Constr. page

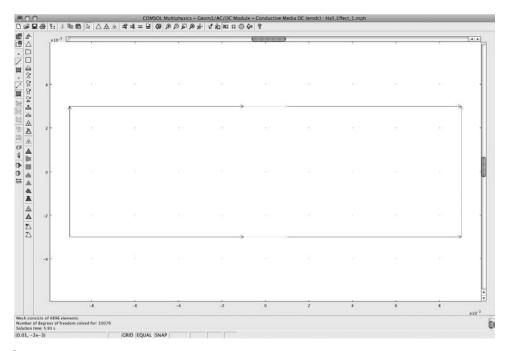


FIGURE 4.98 2D Hall\_Effect\_1 model boundary settings, final configuration

Click OK. The final configuration of the boundary settings is shown in Figure 4.98.

# **2D Hall Effect Mesh Generation**

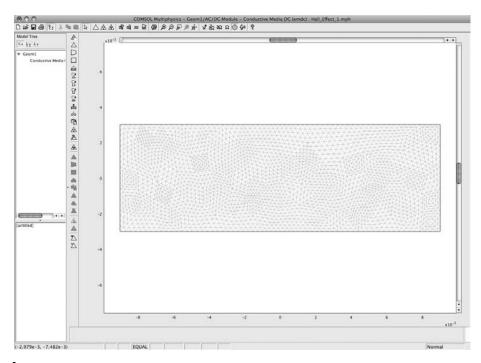
Using the menu bar, select Mesh > Initialize Mesh. Click the Refine Mesh button twice. See Figure 4.99.

# Solving the 2D Hall Effect Model

Using the menu bar, select Solve > Solver Parameters. In the Solver selection window, select "Parametric." In the Parameter names edit window, enter Bz. In the Parameter values edit window, enter 0:0.1:2.0. See Figure 4.100. Click OK.

The Parametric Solver is chosen, in this case, so that the modeler can solve the Hall\_Effect\_1 model quasi-statically over a range of Bz. This allows the modeler to see solutions for a wide range of magnetic field values.

Using the menu bar, select Solve > Solve Problem.



**FIGURE 4.99** 2D Hall\_Effect\_1 model mesh

#### Postprocessing

The default plot is a 2D surface plot of the voltage distribution at the highest value of the magnetic field (Bz = 2 tesla). See Figure 4.101.

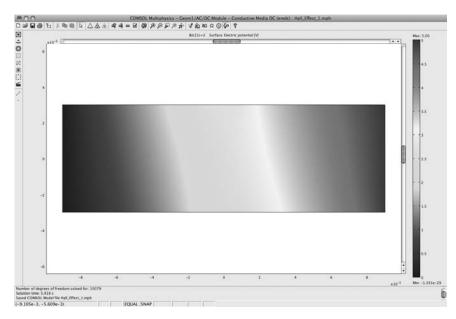
More detailed information is displayed by adding contour lines. Using the menu bar, select Postprocessing > Plot Parameters. Click the Contour tab. Place a check mark in the Contour plot check box. Click the Uniform color radio button. Click the Color select button, and select "Black." Click OK. See Figure 4.102.

The Hall effect voltage  $(V_{\rm H})$  can be seen as the voltage difference (color difference) between the top electrode and the bottom electrode, as shown in Figure 4.103.

There are two methods by which the voltage difference between the upper and lower surfaces can be determined in Figure 4.103. The first is by the color difference, as indicated by the color bar on the right side of the plot. The second is by the incremental position of the contour lines. If the voltage is constant in the vertical direction, the contour line will be straight and vertical. If the voltage changes, that change is reflected in the shape of the contour line.

000	Solver Parameters
Analysis types	General Parametric Stationary Adaptive Optimization/Sensitivity Advanced
Auto select solver  Auto sol	General       Parametric       Stationary       Adaptive       Optimization/Sensitivity       Advanced         Parameters       Parameter names:       Bz       Image: Constraint of the sensitivity       Advanced         Parameter names:       Bz       Image: Constraint of the sensitivity       Load Parameter Values From File         Linear system solver       Direct (UMFPACK)       Image: Constraint of the sensitivity       Settings         Preconditioner.       Image: Constraint of the sensitivity       Automatic       Image: Constraint of the sensitivity         Matrix symmetry:       Automatic       Image: Constraint of the sensitivity       Image: Constraint of the sensitivity
Plot Settings	(Help) (Apply) (Cancel) (OK

**FIGURE 4.100** 2D Hall\_Effect\_1 model Solver Parameters window



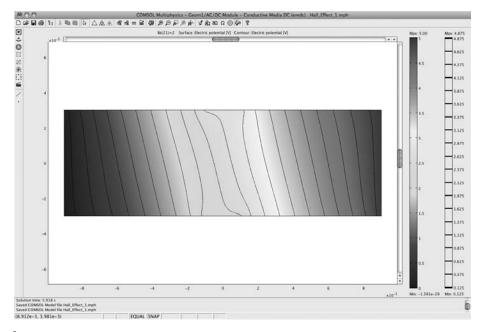


Contour plot Contour Data Height Data Color Data Predefined quantities: Electric potential Expression: V Unit: V Contour levels Number of levels Vector with isolevels evels:  20 Labels	Contour plot  Contour Data Height Data Color Data  Predefined quantities: Electric potential  Freedefined quantities: Electric potential  Contour levels  Number of levels  Vector with isolevels  Levels:    Vector with isolevels  Contour color  Contour color	0		Plot Parame			
Contour Data Height Data Color Data     Predefined quantities: Electric potential Smooth   Expression: V Recover   Unit: V Recover   Contour levels   Number of levels Vector with isolevels   evels: 20 O   Contour color	Contour Data Height Data Color Data   Predefined quantities: Electric potential Smooth   Expression: V Recover   Unit: V Recover   Contour levels   Number of levels Vector with isolevels   evels: 20   Labels   Contour color   Colormap: jet   Image: Secondary Color scale   Image: Uniform color: Color	ieneral Surface	Contour	Boundary	Arrow	Principal	Streamline
Predefined quantities: Electric potential Smooth Expression: V Unit: V Contour levels Number of levels Vector with isolevels evels:  20 Labels Contour color	Predefined quantities: Electric potential Smooth Expression: V Unit: V Contour levels Number of levels Number of levels Vector with isolevels evels: 20 Labels Contour color Colormap: jet ? Reverse Color scale Uniform color: Color	🗹 Contour plot					
Expression: V Unit: V Contour levels Number of levels Vector with isolevels Labels Contour color	Expression: V Unit: V Contour levels Number of levels Vector with isolevels Levels:  20 Labels Contour color Colormap: jet jet Color scale Uniform color: Color	Co	ntour Data	Height Data	Color Da	ta	
Unit: V  Contour levels Number of levels Vector with isolevels evels:  20 Labels Contour color	Unit: V Contour levels Number of levels Vector with isolevels Levels:  20 Labels Contour color Colormap: jet jet Color scale Uniform color: Color	Predefined quantities:	Electric p	ootential	\$	🗹 Smooth	
Contour levels       Number of levels     Vector with isolevels       Levels:           20        Labels          Contour color	Contour levels Number of levels Vector with isolevels Levels: Levels: Labels Contour color Colormap: jet jet Color. Color. Color. Color	Expression:	V			Recover	
Number of levels     Vector with isolevels       Labels     20	Number of levels     Vector with isolevels       Levels:     20       Labels       Contour color       Colormap:     jet       Jet       Uniform color:       Color	Unit:	V		\$		
Number of levels     Vector with isolevels       Labels     20	Number of levels     Vector with isolevels       Levels:     20       Labels       Contour color       Colormap:     jet       Jet       Uniform color:       Color						
		Levels: Labels Contour color Colormap:	jet	0			scale
		Filled					
Filled							
] Filled							
Filled							
Filled							
Filled							
Filled							

**FIGURE 4.102** 2D Hall\_Effect\_1 model Plot Parameters window, Contour Data page

The exact voltage difference at any point in the model can be determined by creating a cross-section plot. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters. Select the 2T solution in the Solutions to use window. See Figure 4.104.

Click the Line/Extrusion tab. Enter the coordinates shown in Table 4.15 on the Cross-Section Plot Parameters page. See Figure 4.105.





Plot type	usion plot	) Point plot
Solutions to us	e	Solution at angle (phase):
Select via:		¢ 0 degrees
1.1		Frame:
1.2		
1.3		· · · · · · · · · · · · · · · · · · ·
1.4		
1.5		
1.6		
1.7		
1.8		U
1.9		*
2		<b>T</b>
Times:		
lot in: New	figure 🛟 👔	Keep current plot
	-	
Display cro	ss section in n	main axes Color
_		
Smoothing.	Titla	Axis
Smootning.		AXIS

**FIGURE 4.104** 2D Hall\_Effect\_1 model Cross-Section Plot Parameters, General page

Line Data	Value
x0	0e-3
x1	0e-3
у0	-3e-3
y1	3e-3

### Table 4.15 Cross-Section Line Data Edit Window

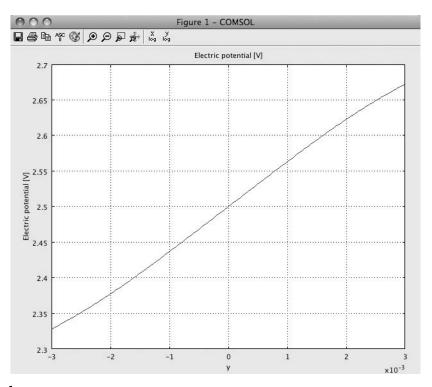
Click OK. Figure 4.106 shows the voltage difference  $(V_{\rm H})$  between the electrode (top) and the modeled Si sample (bottom), for the line x = 0. In this case  $V_{\rm H} = 0.340$  volt  $(V_{\rm high} - V_{\rm low} = 0.340$  V).

### **Postprocessing Animation**

This solution to the 2D Hall\_Effect\_1 model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters.

Plot type Line plot	O Extrusion p	lot
y-axis data		
Predefined quantities:	Electric potential	Recover
Expression:	v	
Unit:	v	•
x-axis data	Cross-section line d	ata
• у	x0: 0e-3 >	1: 0e-3
O Expression	y0: -3e-3 y	1: 3e-3
	Line resolution:	200
Multiple parallel lin	es	I
Number of lines	Vector with dista	nces
• 5	0	

**FIGURE 4.105** 2D Hall\_Effect\_1 model Cross-Section Plot Parameters, Line/Extrusion page



**FIGURE 4.106** 2D Hall\_Effect\_1 model plot V<sub>H</sub>

Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.107). Click the Start Animation button. Save this 2D Hall effect model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4\_HE\_1.avi that was supplied with this book.

# **First Variation on the 2D Hall Effect Model**

This model reflects a closer approach to the construction of a specimen as would be constructed from a silicon wafer. In this model, both Si end contacts and side contacts have been added, as would be the case for a fabricated Si sample.

To start building the Hall\_Effect\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Conductive Media DC. Click the Multiphysics button, and then click the Add button. See Figure 4.108.

Click the Application Mode Properties button. Select "On" from the Weak constraints pull-down list. Select "Non-ideal" from the Constraint type pull-down list. See Figure 4.109. Click OK.

	amline Particle Tracing	Max/Min Deform Animat
Movie settings		Solutions to use
Output type:	Movie	Select via:
File type:	AVI	0
Width (in pixels):	640	0.1 0.2
Height (in pixels):	480	0.3 0.4
Frames per second:	10	0.5
	Advanced	0.6
Static / Eigenfunction	animation	0.8
Cycle type:	Full harmonic	Times:
Number of frames:	11	
Reverse direction           Use camera setting	s from main window	
<u> </u>	s from main window	

**FIGURE 4.107** 2D Hall\_Effect\_1 model animation Plot Parameters window

Model Navigate	or
Model Library User Model	s Open Settings
2D 🗘	Multiphysics
re Media DC tics tics, Generalized tatics , Electric , Magnetic , Electromagnetic hinery	Conductive Media DC (emdc)  Conductive Media DC (emdc)  Dependent variables: V  Application Mode Propertie  Add Geometry
V2	Add Frame Ruling application mode:
emdc2	Conductive Media DC (emdc)
	Model Library User Model 2D vsics re Media DC tics tics, Generalized tatics , Electric , Magnetic , Electromagnetic ninery nal Interaction V2 emdc2

**FIGURE 4.108** Multiphysics Model Navigator window

n Mode Properties	
Lagrange - Quadratic	\$
On	\$
Non-ideal	\$
	Lagrange – Quadratic On

**FIGURE 4.109** Application Mode Properties window

Name	Expression	Value	Description
sigma0	1.04e3[S/m]	1040[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	$(1.25e-4)[m^3/(s \cdot A)]$	Hall coefficient
Bz	0.1[T]	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	1039.82427[S/m]	Conductivity anisotropy 2
V0	5.0[V]	5[V]	Applied voltage
t_Si	1.0e-3[m]	0.001[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	0.013[1]	Conductivity anisotropy 1
s11	coeff0	1039.82427[S/m]	Conductivity matrix term 11
s12	coeff0*coeff1	13.517716[S/m]	Conductivity matrix term 12
s21	-coeff0*coeff1	-13.517716[S/m]	Conductivity matrix term 21
s22	coeff0	1039.82427[S/m]	Conductivity matrix term 22
=			) + +

**FIGURE 4.110** 2D Hall\_Effect\_2 model Constants edit window

### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.16; also see Figure 4.110. Click OK.

# **2D Hall Effect Geometry**

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.8e-2, and a height of 6e-3. Select "Base: Corner" x and set equal to -9e-3 and y equal to -3e-3 in the Rectangle edit window. See Figure 4.111.

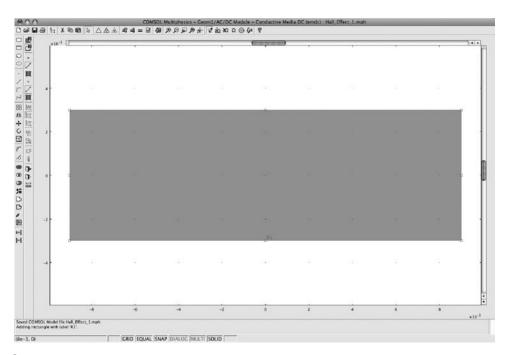
Click OK, and then the click the Zoom Extents button. See Figure 4.112.

Name	Expression	Description
sigma0	1.04e3[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	Hall coefficient
Bz	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	Conductivity anisotropy 2
V0	5.0[V]	Applied voltage
t_Si	1.0e-3[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	Conductivity anisotropy 1
s11	coeff0	Conductivity matrix term 11
s12	coeff0*coeff1	Conductivity matrix term 12
s21	-coeff0*coeff1	Conductivity matrix term 21
s22	coeff0	Conductivity matrix term 22

### Table 4.16 Constants Edit Window

Size		Rotat	ion angle-	
Width:	1.8e-2	α:	0	(degrees)
Height:	6e-3			
Position	1			
Base:	Corner	\$ Style:	Solid	\$
		Name:	R1	
<b>x</b> :	-9e-3			

**FIGURE 4.111** 2D Hall\_Effect\_2 model Rectangle edit window



**FIGURE 4.112** 2D Hall\_Effect\_2 model rectangle geometry

Rectangle Number	Width	Height	Base	x Location	y Location
1	2e-3	1e-3	Corner	-1e-3	3e-3
2	2e-3	1e-3	Corner	-1e-3	-4e-3
3	1e-3	6e-3	Corner	-1e-2	-3e-3
4	1e-3	6e-3	Corner	9e-3	-3e-3

#### Table 4.17 Rectangle Edit Window

In this model, rectangles will be added to the boundary of the rectangle to define the location of the positions of the contacts and the floating contacts. Using the menu bar, select Draw > Specify Objects > Rectangle. In the Draw > Specify Objects > Rectangle edit window, individually create each of the rectangles shown in Table 4.17 by selecting the window, entering the data, and then clicking OK. The final geometry with all four added rectangles is shown in Figure 4.114.

Click the Zoom Extents button. Select Draw > Create Composite Object. Select all of the rectangles. Verify that the Keep interior boundaries check box is checked. see Figure 4.113. Click OK. Figure 4.114 shows the composite object.

NOTE The contact rectangles are added to the boundary of the first rectangle so that the contacts and the floating contacts are precisely defined.

#### 2D Hall Effect Subdomain Settings

Using the menu bar, select Physics > Subdomain Settings. Select subdomains 1, 3, 4, and 5 in the Subdomain selection window. Enter t\_Si in the d (Thickness) edit window.

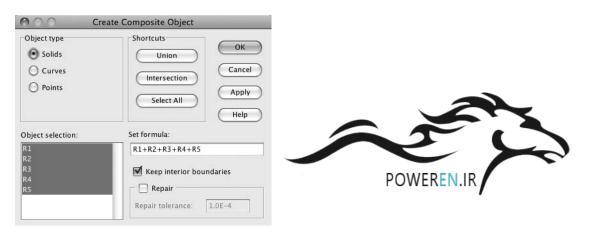
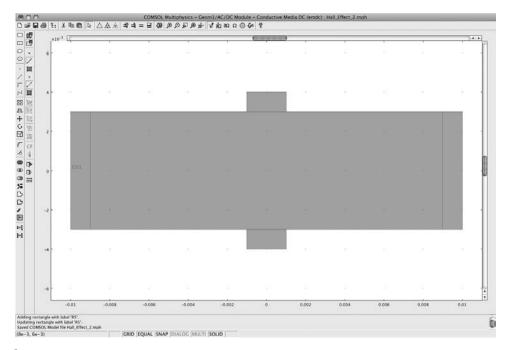


FIGURE 4.113 2D Hall\_Effect\_2 model, Create Composite Object window



**FIGURE 4.114** 2D Hall\_Effect\_2 model geometry with added rectangles

Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Enter sigma0 in the Electric conductivity window. See Figure 4.115.

Select subdomain 2 in the Subdomain selection window. Enter: t\_Si in the d (Thickness) edit window. Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Click in the Electric conductivity edit window. Select "Anisotropic-full" from the Conductivity type pull-down list. Enter the matrix elements as shown in Table 4.18; see Figures 4.116 and 4.117. Click OK.

NOTE These matrix elements define the anisotropic coupling of the magnetic field and the current flowing in the silicon sample.

Matrix Element Number	Value
11	s11
12	s12
21	s21
22	s22

Table 4.18 Matrix Elei	ments Edit Window
------------------------	-------------------

Subdomains Groups	Physics	s Infinite Elements	Init Ele	ment Color
ubdomain selection	Material properties and	l sources		
2	Library material:	÷ (1	oad)	
3	J <sup>e</sup> Q <sub>j</sub> d Conductivity relation:	Value/Expression       0     0       0     0       t_Si     Conductivity	Unit A/m <sup>2</sup> A/m <sup>3</sup> m	<b>Description</b> External current density Current source Thickness
iroup: 🔹 🗘	σ	sigma0	S/m	Electric conductivity

**FIGURE 4.115** 2D Hall\_Effect\_2 model geometry, Subdomain Settings (1, 3, 4, 5)

Subdomains Groups	Physic	2	lnit Ele	ment Color
Subdomain selection	Material properties and Library material:		oad)	
2 3 4 5	Quantity J <sup>e</sup> Q <sub>j</sub> d Conductivity relation: σ	Value/Expression       0     0       0     0       t_Si     Conductivity	Unit A/m <sup>2</sup> A/m <sup>3</sup> m	Description External current density Current source Thickness
roup: 🔹		s11 s21 s12 s22		Electric conductivity

**FIGURE 4.116** 2D Hall\_Effect\_2 model geometry, Subdomain Settings (2)

# Table 4.19Boundary Settings

Boundary Number	Condition	Group Index	Source Current/Potential	Figure
1	Ground	—	_	4.118
2, 3, 5–7,	Electric insulation	—	_	4.119
10, 13–16,				
18, 19				
8	Floating potential	2	0	4.120
12	Floating potential	1	0	4.121
20	Electric potential	—	VO	4.122

# **2D Hall Effect Boundary Settings**

Using the menu bar, select Physics > Boundary Settings. Enter the boundary settings as shown in Table 4.19. See Figures 4.118, 4.119, 4.120, 4.121, and 4.122.

s11	s12	
s21	\$22	
Anisotropic -	full	

**FIGURE 4.117** 2D Hall\_Effect\_2 model conductivity matrix elements

Boundaries Groups	Conditions Port Weak Constr. Color/Style
Boundary selection	Boundary sources and constraints Library material:
Group: +	

FIGURE 4.118 2D Hall\_Effect\_2 model Boundary Settings (1)

<b>J</b> = 0	
Boundaries Groups Boundary selection	Conditions       Port       Weak Constr.       Color/Style         Boundary sources and constraints
Interior boundaries	

FIGURE 4.119 2D Hall\_Effect\_2 model Boundary Settings (2, 3, 5–7, 10, 13–16, 18, 19)

Boundaries Groups	Conditio	ns Port Weak Const	tr. Color/Style
oundary selection	Boundary sources and cor	istraints	
ñ	Library material:	Load.	.)
0	Boundary condition	Floating potential	\$
0	Quantity	Value/Expression	Unit Description
2 4		2	1 Group index
roup:	Io	0	A Source current
Select by group			
Interior boundaries			

**FIGURE 4.120** 2D Hall\_Effect\_2 model Boundary Settings (8)

Boundaries Groups	Conditio	ons Port Weak Cons	str. Color/Style
Soundary selection	Boundary sources and co Library material: Boundary condition Quantity	toad	Unit Description 1 Group index A Source current
Select by group			

**FIGURE 4.121** 2D Hall\_Effect\_2 model Boundary Settings (12)

**NOTE** The addition of the group index designation decouples the two floating contacts from each other.

Select the Weak Constr. tab. Verify that the Use weak constraints check box is checked. See Figure 4.123.

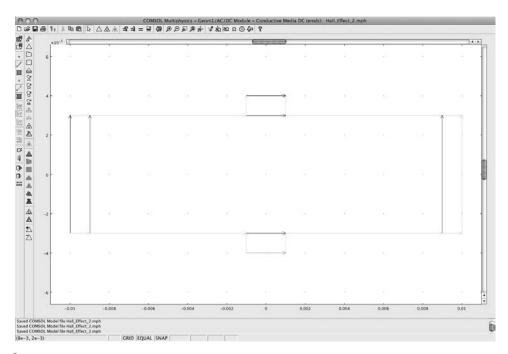
Click OK. The final configuration of the boundary settings is shown in Figure 4.124.

Boundaries Groups	Condition	is Port Weak Con	str. Color/Style
oundary selection	Boundary sources and con Library material: Boundary condition: Quantity V <sub>0</sub>	t Load	Unit Description
Select by group			

**FIGURE 4.122** 2D Hall\_Effect\_2 model Boundary Settings (20)

Boundaries Groups	Predefined e	ak constraints	Color/Style Description Shape functions Integration order Initial value
Select by group			

**FIGURE 4.123** 2D Hall\_Effect\_2 model Boundary Settings, Weak Constr. page



**FIGURE 4.124** 2D Hall\_Effect\_2 model boundary settings, final configuration

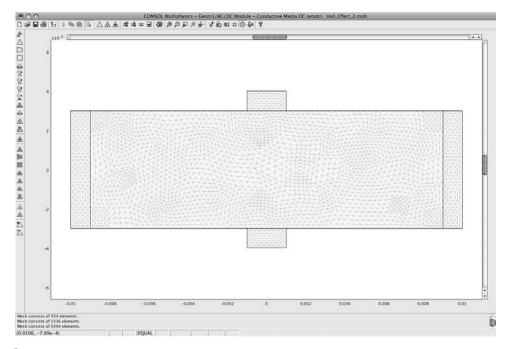


FIGURE 4.125 2D Hall\_Effect\_2 model mesh

# **2D Hall Effect Mesh Generation**

Using the menu bar, select Mesh > Initialize Mesh. Click the Refine Mesh button twice. See Figure 4.125.

# Solving the First Variation on the 2D Hall Effect Model

Using the menu bar, select Solve > Solver Parameters. In the Solver selection window, select "Parametric." In the Parameter name edit window, enter Bz. In the Parameter values edit window, enter 0:0.1:2.0. See Figure 4.126. Click OK.

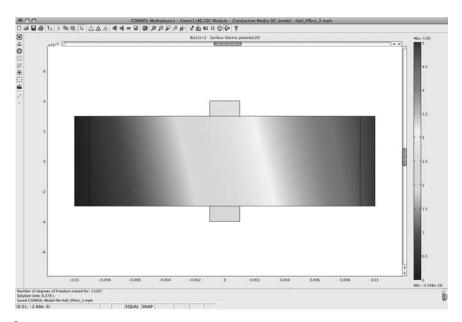
Note The Parametric Solver is chosen, in this case, so that the modeler can solve the Hall\_Effect\_2 Model quasi-statically. This allows the modeler to see solutions over a wide range of magnetic field values.

Using the menu bar, select Solve > Solve Problem.

# Postprocessing

The default plot is a 2D surface plot of the voltage distribution at the highest value of the magnetic field (Bz = 2 Tesla). See Figure 4.127.

**FIGURE 4.126** 2D Hall\_Effect\_2 model Solver Parameters window



**FIGURE 4.127** 2D Hall\_Effect\_2 model default surface voltage distribution plot

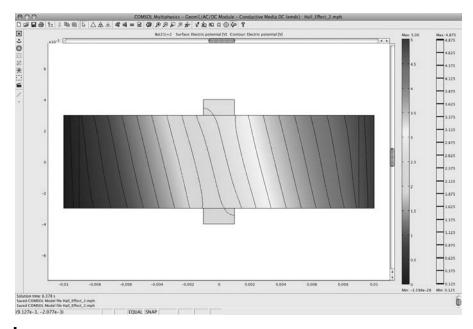
General Surface	Contour	Boundary	Arrow Principal
General Surface	contour	boundary	Arrow Principal
🗹 Contour plot			
Con	tour Data He	ight Data Color I	Data
		-	
Predefined quantities:	Electric poten	tial 🗘	
Expression:	v		Smooth
Unit:	V	\$	
Contour levels			
Numb	er of levels	Vector with isol	evels
Levels: 💿 20		C	
Labels			
Contour color			
O Colormap:	cool 🛟	Colors: 1024	Color scale
Uniform color: (	Color		
0 011101111 (			
Filled			

**FIGURE 4.128** 2D Hall\_Effect\_2 model Plot Parameters, Contour Data page

More detailed information is displayed by adding contour lines. Using the menu bar, select Postprocessing > Plot Parameters. Click the Contour tab. Place a check mark in the Contour plot check box. Click the Uniform color radio button. Click the Color select button, and select "Black." See Figure 4.128.

Click OK. The Hall effect voltage  $(V_{\rm H})$  can be seen as the voltage difference between the top electrode and the bottom electrode, as shown in Figure 4.129.

The exact voltage difference at any point in the model can be determined by creating a cross-section plot. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters. Select the 2T solution in the Solutions to use window. See Figure 4.130.



**FIGURE 4.129** 2D Hall\_Effect\_2 model surface voltage distribution plot (2T), with contour lines

Plot type Line/Extrusion	plot 🔿 Point	plot	
olutions to use		Solution a	Langle (phase).
Select via:	(¢)	0	degrees
1.1	1221	Frame:	1
1.2		(	100
1.3			\$
1.4			
1.5			
1.6	m		
1.7			
1.8	U		
1.9	+		
2	<b>•</b>		
Fimes:			
ot In: New figure	Keer	current plo	t
		/ current pic	
Display cross-se	ction in main axe	s Col	pr
in property cross of		- <u>_</u>	
( litle/Axis )			

**FIGURE 4.130** 2D Hall\_Effect\_2 model Cross-Section Plot Parameters, General page

Line Data	Value
x0	0e-3
x1	0e-3
у0	-4e-3
y1	4e-3

Table 4.20Cross-Section Line Data Edit Window

Click the Line/Extrusion tab. Enter the coordinates shown in Table 4.20 on the Cross-Section Plot Parameters page. See Figure 4.131.

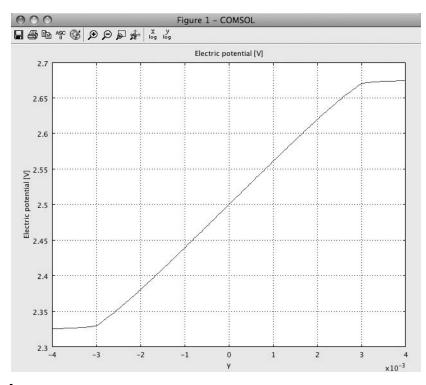
Click OK. Figure 4.132 shows the voltage difference  $(V_{\rm H})$  between the electrode (top) and the modeled Si sample (bottom), for the line x = 0. In this case,  $V_{\rm H} = 0.350$  volts  $(V_{\rm high} - V_{\rm low} = 0.350$  V).

#### **Postprocessing Animation**

This solution to the 2D Hall\_Effect\_2 model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.133). Click the

Line/Extrusion	plot					
Not type			~ ~			
Line plot			O Extrus	on plot		
-axis data						
Predefined quantitie	s: Ele	ctric p	otential	_	•	
Expression:	V				_	
	-					
Jnit:	V					
(-axis data		Cro	ss-section	line data		
• у	\$	×0:	0e-3	×1:	0e-3	
O Expressio	m)	y0:	-1e-3	y1:	4e-3	
		Line	resolution		200	
		-				
Multiple paralle		14	ector with d			
(*) 5	nes	0	scror with u	istances		
0 3		0				
Line Settings	5 0		e Settings	-		
Line Settings	20	Surrac	e settings.	9		

FIGURE 4.131 2D Hall\_Effect\_2 model Cross-Section Plot Parameters, Line/Extrusion page



**FIGURE 4.132** 2D Hall\_Effect\_2 model plot V<sub>H</sub>

Start Animation button. Save this 2D Hall effect model animation by clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4\_HE\_2.avi that was supplied with this book.

# Second Variation on the 2D Hall Effect Model

This Hall effect model demonstrates the effect of having holes as the carrier in this electronic material (a p-type semiconductor). A second, lower contact has been added to allow the use of an external balancing circuit. A change is made in the value of the conductivity, because holes are less mobile than electrons in Si.

Note Semiconductors have two types of carriers: electrons (n-type) and holes (p-type). In a purified semiconductor, such as silicon (Si) or germanium (Ge), both carriers are thermally activated and exist in equal numbers. That native conduction mode is called the intrinsic conduction mode.

To fabricate electronic device structures, foreign atoms (As or P for n-type and Al or B for p-type) are added to the host lattice (Si). The carriers are more easily thermally activated from the foreign atoms (dopant atoms) at room temperature. This non-native



FIGURE 4.133 2D Hall\_Effect\_2 model animation Plot Parameters window

conduction mode is called the extrinsic conduction mode. In the extrinsic mode, the carriers activated from the dopant atoms and the small number of carriers activated intrinsically become the majority carriers (e.g., electrons). The second carrier (holes in this example) becomes the minority carrier. The electron and hole carrier densities are related by the mass action law:<sup>18</sup>

$$np = n_i^2 \tag{4.10}$$

where	$n_i$ = intrinsic carrier density	
	n = electron carrier density	
	p = hole carrier density	

New	Model Library User Models	Open Settings
Space dimension:	2D 🗘	Multiphysics
Application Modes	ñ	(Add) (Remove
COMSOL Multiphy	sics	Geom1 (2D)
<ul> <li>AC/DC Module</li> <li>Statics</li> </ul>		Conductive Media DC (emdc
	e Media DC	
Electrosta		
Electrosta	tics, Generalized	
Magnetos		
Quasi-Statics		Dependent variables: V
Quasi-Statics	, Magnetic , Electromagnetic	Application Mode Propertie
Rotating Mach		
Electro-Therr	nal Interaction	Add Geometry
		Add Frame
Dependent variables:	V2	Ruling application mode:
Application mode name:	emdc2	Conductive Media DC (emdc)
Element:	Lagrange - Quadra	Multiphysics

**FIGURE 4.134** Multiphysics Model Navigator window

To start building the Hall\_Effect\_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Conductive Media DC. Click the Multiphysics button, and then click the Add button. See Figure 4.134.

Click the Application Mode Properties button. Select "On" from the Weak constraints pull-down list. Select "Non-ideal" from the Constraint type pull-down list. See Figure 4.135. Click OK.

Default element type:	Lagrange - Quadratic
Weak constraints:	On
Constraint type:	Non-ideal

FIGURE 4.135 Application Mode Properties window

Name	Expression	Description
sigma0	2.4e2[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	Hall coefficient
Bz	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	Conductivity anisotropy 2
V0	5.0[V]	Applied voltage
t_Si	1.0e-3[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	Conductivity anisotropy 1
s11	coeff0	Conductivity matrix term 11
s12	-coeff0*coeff1	Conductivity matrix term 12
s21	coeff0*coeff1	Conductivity matrix term 21
s22	coeff0	Conductivity matrix term 22

#### Table 4.21 Constants Edit Window

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 4.21; also see Figure 4.136. Click OK.

#### **2D Hall Effect Geometry**

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.8e-2, and a height of 6e-3. Select "Base: Corner" and set x equal to -9e-3, and y equal to -3e-3 in the Rectangle edit window. See Figure 4.137.

Name	Expression	Value	Description
sigma0	2.4e2[S/m]	240[S/m]	Silicon conductivity
Rh	1.25e-4[m^3/C]	$(1.25e-4)[m^3/(s \cdot A)]$	Hall coefficient
Bz	0.1[T]	0.1[T]	Magnetic field
coeff0	sigma0/(1+(sigma0*Rh*Bz)^2)	239.99784[S/m]	Conductivity anisotropy 2
V0	5.0[V]	5[V]	Applied voltage
t_Si	1.0e-3[m]	0.001[m]	Silicon thickness
coeff1	sigma0*Rh*Bz	0.003[1]	Conductivity anisotropy 1
s11	coeff0	239.99784[S/m]	Conductivity matrix term 11
s12	-coeff0*coeff1	-0.719994[S/m]	Conductivity matrix term 12
s21	coeff0*coeff1	0.719994[S/m]	Conductivity matrix term 21
s22	coeff0	239.99784[S/m]	Conductivity matrix term 22

FIGURE 4.136 2D Hall\_Effect\_3 model Constants edit window

Size		Rota	tion angle	
Width:	1.8e-2	α:	0	(degrees)
Height:	6e-3			
Position				
Base:	Corner	\$ Style:	Solid	\$
	-9e-3	Name:	R1	
x:	1			

FIGURE 4.137 2D Hall\_Effect\_3 model Rectangle edit window

Click OK, and then click the Zoom Extents button. See Figure 4.138.

In this model, rectangles will be added to the boundary of the rectangle to define the location of the positions of the contacts and the floating contacts. Using the menu bar, select Draw > Specify Objects > Rectangle. In the Draw > Specify Objects > Rectangle edit window, individually create each of the rectangles shown in Table 4.22 by selecting the window, entering the data, and then clicking OK.

Click the Zoom Extents button. Select Draw > Create Composite Object. Select all of the rectangles. Verify that the Keep interior boundaries check box is checked. See Figure 4.139.

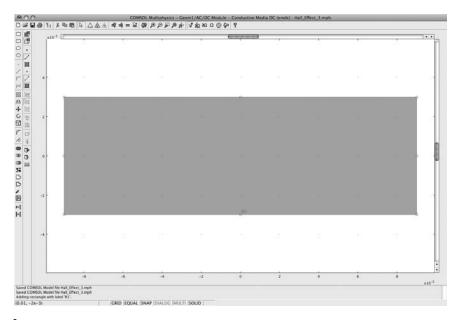


FIGURE 4.138 2D Hall\_Effect\_3 model rectangle geometry

Rectangle Number	Width	Height	Base	x Location	y Location
1	2e-3	1e-3	Corner	-1e-3	3e-3
2	2e-3	1e-3	Corner	-4e-3	-4e-3
3	2e-3	1e-3	Corner	2e-3	-4e-3
4	1e-3	6e-3	Corner	-1e-2	-3e-3
5	1e-3	6e-3	Corner	9e-3	-3e-3

Table 4.22Rectangle Edit Window

Click OK. Figure 4.140 shows the composite object.

NOTE One contact rectangle at the top of the Hall effect model, two contact rectangles at the bottom, and two contact rectangles on the ends are added to the boundary of the first rectangle so that the contacts and the floating contacts are precisely defined. The three contacts (top and bottom) are a typical experimental configuration to allow the measuring instrument to balance the circuit and offset any unintended error voltages.

#### **2D Hall Effect Subdomain Settings**

Using the menu bar, select Physics > Subdomain Settings. Select subdomains 1, 3, 4, 5, and 6 in the Subdomain selection window. Enter t\_Si in the d (Thickness) edit window. Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Enter sigma0 in the Electric conductivity window. See Figure 4.141.

000	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: R1 R2 R3 R4 R5 R6	Set formula: R1+R2+R3+R4+R5+R6 Keep interior boundaries Repair Repair Repair 1.0E-4

FIGURE 4.139 2D Hall\_Effect\_3 model, Create Composite Object window

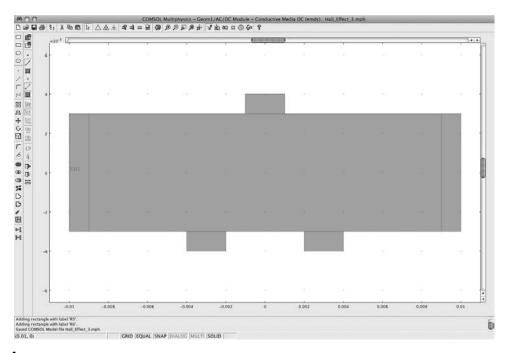


FIGURE 4.140 2D Hall\_Effect\_3 model geometry with added rectangles

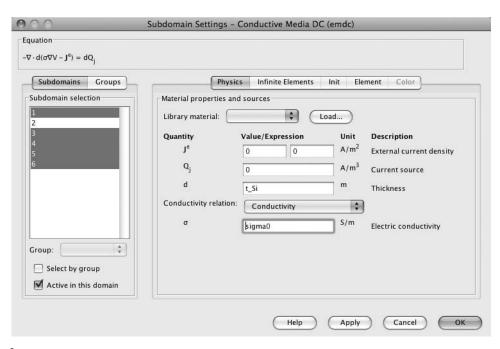


FIGURE 4.141 2D Hall\_Effect\_3 model geometry, Subdomain Settings (1, 3, 4, 5, 6)

Matrix Element Number	Value
11	s11
12	s12
21	s21
22	s22

Table 4.23 Matrix Elements Edit Window

NOTE These matrix elements define the anisotropic coupling of the magnetic field and the current flowing in the silicon sample.

Select subdomain 2 in the Subdomain selection window. Enter t\_Si in the d (Thickness) edit window. Verify that "Conductivity" is selected in the Conductivity relation pull-down list. Click in the Electric conductivity edit window. Select "Anisotropic-full" from the Conductivity type pull-down list. Enter the matrix elements as shown in Table 4.23; see Figures 4.142 and 4.143. Click OK.

Subdomains Groups	Physic	s Infinite Elements	Init Ele	ment Color
Subdomain selection	Material properties and Library material		oad	
3 4 5 6	Quantity J <sup>a</sup> Qj d Conductivity relation o	Value/Expression           0           0           0           c           c           Conductivity           \$11 s21 s12 s22	Unit A/m <sup>2</sup> A/m <sup>3</sup> m	Description External current density Current source Thickness Electric conductivity
Group: Group				

**FIGURE 4.142** 2D Hall\_Effect\_3 model geometry, Subdomain Settings (2)

s22	
	s22

**FIGURE 4.143** 2D Hall\_Effect\_3 model conductivity matrix elements

Boundary Number	Condition	Group Index	Source Current/Potential	Figure
1	Ground	—		4.144
2, 3, 5–7, 10–12,	Electric insulation	_	_	4.145
15–17, 20, 21				
23, 24				
8	Floating potential	1	0	4.146
14	Floating potential	2	0	4.147
18	Floating potential	3	0	4.148
25	Electric potential	—	VO	4.149

Table 4.24 Boundary Settings

**NOTE** The addition of the group index designation decouples the three floating contacts from one another.

# **2D Hall Effect Boundary Settings**

Using the menu bar, select Physics > Boundary Settings. Enter the boundary settings as shown in Table 4.24. See Figures 4.144, 4.145, 4.146, 4.147, 4.148, and 4.149.

Boundaries Groups	Conditions Port Weak Constr. Color/Style
Boundary selection	Boundary sources and constraints Library material:
Select by group	

**FIGURE 4.144** 2D Hall\_Effect\_3 model Boundary Settings (1)

Boundaries Groups	Conditions Port Weak Constr. Color/Style
oundary selection	Boundary sources and constraints
18	Library material:
19 20	
20	Boundary condition: Electric insulation
22	
23 24	
25 💌	
iroup:	
Select by group	
Interior boundaries	

**FIGURE 4.145** 2D Hall\_Effect\_3 model Boundary Settings (2, 3, 5–7, 10–12, 15–17, 20, 21, 23, 24)

Boundaries Groups	Conditi	ions Port Weak Cons	tr. Color/Style
oundary selection	Boundary sources and c	onstraints	
ñ	Library material:	toad	
0	Boundary conditio	n: Floating potential	•
.0	Quantity	Value/Expression	Unit Description
1 2		1	1 Group index
roup:	l <sub>o</sub>	0	A Source current
Select by group			
Interior boundaries			

**FIGURE 4.146** 2D Hall\_Effect\_3 model Boundary Settings (8)

Boundaries Groups	Co	nditions Port Weak Con	str. Color/Style
oundary selection	Boundary sources a	nd constraints	
11 12 13 14 15 16 17 18 18 18 10 10 10 10 10 10 10 10 10 10	Library material: ( Boundary con Quantity <sup>I</sup> 0	total	Unit     Description       1     Group index       A     Source current

**FIGURE 4.147** 2D Hall\_Effect\_3 model Boundary Settings (14)

Boundaries Groups	Condition	ns Port Weak Const	tr. Color/Style
oundary selection	Boundary sources and con	straints	
11 22 33 44 55 16 77 82 <b>V</b>	Library material: Boundary condition: Quantity	Value/Expression	Unit Description
roup: *	lo	0	A Source current

**FIGURE 4.148** 2D Hall\_Effect\_3 model Boundary Settings (18)

Boundaries Groups	Conditions Port Weak Constr. Color/Style
Boundary selection     18     19     20     21     22     23     24     25     Group:     Select by group     Interior boundaries	Boundary sources and constraints Library material:  Library material:  Library material:  Library material:  Library material:  Load Boundary condition: Electric potential Quantity Value/Expression Unit Description V_0 V0 V Electric potential

**FIGURE 4.149** 2D Hall\_Effect\_3 model Boundary Settings (25)

Click the Weak Constr. tab. Verify that the Use weak constraints check box is checked. See Figure 4.150.

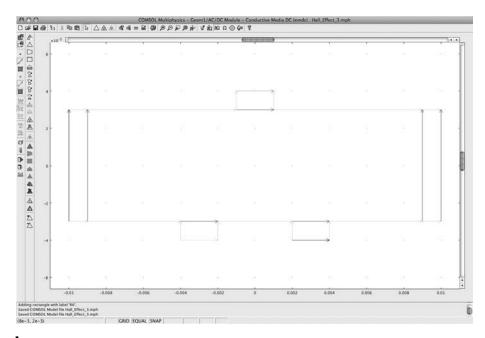
Click OK. The final configuration of the boundary settings is shown in Figure 4.151.

#### **2D Hall Effect Mesh Generation**

Using the menu bar, select Mesh > Initialize Mesh. Click the Refine Mesh button twice. See Figure 4.152.

-Weak constrai	nts	
🗹 Use wea	constraints	
Predefined e	ements: Lagrange - Quadratic	
		Description
wcshape		Shape functions
wcaporder	I de la companya de la	
	30	Integration order
weinit	0	Initial value
	Predefined el	wrgporder 30

**FIGURE 4.150** 2D Hall\_Effect\_3 model Boundary Settings, Weak Constr. page



**FIGURE 4.151** 2D Hall\_Effect\_3 model boundary settings, final configuration

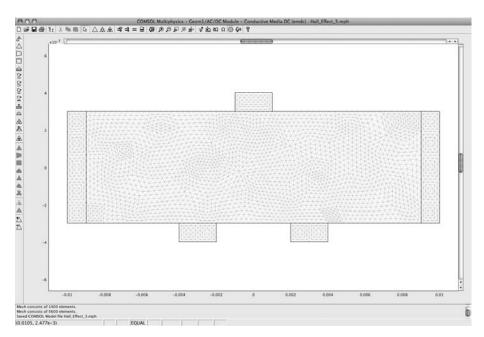


FIGURE 4.152 2D Hall\_Effect\_3 model mesh

000	Solver Par	rameters	
Analysis: Analysis:  Analysis:  Auto select solver Solver: Stationary Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated		rametric Stationar	Advanced
Adaptive mesh refinement	Matrix symmetry:	Automatic	 Settings
		Help A	ancel OK

FIGURE 4.153 2D Hall\_Effect\_3 model Solver Parameters window

# Solving the Second Variation on the 2D Hall Effect Model

Using the menu bar, select Solve > Solver Parameters. In the Solver selection window, select "Parametric." In the Parameter name edit window, enter Bz. In the Parameter values edit window, enter 0:0.1:2.0. See Figure 4.153. Click OK.

Note The Parametric Solver is chosen, in this case, so that the modeler can solve the Hall\_Effect\_3 Model quasi-statically. This allows the modeler to see solutions over a wide range of magnetic field values.

Using the menu bar, select Solve > Solve Problem.

#### Postprocessing

The default plot is a 2D surface plot of the voltage distribution at the highest value of the magnetic field (Bz = 2 Tesla). See Figure 4.154.

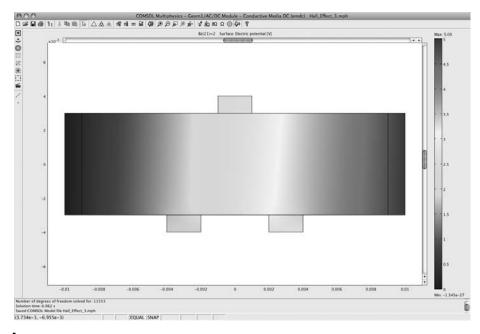


FIGURE 4.154 2D Hall\_Effect\_3 model default surface voltage distribution plot

More detailed information is displayed by adding contour lines. Using the menu bar, select Postprocessing > Plot Parameters. Click the Contour tab. Place a check mark in the Contour plot check box. Click the Uniform color radio button. Click the Color select button, and select "Black." See Figure 4.155.

Click OK. See Figure 4.156.

The exact voltage difference at any point in the model can be determined by creating a cross-section plot. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters. Select the 2T solution in the Solutions to use window. See Figure 4.157.

Click the Line/Extrusion tab. Enter the coordinates shown in Table 4.25 on the Cross-Section Plot Parameters page.

#### Table 4.25 Cross-Section Line Data Edit Window

Line Data	Value
x0	0e-3
x1	0e-3
у0	-3e-3
y1	4e-3

General Surface	Contour Boundary Arrow Principal
🗹 Contour plot	
C	ontour Data Height Data Color Data
Predefined quantities	Electric potential
Expression:	V Smooth
Unit:	V 🗘
Contour levels	mber of levels Vector with isolevels
Levels:	
Labels	
Contour color	
O Colormap:	cool Colors: 1024 Color scale
Uniform color:	Color
Filled	

FIGURE 4.155 2D Hall\_Effect\_3 model Plot Parameters, Contour Data page

Select "y" on the x-axis data pull-down list. See Figure 4.158.

Click OK. Figure 4.159 shows the voltage difference  $(V_{\rm H})$  between the electrode (top) and the modeled Si sample (bottom), for the line x = 0. In this case,  $V_{\rm H} = 0.085$  volts  $(V_{\rm high} - V_{\rm low} = 0.085 \text{ V})$ .

#### **Postprocessing Animation**

This solution to the 2D Hall\_Effect\_3 model can also be viewed as an animation. To view the solution as a movie, using the menu bar, select Postprocessing > Plot Parameters. Once the Plot Parameters window appears, click the Animate tab. On the Animate page, select all the solutions in the Stored output times window (see Figure 4.160). Click the Start Animation button. Save this 2D Hall effect model animation by

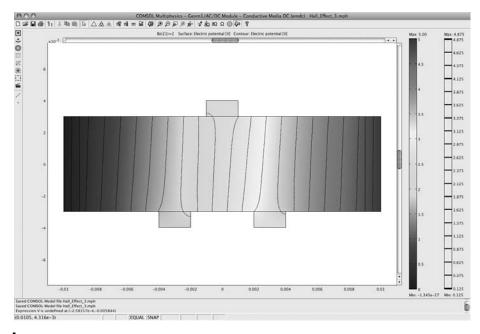


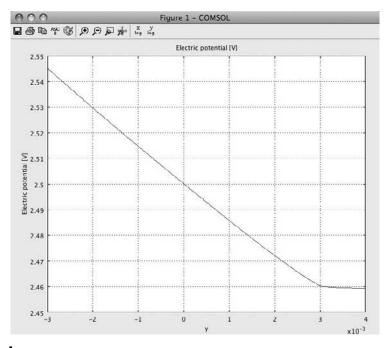
FIGURE 4.156 2D Hall\_Effect\_3 model surface voltage distribution plot (2T), with contour lines

0.0	Cross-Section	on Plot Parameters
	General Lir	re/Extrusion Point
Plot type		
• Line/Extr	usion plot 🔘 Po	pint plot
Solutions to us	ie	Solution at angle (phase):
Select via:		
Select via:	¥	0 degrees
1.1		Frame:
1.2		( ) ( ) ( ) ( ) ( ) ( ) ( ) ( ) ( ) ( )
1.3		· · · · · ·
1.4		
1.5		
1.6		
1.7		
1.8		
1.9		<b>A</b>
2		<b>T</b>
Times:		
lot in: New	figure 🗘 🗌	Keep current plot
d .		
Display cro	oss-section in main	axes Color
Title/Axis		

FIGURE 4.157 2D Hall\_Effect\_3 model Cross-Section Plot Parameters, General page

Plot type Line plot	C Extrusion plot	t
-axis data		
Predefined quantities:	Electric potential	¢
xpression:	V	
Jnit:	V	\$
-axis data	Cross-section line dat	a
• Y	x0: 0e-3 x1	0e-3
O Expression	y0: -3e-3 y1:	4e-3
	Line resolution:	200
Multiple parallel Number of line		15

**FIGURE 4.158** 2D Hall\_Effect\_3 model Cross-Section Plot Parameters, Line/Extrusion page



**FIGURE 4.159** 2D Hall\_Effect\_3 model plot V<sub>H</sub>

Streamline	Particle Tracing	Max/Min Deform	Animat
Movie settings		Solutions to use	
File type:	AVI	Select via:	\$
Width (in pixels):	640	0	
Height (in pixels):	480	0.1	
		0.2 0.3	U
Frames per second:	10	0.4	
	Advanced	0.5 0.6	
Static / Eigenfunctio	n animation	0.7 0.8	×
Cycle type:	Full harmonic	0.9	•
	11	Times:	
Number of frames: Reverse direction Use camera setti	L		
Reverse direction	1		Animation

FIGURE 4.160 2D Hall\_Effect\_3 model animation Plot Parameters window

clicking on the disk icon on the player screen. Alternatively, you can play the file Movie4\_HE\_3.avi that was supplied with this book.

# 2D Hall Effect Models: Summary and Conclusions

The models presented in this section of Chapter 4 have introduced the following new concepts: two-dimensional modeling (2D), the Hall effect, AC/DC Module, Conductive Media DC, weak constraints, floating contacts, anisotropic conductivity, semiconductor dual-carrier types, and imbalance-offset geometry. The 2D Hall effect model is a powerful tool that can be used to model Hall effect magnetic sensors for sensing fluid flow, rotating or linear motion, proximity, current, pressure, and orientation. A comparison of the calculated results for the three Hall effect models is shown in Table 4.26.

Model	Floating Contacts	Carrier Type	<i>V</i> <sub>H</sub> (V)	( <i>V</i> <sub>H</sub> ∆%)
HE_1	Point defined	n-type (electron)	0.340	_
HE_2	Dual rectangle defined	n-type (electron)	0.350	~+3
HE_3	Triple rectangle defined	p-type (hole)	-0.085	~-75

The differences between the calculations for the tested models for the n-type carrier are in the range of a few percentage points. It can clearly be seen that the p-type silicon is only one-fourth as sensitive as the n-type material. That reduction in sensitivity is attributable to the low hole mobility. It is left to the modeler to explore other differences between the models by variation of the parameters, as suggested in the exercises at the end of this chapter.

#### References

- 1. http://en.wikipedia.org/wiki/Electrochemical
- 2. http://en.wikipedia.org/wiki/Electropolishing
- 3. http://en.wikipedia.org/wiki/William\_Gilbert
- 4. http://en.wikipedia.org/wiki/Charles-Augustin\_de\_Coulomb
- 5. http://en.wikipedia.org/wiki/Joseph\_Priestley
- 6. http://en.wikipedia.org/wiki/Georg\_Ohm
- 7. http://en.wikipedia.org/wiki/Michael\_Faraday
- 8. "Deformed Meshes" in COMSOL Multiphysics Modeling Guide, Version 3.4, 392–405.
- 9. http://en.wikipedia.org/wiki/Ohm%27s\_law
- 10. C. Kittel, *Introduction to Solid State Physics* New York: John Wiley & Sons, 1986), 206–214.
- S. M. Sze, Semiconductor Devices, Physics and Technology (New York: John Wiley & Sons), 1985, 34–40.
- 12. J. M. Ziman, *Principles of the Theory of Solids* (Cambridge, UK: Cambridge University Press), 1969, 185.
- 13. http://en.wikipedia.org/wiki/Hall\_Effect
- 14. http://en.wikipedia.org/wiki/Edwin\_Hall
- 15. http://en.wikipedia.org/wiki/Lorentz\_Force

- R. A. Smith, *Semiconductors* (Cambridge, UK: Cambridge University Press, 1968), 100–107.
- 17. COMSOL Multiphysics Software Models Database, Hall Plate with Floating Contacts.
- S. M. Sze, Semiconductor Devices, Physics and Technology (New York: John Wiley & Sons), 1985, 16–20.

# Exercises

- 1. Build, mesh, and solve the COMSOL 2D electrochemical polishing model problem presented in this chapter.
- 2. Build, mesh, and solve the first variation of the 2D electrochemical polishing model problem presented in this chapter.
- 3. Build, mesh, and solve the second variation of the 2D electrochemical polishing model problem presented in this chapter.
- 4. Build, mesh, and solve the Hall effect model presented in this chapter.
- 5. Build, mesh, and solve the first variation of the Hall effect model presented in this chapter.
- 6. Build, mesh, and solve the second variation of the Hall effect model presented in this chapter.
- 7. Explore other variations of the arguments in the COMSOL 2D electrochemical polishing models.
- 8. Explore other variations of the arguments in the Hall effect models.
- 9. Explore how an increase in the run time modifies the behavior of the COMSOL 2D electrochemical polishing model.
- 10. Explore how changes in the sample geometry affect the behavior of the Hall effect model.

# 5

# **2D Axisymmetric Modeling**

# In This Chapter

2D Axisymmetric Guidelines for New COMSOL® Multiphysics® Modelers
2D Axisymmetric Modeling Considerations
2D Axisymmetric Coordinate System
Heat Conduction Theory
2D Axisymmetric Heat Conduction Modeling
2D Axisymmetric Cylinder Conduction Model
First Variation on the 2D Axisymmetric Cylinder Conduction Model
Second Variation on the 2D Axisymmetric Cylinder Conduction Model,
Including a Vacuum Cavity
2D Axisymmetric Cylinder Conduction Models: Summary and Conclusions
2D Axisymmetric Insulated Container Design
2D Axisymmetric Thermos_Container Model
First Variation on the 2D Axisymmetric Thermos_Container Model
Second Variation on the 2D Axisymmetric Thermos_Container Model
2D Axisymmetric Thermos_Container Models: Summary and Conclusions

# **2D** Axisymmetric Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers

# 2D Axisymmetric Modeling Considerations

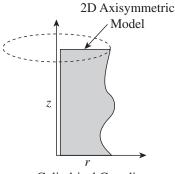
2D axisymmetric modeling can be less difficult than 1D modeling and is about the same level of difficulty as 2D modeling. Specifically, 2D axisymmetric modeling has fewer implicit assumptions than 1D modeling. The 2D axisymmetric model requires the modeler to think in terms of cylindrical coordinates and rotational symmetry. Such models can be challenging to build, depending on the underlying physics involved. The least difficult aspect of 2D axisymmetric model building arises from the fact that the geometry is still relatively simple. (In a 2D model, the modeler has only a single plane as the modeling space.) However, the physics and the rotational nature of the geometry in a 2D axisymmetric model can range from relatively easy to extremely complex.

NOTE COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software has two 2D modeling modes: 2D (beginning-level through advanced-level 2D modeling) and 2D axisymmetric (advanced-level 2D modeling). In keeping with the introductory focus of the material in this text, both of the model types—that is, the 2D model introduced in Chapter 4, and the 2D axisymmetric model introduced in this chapter-along with the associated physics and the related methodology for use of the models, are introduced in this book. Significantly more advanced 2D modeling techniques exist than those presented here in Chapters 4 and 5. Examples of some of those more difficult techniques are reserved for introduction in later chapters (6 and 7). For further expansion of your 2D modeling horizons, refer to the COMSOL Manuals, the COMSOL website, and the general COMSOL Multiphysics software-related research literature.

The 2D axisymmetric model implicitly assumes, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables that are of interest are homogeneous, isotropic, or constant, unless otherwise specified, throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model. Bearing that in mind, the modeler needs to ensure that all of the modeling conditions and associated parameters (default settings) in each new model created have been properly considered, defined, or set to the appropriate value(s).

The modeler also needs to seriously consider the steps that will be required to establish the correct postprocessing and visualization settings so as to extract the desired information from the modeling solution. The default parameter settings on any given model will probably not present exactly the information that the modeler needs or desires, although it may come close to meeting the modeler's demands. It is the responsibility of the modeler to determine exactly which of the myriad of postprocessing and visualization choices available in the COMSOL Multiphysics software to employ.

Note As mentioned previously, it is always preferable for the modeler to be able to accurately anticipate the expected behavior (results) of the model and the presentation of its results. Do not assume that the default values that are initially present when the model is first created will suit the needs of the new model. Always verify that the values employed in the model are the correct values needed for that model. Calculated solution values that significantly deviate from the expected values or from comparison values measured in experimentally derived realistic models are probably indicative of one or more modeling errors either in the original model design, in the earlier model analysis, or in the understanding of the underlying physics, or they may simply be due to human error.



Cylindrical Coordinates

**FIGURE 5.1** 2D axisymmetric coordinate system

# 2D Axisymmetric Coordinate System

In 2D axisymmetric models, there are two geometrical coordinates: space (r) and space (z). See Figure 5.1.

In the steady-state solution to any 2D axisymmetric model, parameters can vary only as a function of the radial position in space (r) and the axial position space (z) coordinates. Such a model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (r) and space (z) and in time (t). The transient solution model is essentially a sequential collection of steady-state (quasi-static) solutions. The space coordinates (r) and (z) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ( $r_{min} \le r \le r_{max}$ ) and ( $z_{min} \le z \le z_{max}$ ). The time coordinate (t) represents the range of values ( $t_{min} \le t \le t_{max}$ ) from the beginning of the observation period ( $t_{min}$ ) to the end of the observation period ( $t_{max}$ ).

To assist the reader in achieving a broader exposure to the applicability of the physics discussed in this chapter and to demonstrate the power of the basic COMSOL 2D axisymmetric modeling techniques, the examples in this chapter demonstrate heat transfer modeling techniques from two substantially different approaches. Heat transfer is an extremely important design consideration. It is one of the most widely needed and applied technologies employed in applied engineering and physics. Most modern products or processes require an understanding of heat transfer either during development or during the use of the product or process (e.g., automobiles, plate glass fabrication, plastic extrusion, plastic products, houses, ice cream).

Note: Heat transfer concerns have existed since the beginning of prehistory. The science of thermodynamics, and consequently the present understanding of heat transfer, started with the work of Nicolas Leonard Sadi Carnot, as published in his 1824 paper titled "Reflections on the Motive Power of Fire."<sup>1</sup> The first use of the term "thermodynamics" is attributed to William Thomson (Lord Kelvin).<sup>2</sup> Subsequent contributions to the understanding of heat, heat transfer, and thermodynamics in general were made by James Prescott Joule,<sup>3</sup> Ludwig Boltzmann,<sup>4</sup> James Clerk Maxwell,<sup>5</sup> Max Planck,<sup>6</sup> and numerous others. The physical understanding and engineering use of thermodynamics play a very important role in the technological aspects of machine and process design in modern applied science, engineering, and medicine.

The first example presented in this chapter, on cylinder conduction, explores the 2D axisymmetric steady-state modeling of heat transfer and temperature profiling for a thermally conductive material, implemented through use of the COMSOL Heat Transfer Module. In the first variation on the cylinder conduction model, a model is built using the basic COMSOL Multiphysics software. The calculated modeling results are then compared. The second variation on the cylinder conduction model explores the use of heat transfer modeling for low-pressure gas/vacuum environments.

The second 2D axisymmetric modeling example in this chapter, titled Thermos\_ Container, explores the modeling of heat loss for thermally insulated containers.

NOTE Insulated containers can be found applied in many different applications in modern society. Examples include Thermos containers, water heaters, and refrigerated liquid containers (for fuels, liquified gases, heat exchangers, and so on).

#### Heat Conduction Theory

Heat conduction is a naturally occurring process that is readily observed in many aspects of modern life (e.g., refrigerators, freezers, microwave ovens, thermal ovens, engines). The heat transfer process allows both linear and rotational work to be done in the generation of electricity and the movement of vehicles. The initial understanding of transient heat transfer was developed by Newton<sup>7</sup> and started with Newton's law of cooling:<sup>8</sup>

$$\frac{dQ}{dt} = h^* A^* (T_{\rm S} - T_{\rm E}) \tag{5.1}$$

where

 $\frac{dQ}{dt}$  is the incremental energy lost in joules per unit time (J/s) A is the energy transmission surface area (m<sup>2</sup>) h is the heat transfer coefficient [W/(m<sup>2</sup>\*K)]  $T_{\rm S}$  is the surface temperature of the object losing heat (K)  $T_{\rm F}$  is the temperature of the environment gaining heat (K) Subsequent work by Jean Baptiste Joseph Fourier,<sup>9</sup> based on Newton's law of cooling, developed the law for steady-state heat conduction (known as Fourier's law<sup>10</sup>). Fourier's law is expressed here in differential form:

$$q = -k\nabla T \tag{5.2}$$

where q is the heat flux in watts per square meter (W/m<sup>2</sup>) k is the thermal conductivity of the material [W/(m\*K)]  $\nabla T$  is the temperature gradient (K/m)

# 2D Axisymmetric Heat Conduction Modeling

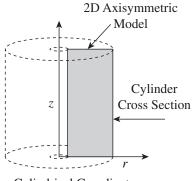
The following numerical solution model (cylinder conduction) was originally developed by COMSOL as a tutorial model based on an example from the NAFEMS collection.<sup>11</sup> It was developed for distribution with the Multiphysics software as a COMSOL Multiphysics General Heat Transfer Application Mode Model in the Heat Transfer Module Model Library. This model introduces two important basic concepts that apply to both applied physics and applied modeling: axisymmetric geometry (cylindrical) modeling and heat transfer modeling.

Note It is important for the new modeler to personally build each model presented within the text. There is no substitute in the path to an understanding of the modeling process for the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and subsequently correct errors, thereby adding to his or her experience and fund of modeling knowledge. Even building the simplest models will expand the modeler's fund of knowledge.

Heat transfer modeling is important in physical design and applied engineering problems. Typically, the modeler desires to understand heat generation during a process and to either add heat or remove heat to achieve or maintain a desired temperature. Figure 5.2 shows a 3D rendition of the 2D axisymmetric cylinder conduction geometry, as will be modeled here. The dashed-line ellipses in Figure 5.2 indicate the 3D rotation that would need to occur to generate the 3D solid object from the 2D cross section shown.

# 2D Axisymmetric Cylinder Conduction Model

This model is derived from the COMSOL cylinder conduction model. In this model, however, the selected thermally conductive solid is niobium (Nb).<sup>12,13</sup> Niobium has a variety of uses—as an alloying element in steels, as an alloying element in titanium turbine blades, in superconductors, as an anticorrosion coating, as an optical coating, and as an alloy in coinage.



Cylindrical Coordinates

**FIGURE 5.2** 3D rendition of the 2D axisymmetric cylinder conduction model

To start building the Cylinder\_Conduction\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Next select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.3. Click OK.

0 0	Model Navig	ator
<ul> <li>☐ Transient</li> <li>☐ Bioheat Equat</li> <li>☐ Weakly Comp</li> <li>☐ k-ɛ Turbulent</li> <li>☐ k-ω Turbulent</li> </ul>	Model Library User Mod Axial symmetry (2D)	
Element:	Lagrange - T <sub>2</sub> J <sub>1</sub>	Multiphysics

**FIGURE 5.3** 2D axisymmetric Cylinder\_Conduction\_1 Model Navigator setup

Name	Expression	Description
k_Nb	52.335[W/(m*K)]	Thermal conductivity Nb
rho_Nb	8.57e3[kg/m^3]	Density Nb
Cp_Nb	2.7e2[J/(kg*K)]	Heat capacity of Nb
T_0	2.7315e2[K]	Boundary temperature
q_0	5e5[W/m^2]	Heat flux

### Table 5.1 Constants Edit Window

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.1; also see Figure 5.4. Click OK.

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Since the settings in Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.08 and a height of 0.14. Select "Base: Corner" and set r equal to 0.02 and z equal to 0 in the Rectangle edit window. See Figure 5.5.

Click OK, and then click the Zoom Extents button. See Figure 5.6.

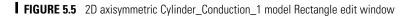
Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter the information shown in Table 5.2. See Figure 5.7.

Click OK. See Figure 5.8.

Name	Expression	Value	Description	
k Nb	52.335[W/(m*K)]	52.335[W/(m·K)] 8570[kg/m <sup>3</sup> ]	Thermal conductivity Nb	
rho_Nb	8.57e3[kg/m^3]		Density Nb	
Cp_Nb	2.7e2[J/(kg*K)]	270[J/(kg·K)]	Heat capacity	
T_0	2.7315e2[K]	273.15[K]	Boundary temperature	
q_0	5e5[W/m^2]	5e5[W/m <sup>2</sup> ]	Heat flux	

FIGURE 5.4 2D axisymmetric Cylinder\_Conduction\_1 model Constants edit window

Size		Rotat	ion angle-	
Width:	0.08	α:	0	(degrees
Height:	0.14			
Position				
Base:	Corner	Style:	Solid	\$
r:	0.02	Name:	R1	
z:	0			



	10			-									
0.14				a come a come a second	9		.0						
0,12	<b>.</b>												
0.1													
0.08					83								
0.06													
0.04													
0.02	•												
0	0.0	12	8	r=0						S.,	2	22	12
	-0.05	-0.04	-0.02	0	0.02	0.04	0.06	0.08	0.1	0.12	0.14	0.16	0.18

**FIGURE 5.6** 2D axisymmetric Cylinder\_Conduction\_1 model cylinder rectangle

# Table 5.2Point Edit Window

Name	r Location	z Location
Point 1	0.02	0.04
Point 2	0.02	0.1

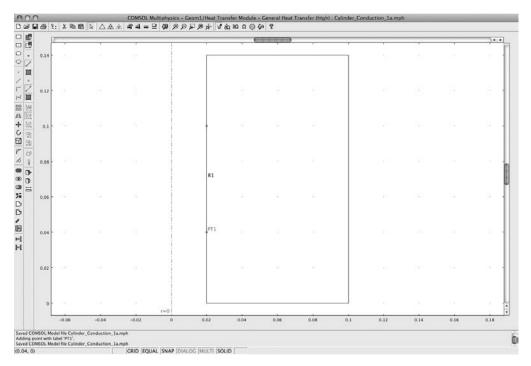
Coor	dinates	
r: 0	.02 0.02	ОК
z: 0	.04 0.1	Cancel
Name:	PT1	Apply

FIGURE 5.7 2D axisymmetric Cylinder\_Conduction\_1 model Point edit window

**NOTE** Two points have been added on the interior (small r value) boundary of the rectangle (cylinder cross section) to define the upper (larger z value) and lower (smaller z value) bounds of the heat-flux application region.

# **Physics Subdomain Settings: General Heat Transfer**

Having established the geometry for the 2D axisymmetric Cylinder\_Conduction\_1 model (a rectangle with two points on the boundary), the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain).



**FIGURE 5.8** 2D axisymmetric Cylinder\_Conduction\_1 model rectangle with points

Name	Expression	Description
k (isotropic)	k_Nb	Thermal conductivity
ρ	rho_Nb	Density
CP	Cp_Nb	Heat capacity

Table 5.3	Subdomain	Edit Windows
	oasaomam	

In the Subdomain edit windows, enter the information shown in Table 5.3; also see Figure 5.9. Click OK.

For static and quasi-static calculations, the only physical property value required for the conduction calculation is k (k\_Nb). From the point of view of physical consistency, however, the density (rho\_Nb) and the heat capacity (Cp\_Nb) should be included as well. If *C*p and rho are set to zero, the implication is the model includes a perfect vacuum, which is logically inconsistent with the stated value of k. Also, by including the values for *C*p and rho in this location, they are conveniently available should the modeler wish to modify the model for transient analysis.

quation '·(-k⊽T) = Q = temperature		
Subdomains Groups	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:	sure
Group: 🔅 🛟		

FIGURE 5.9 2D axisymmetric Cylinder\_Conduction\_1 model Subdomain Settings edit window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 4	Thermal insulation	_	5.10
2, 5, 6	Temperature	T_0	5.11
3	Heat flux	q_0	5.12

Table 5.4	<b>Boundary Settings</b>	-General Heat T	ransfer Edit Window
-----------	--------------------------	-----------------	---------------------

## **Physics Boundary Settings: General Heat Transfer**

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select and/or enter the given boundary condition and value as shown in Table 5.4. Click OK. See Figures 5.10, 5.11, and 5.12.

### **Mesh Generation**

On the toolbar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 2200 elements. See Figure 5.13.

Boundary Condi	tion Highly Conduction	ve Layer	Element Color/Style ]
Boundary sources and	l constraints		
Library coefficient: (	:	Load	
Boundary condition:	Thermal insulation	•	
Quantity	Value/Expression	Unit	Description
۹ <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
T <sub>inf</sub>	273.15	к	External temperature
Τ <sub>0</sub>	273.15	к	Temperature
Radiation type:	None	\$	
E	0	1	Surface emissivity
T <sub>amb</sub>	0	к	Ambient temperature
Jo	ensilon htab*siama h	W/m <sup>2</sup>	Surface radiosity expression
Member of group(s):		1	
	Boundary sources and Library coefficient: Boundary condition: Quantity q <sub>0</sub> h T <sub>inf</sub> T <sub>0</sub> Radiation type: ε T <sub>amb</sub> J <sub>0</sub>	Boundary sources and constraints         Library coefficient:       Thermal insulation         Quantity       Value/Expression         q0       0         h       0         Tinf       273.15         Tq       273.15         Radiation type:       None         ε       0         Tamb       0         J₀       epsilon_htgh*sigma_h	Boundary sources and constraints         Library coefficient:       Load         Boundary condition:       Thermal insulation       Load         Quantity       Value/Expression       Unit         Qo       0       W/m²       W/m²         h       0       W/m²       K         Tinf       273.15       K         Radiation type:       None       I         ε       0       I         Tamb       0       K         Jo       epsilon_htgh*sigma_h       W/m²

**FIGURE 5.10** 2D axisymmetric Cylinder\_Conduction\_1 model Boundary Settings (1, 4) edit window

	Boundary Settings – G	eneral Heat Transfer	(htgh)	
uation				
= T <sub>0</sub>				
Boundaries Groups	Boundary Cond	ition Highly Conductiv	e Layer	Element Color/Style
Boundary selection	Boundary sources and	d constraints	11	
1	Library coefficient:	•	Load	
3	Boundary condition:	( <b>-</b>		
4 5			Unit	Description
5	Quantity q <sub>0</sub>	Value/Expression	W/m <sup>2</sup>	Description
	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
	T <sub>inf</sub>	273.15	к	
	To		к	External temperature
		T_0		Temperature
	Radiation type:	None	\$	
	ε	0	1	Surface emissivity
iroup:	Tamb	0	к	Ambient temperature
Select by group	J <sub>0</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

**FIGURE 5.11** 2D axisymmetric Cylinder\_Conduction\_1 model Boundary Settings (2, 5, 6) edit window

## Solving the 2D Axisymmetric Cylinder\_Conduction\_1 Model

Using the menu bar, select Solve > Solve Problem. The COMSOL Multiphysics software automatically selects the Stationary Solver.

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, always change the chosen solver or the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Figure 5.14 shows the modeling solution results obtained using single-value parameters with the default solver (UMFPACK).

### Parametric Solving of the 2D Axisymmetric Cylinder\_Conduction\_1 Model

Now that the model has been built, it is relatively easy to expand the model to calculate other quasi-static solutions.

quation				
$\mathbf{n} \cdot (-k\nabla T) = \mathbf{q}_0 + \mathbf{h}(T_{inf} - T)$				
Boundaries Groups	Boundary Condi	tion Highly Conduction	ve Layer	Element Color/Style
Boundary selection	Boundary sources and	constraints		
1	Library coefficient:	•	Load )	
2	Boundary condition:			
4		Heat flux Value/Expression	Unit	Description
6	Quantity Q <sub>0</sub>	q_0	W/m <sup>2</sup>	Inward heat flux
	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
	T <sub>inf</sub>	273.15	к	External temperature
	то	273.15	к	
	Radiation type:	(BARGAR)	_	Temperature
		None	•	
	E	0	1	Surface emissivity
Group:	Tamb	0	к	Ambient temperature
Select by group	J <sub>0</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

**FIGURE 5.12** 2D axisymmetric Cylinder\_Conduction\_1 model Boundary Settings (3) edit window

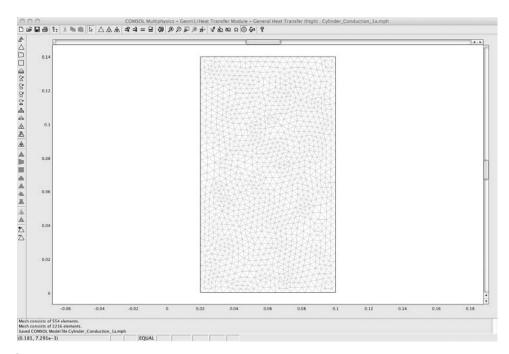


FIGURE 5.13 2D axisymmetric Cylinder\_Conduction\_1 model Mesh window

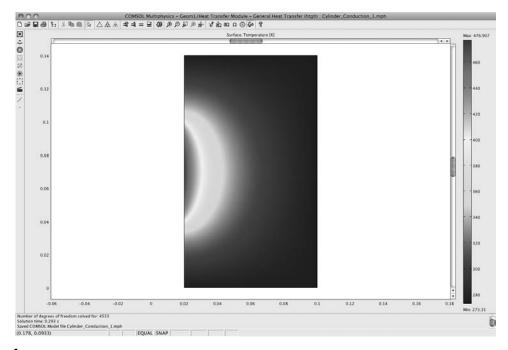


FIGURE 5.14 2D axisymmetric Cylinder\_Conduction\_1 model using the default solver

**NOTE** This time, instead of using the default solver, the model is run using multiple value parameters in the Parametric Solver (UMFPACK) as the initial solver. The Parametric Solver (UMFPACK) results include not only the default solution, but also solutions at a number of other values of heat flux  $(q_0)$ .

Using the menu bar, select File > Save as. Enter Cylinder\_Conduction\_1p.

From the menu bar, select File > Reset Model >Yes. On the menu bar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 2200 elements.

From the menu bar, select Solve > Solver Parameters > Parametric. Enter  $q_0$  in the Parameter name edit window. Enter 0:5e3:5e5 in the Parameter values edit window. See Figure 5.15. Click OK.

From the menu bar, select Solve > Solve Problem. See Figure 5.16.

### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.17.

Analysis	General Pa	rametric	Stationary	Adaptive	Advanced
Stationary			Stationary	Produptive	Huvaneeu
Auto select solver	Parameter Parameter name:		q_0		
tationary Time dependent	Parameter values:		0:5e3:5e	5	
genvalue arametric rationary segregated arametric segregated	Linear system solver Linear system solver: Preconditioner:	Direct (	UMFPACK)	:	
Adaptive mesh refinement					Setting
	Matrix symmetry:	Automa	itic	¢	

I FIGURE 5.15 2D axisymmetric Cylinder\_Conduction\_1p model Solver Parameters edit window

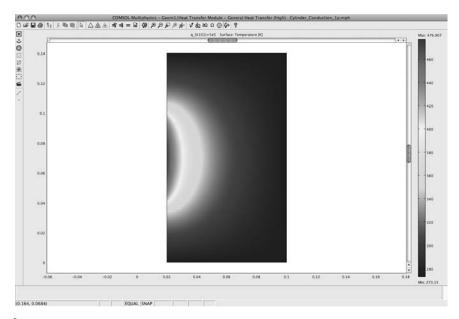


FIGURE 5.16 2D axisymmetric Cylinder\_Conduction\_1p model using the Parametric Solver (UMFPACK)

### 240 Chapter 5 2D Axisymmetric Modeling

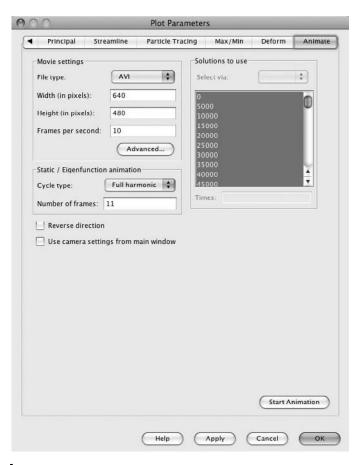


FIGURE 5.17 2D axisymmetric Cylinder\_Conduction\_1p model Plot Parameters window

Click the Start Animation button. See Figure 5.18.

Alternatively, you can play the file Movie5\_CC\_1p.avi that was supplied with this book.

## First Variation on the 2D Axisymmetric Cylinder Conduction Model

This model is derived from the COMSOL cylinder conduction model. This model, however, is built using the basic COMSOL Multiphysics software package, instead of the Heat Transfer Module. The selected thermally conductive solid is, as in the initial model, niobium (Nb). The modeler should note, as mentioned previously, that for static and quasi-static calculations, the only physical property value required for the thermal conduction calculation is k (k\_Nb), the thermal conductivity. That property is the only one that will be used in this model.

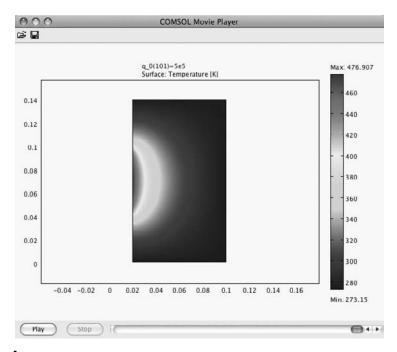


FIGURE 5.18 2D axisymmetric Cylinder\_Conduction\_1p model animation, final frame

To start building the Cylinder\_Conduction\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select COMSOL Multiphysics > Heat Transfer > Conduction > Steady-state analysis. See Figure 5.19. Click OK.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.5; also see Figure 5.20. Click OK.

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.08 and a height of 0.14. Select "Base: Corner" and set r equal to 0.02 and z equal to 0 in the Rectangle edit window. See Figure 5.21.

Name	Expression	Description
k_Nb	52.335[W/(m*K)]	Thermal conductivity Nb
T_0	2.7315e2[K]	Boundary temperature
q_0	5e5[W/m^2]	Heat flux

Table 5.5         Constants Edit Window
---

2	IVIO	del Navigator	8		_
New	Model Library	User Models	Open	Settings	
Space dimension:	Axial symmetry ()	2D)	Heat	Transfer	
<ul> <li>Application Modes</li> <li>COMSOL Multiple</li> <li>Acoustics</li> <li>Convection at Convection at Electromagn</li> </ul>	and Diffusion		4		
▼ 📑 Conduct	r on and Conductior on y-state analysis		heat flux,	n: ifer through conduct convective, and tem conditions.	
Trans		*		ate analysis in 2D ax	ial
Dependent variables	т				
Application mode name:	ht				
Element:	Lagrange - Qu	adra 🛟	$\square$	Multiphysics	

**FIGURE 5.19** 2D axisymmetric Cylinder\_Conduction\_2 Model Navigator setup

000		Constants			
Name	Expression	Value	Description		
k_Nb	52.335[W/(m*K)]	52.335[W/(m·K)]	Thermal conductivity Nb		
T_0	2.7315e2[K]	273.15[K]	Boundary temperature		
q_0	5e5[W/m^2]	5e5[W/m <sup>2</sup> ]	Heat flux		
c		Help Apr	oly Cancel OK		

**FIGURE 5.20** 2D axisymmetric Cylinder\_Conduction\_2 model Constants edit window

Size		Rota	tion angle-	
Width:	0.08	α:	0	(degrees)
Height:	0.14			
Position				
Base:	Corner	\$ Style:	Solid	\$
r:	0.02	Name:	R1	
z:	0			

FIGURE 5.21 2D axisymmetric Cylinder\_Conduction\_2 model Rectangle edit window

Click OK, and then click the Zoom Extents button. See Figure 5.22.

Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter the information shown in Table 5.6. See Figure 5.23.

Click OK. See Figure 5.24.

Two points have been added, as in the earlier model, on the interior (small r value) boundary of the rectangle (cylinder cross section) to define the upper (larger z value) and lower (smaller z value) bounds of the heat-flux application region.

1				
11 m 1 m 1 m 1 m 1				
1				

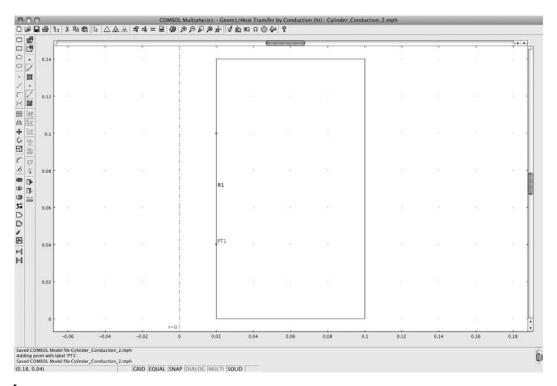
**FIGURE 5.22** 2D axisymmetric Cylinder\_Conduction\_2 model cylinder rectangle

## Table 5.6 Point Edit Window

Name	r Location	z Location
Point 1	0.02	0.04
Point 2	0.02	0.1

Coord	inates	
r: 0.	02 0.02	Сок
z: 0.	04 0.1	Cancel
Name:	PT1	Apply

**FIGURE 5.23** 2D axisymmetric Cylinder\_Conduction\_2 model Point edit window



**FIGURE 5.24** 2D axisymmetric Cylinder\_Conduction\_2 model rectangle with points

Table 5.7 Subdomain Edit Wind
-------------------------------

Name	Expression	Description
k (isotropic)	k_Nb	Thermal conductivity

## **Physics Subdomain Settings: General Heat Transfer**

Having established the geometry for the 2D axisymmetric Cylinder\_Conduction\_2 model (a rectangle with two points on the boundary), the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). In the Subdomain edit windows, enter the information shown in Table 5.7; also see Figure 5.25. Click OK.

**NOTE** For static and quasi-static calculations, the only physical property value required for the calculation is k (k\_Nb).

## **Physics Boundary Settings: General Heat Transfer**

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.8. Click OK. See Figures 5.26, 5.27, and 5.28.

Subdomains Groups	6		nent Color
Subdomain selection	Thermal properties a Library material: Quantity k (isotropic)	Value/Expression	Load Unit Description W/(m·K) Thermal conductivity
Group: 🗘	O k (anisotropic) Q	400 0 0 400 0	W/(m·K) Thermal conductivity W/m <sup>3</sup> Heat source

FIGURE 5.25 2D axisymmetric Cylinder\_Conduction\_2 Model Subdomain Settings edit window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 4	Thermal insulation	_	5.26
2, 5, 6	Temperature	T_0	5.27
3	Heat flux	q_0	5.28

### Table 5.8 Boundary Settings–General Heat Transfer Edit Window

## **Mesh Generation**

On the toolbar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 2200 elements. See Figure 5.29.

## Solving the 2D Axisymmetric Cylinder\_Conduction\_2 Model

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, always change the chosen solver and the parametric settings. This time, instead of using the default solver, the model is run using the Parametric Solver (UMFPACK) as the initial solver. The Parametric Solver (UMFPACK) results include not only the default solution, but also solutions at a number of other values of heat flux ( $q_0$ ).

Boundaries Groups		Coefficients	Color/Style	]
Boundary selection	Boundary sources an	d constraints		-
1	Boundary condition:	Thermal insulation	\$	
3	Quantity	Value/Expression	Unit	Description
4 5	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
6	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
	T <sub>inf</sub>	0	к	External temperature
Group:	Const	0	W/(m <sup>2</sup> ·K <sup>4</sup> )	Problem-dependent constan
Select by group	T <sub>amb</sub>	0	к	Ambient temperature
Interior boundaries	То	0	— к	Temperature

FIGURE 5.26 2D axisymmetric Cylinder\_Conduction\_2 model Boundary Settings (1, 4) edit window

= T <sub>0</sub>				
Boundaries Groups		Coefficients	Color/Style	]
Boundary selection	Boundary sources an	d constraints		
1 2	Boundary condition:	Temperature	\$	
3	Quantity	Value/Expression	Unit	Description
5	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
6	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
	T <sub>inf</sub>	0	к	External temperature
Group:	Const	0	$W/(m^2 \cdot K^4)$	Problem-dependent constant
Select by group	T <sub>amb</sub>	0	к	Ambient temperature
Interior boundaries	то	T_0	к	Temperature

**FIGURE 5.27** 2D axisymmetric Cylinder\_Conduction\_2 model Boundary Settings (2, 5, 6) edit window

Using the menu bar, select Solve > Solver Parameters > Parametric. Enter  $q_0$  in the Parameter name edit window. Enter 0:5e3:5e5 in the Parameter values edit window. See Figure 5.30. Click OK.

From the menu bar, select Solve > Solve Problem. See Figure 5.31.

Boundaries Groups		Coefficients	Color/Style	]
Boundary selection	Boundary sources and	d constraints		_
1 2	Boundary condition:	Heat flux	\$	
3	Quantity	Value/Expression	Unit	Description
4	q <sub>0</sub>	q_0	W/m <sup>2</sup>	Inward heat flux
6	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
	T <sub>inf</sub>	0	к	External temperature
Group:	Const	0	W/(m <sup>2</sup> ·K <sup>4</sup> )	Problem-dependent constan
Select by group	T <sub>amb</sub>	0	к	Ambient temperature
Interior boundaries	то	0	к	Temperature

FIGURE 5.28 2D axisymmetric Cylinder\_Conduction\_2 model Boundary Settings (3) edit window

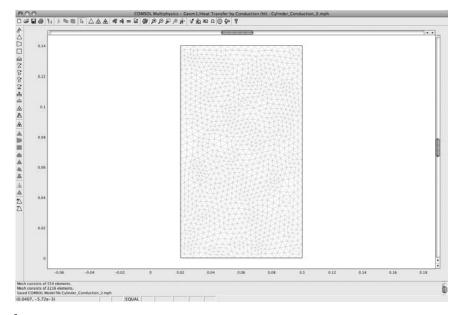
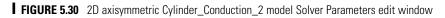


FIGURE 5.29 2D axisymmetric Cylinder\_Conduction\_2 model mesh window

General Par Parameter Parameter name: Parameter values:	rametric Si		otive Advance	ed
Parameter name:				
Parameter values:		q_0		
		0:5e3:5e5		
Linear system solver	Direct (UMF	PACK)		
Preconditioner:			:	
			Set	tings)
Matrix symmetry:	Automatic		•	
	Halp	Analy	( Cancel )	( ок
	Linear system solver: Preconditioner:	Linear system solver: Direct (UMF Preconditioner:	Linear system solver: Direct (UMFPACK) Preconditioner: Matrix symmetry: Automatic	Linear system solver: Direct (UMFPACK)



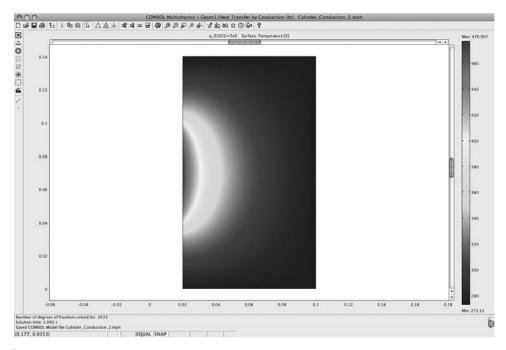


FIGURE 5.31 2D axisymmetric Cylinder\_Conduction\_2 model using the Parametric Solver (UMFPACK)

### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.32.

Click the Start Animation button. See Figure 5.33.

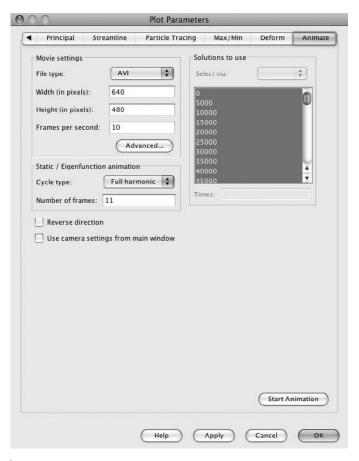
Alternatively, you can play the file Movie5\_CC\_2.avi that was supplied with this book.

### Comparison of Cylinder Conduction Models 1p and 2

As can be readily seen in Table 5.9, the calculated values for Cylinder Conduction Models 1p and 2 are exactly the same for the simple conduction calculation, as would be expected. The advantage of using the Heat Transfer Module, as needed, is that it can accommodate more complex physics. See Figures 5.34 and 5.35.

### Table 5.9 Comparison of T-max for Cylinder Conduction Models 1p and 2

Model Number	Module Used	T-max	Figure Number
1р	Heat Transfer Module	476.907 K	5.34
2	Basic Heat Transfer	476.907 K	5.35

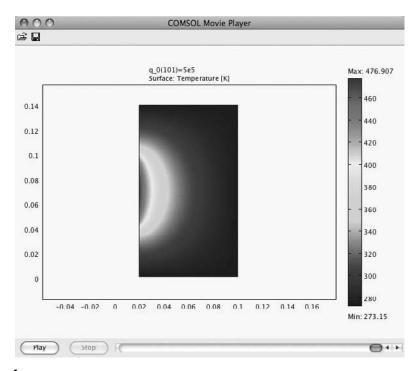


**FIGURE 5.32** 2D axisymmetric Cylinder\_Conduction\_2 model Plot Parameters window

# Second Variation on the 2D Axisymmetric Cylinder Conduction Model, Including a Vacuum Cavity

**NOTE** This model is derived from the COMSOL cylinder conduction model. In this model, the selected thermally conductive solid is niobium (Nb).<sup>12,13</sup> A vacuum cavity has been added to the cylinder geometry. With the added vacuum cavity, the modeler can explore some of the additional heat transfer modeling capabilities of the Heat Transfer Module. Vacuum isolation is a valuable tool in lowering heat loss in modern machines.

To start building the Cylinder\_Conduction\_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.36. Click OK.



**FIGURE 5.33** 2D axisymmetric Cylinder\_Conduction\_2 model animation, final frame

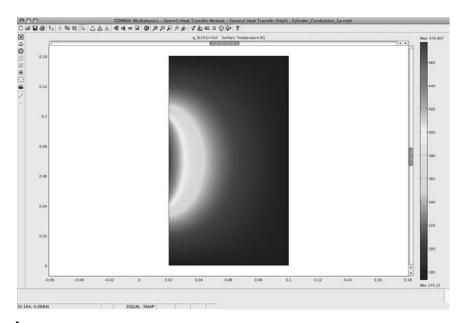
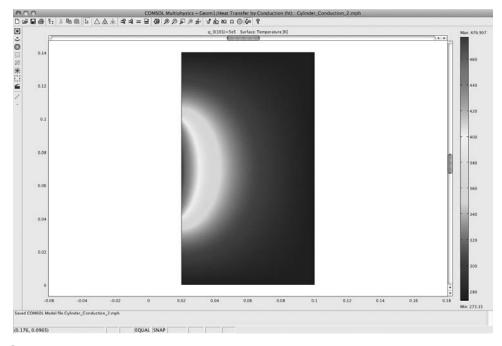
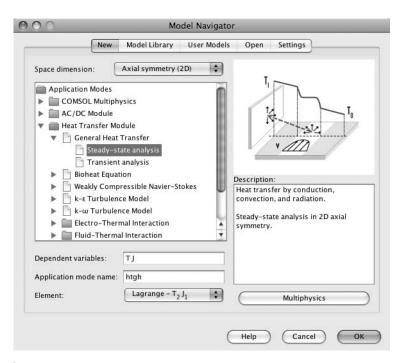
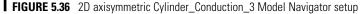


FIGURE 5.34 2D axisymmetric Cylinder\_Conduction\_1p model, final frame



**FIGURE 5.35** 2D axisymmetric Cylinder\_Conduction\_2 model, final frame





Name	Expression	Description
k_Nb	52.335[W/(m*K)]	Thermal conductivity Nb
rho_Nb	8.57e3[kg/m^3]	Density Nb
Cp_Nb	2.7e2[J/(kg*K)]	Heat capacity Nb
T_0	2.7315e2[K]	Boundary temperature
q_0	5e5[W/m^2]	Heat flux
p_0	1.33e-7[Pa]	Pressure in vacuum

## Table 5.10 Constants Edit Window

## Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.10; also see Figure 5.37. Click OK.

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Create the two rectangles indicated in Table 5.11.

Rectangle Number	Width	Height	Base	r	Z	Figure Number
1	0.08	0.14	Corner	0.02	0	5.38
2	0.002	0.139	Corner	0.06	0.0005	5.39

## Table 5.11 Rectangle Edit Window

Name	Expression	Value	Description	
k_Nb	52.335[W/(m*K)]	52.335[W/(m·K)]	Thermal conductivity Nb	
rho_Nb	8.57e3[kg/m^3]	8570[kg/m <sup>3</sup> ]	Density Nb	
Cp_Nb	2.7e2[J/(kg*K)]	270[J/(kg·K)]	Heat capacity Nb	
T_0	2.7315e2(K)	273.15[K]	Boundary temperature	
q_0	5e5[W/m^2]	5e5[W/m <sup>2</sup> ]	Heat flux	
p_0	1.33e-7[Pa]	(1.33e-7)[Pa]	Pressure in vacuum	

FIGURE 5.37 2D axisymmetric Cylinder\_Conduction\_3 model Constants edit window

) 0 0		Rectar	ngle	_	_
Size			Rotat	ion angle	
Width:	0.08		α:	0	(degrees)
Height:	0.14				
Position					
Base:	Corner	\$	Style:	Solid	\$
rs	0.02		Name:	R2	
	0				

I FIGURE 5.38 2D axisymmetric Cylinder\_Conduction\_3 model Rectangle edit window (1)

Click OK, and then click the Zoom Extents button. See Figure 5.40. Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter the information shown in Table 5.12. See Figure 5.41.

Click OK. See Figure 5.42

NOTE Two points have been added on the interior (small r value) boundary of the rectangle (cylinder cross section) to define the upper (larger z value) and lower (smaller z value) bounds of the heat-flux application region.

Height: 0.139 Position Base: Corner \$ Style: Solid	_
Height: 0.139 Position Base: Corner 🛟 Style: Solid	
Position Base: Corner 🛟 Style: Solid	egrees)
r: 0.06 Name: R2	\$
z: 0.0005	

**FIGURE 5.39** 2D axisymmetric Cylinder\_Conduction\_3 model Rectangle edit window (2)

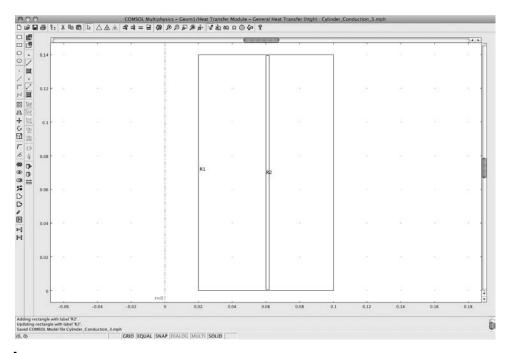


FIGURE 5.40 2D axisymmetric Cylinder\_Conduction\_3 model cylinder rectangles (1 and 2)

Using the menu bar, select Draw > Create Composite Object. Enter R1 + R2 in the Set formula edit window. Verify or check the Keep interior boundaries check box. See Figure 5.43.

Click OK. See Figure 5.44

Table 5.12 Point Edit Window

Name	r Location	z Location
Point 1	0.02	0.04
Point 2	0.02	0.1

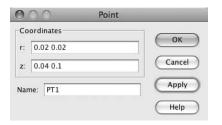


FIGURE 5.41 2D axisymmetric Cylinder\_Conduction\_3 model Point edit window

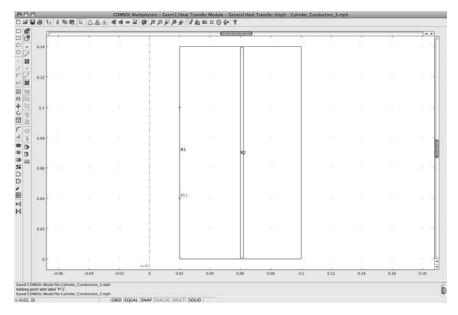


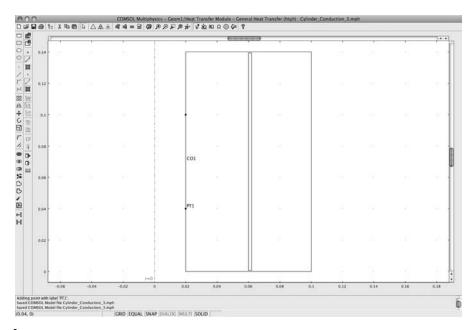
FIGURE 5.42 2D axisymmetric Cylinder\_Conduction\_3 model rectangles with points

## **Physics Subdomain Settings: General Heat Transfer**

Having established the geometry for the 2D axisymmetric Cylinder\_Conduction\_3 model (a rectangle with two points on the boundary), the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select "subdomain 1" in the Subdomain selection window. In the Subdomain edit windows, enter the information shown in Table 5.13. See Figure 5.45.

OOO Crea	ate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: R1 R2	Set formula: R1+R2 Keep interior boundaries Repair Repair tolerance: 1.0E-4

**FIGURE 5.43** 2D axisymmetric Cylinder\_Conduction\_3 model Create Composite Object edit window



**FIGURE 5.44** 2D axisymmetric Cylinder\_Conduction\_3 model composite object

·(-kVT) = Q = temperature		
Subdomains Groups Subdomain selection	Conduction       Convection       Ideal Cas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:	pressur

**FIGURE 5.45** 2D axisymmetric Cylinder\_Conduction\_3 model Subdomain Settings (1) edit window

Name	Expression	Description
k (isotropic)	k_Nb	Thermal conductivity
ρ	rho_Nb	Density
C <sub>P</sub>	Cp_Nb	Heat capacity

Table 5.13	Subdomain	Edit Window
Table 5.13	Subdomain	East window

Select "subdomain 2" in the Subdomain selection window. Click the Library material Load button. Select Liquids and Gases > Gases > Air, I atm. Click OK.

Enter the term  $p_0$  in place of p in the expression rho(p...in the Density edit window. Click on the Opacity pull-down list. Select "Transparent." See Figure 5.46. Click OK.

**NOTE** The insertion of  $p_0$  into the density function for air sets the pressure in the vacuum cavity. The selection of "Transparent" allows energy transfer by radiation.

uation (-k⊽T) = Q	
= temperature	
Subdomains Groups	Conduction Convection Ideal Gas Init Element Color
Subdomain selection	Control       Contro       Control       Control
Group: 🔅 🛟	

FIGURE 5.46 2D axisymmetric Cylinder\_Conduction\_3 model Subdomain Settings (2) edit window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 4	Thermal insulation	—	5.47
2, 5,10	Temperature	T_0	5.48
3	Heat flux	q_0	5.49

Table 5.14	<b>Boundary Settin</b>	gs–General Heat 1	Fransfer Edit Window
------------	------------------------	-------------------	----------------------

For static and quasi-static calculations, the only physical property value required for the conduction calculation is k (k\_Nb). However, from the point of view of physical consistency, the density (rho\_Nb) and the heat capacity (Cp\_Nb) should be included. If *C*p and rho are set to zero, the implication is that of a perfect vacuum, which is logically inconsistent with the stated value of k. Also, by including the values for *C*p and rho in this location, they are conveniently available should the modeler wish to modify the model for transient analysis.

## **Physics Boundary Settings: General Heat Transfer**

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.14. Click OK. See Figures 5.47, 5.48, and 5.49.

Boundaries Groups	Boundary Cond	ition Highly Conducti	re Layer	Element Color/Style
Boundary selection	Boundary sources and	d constraints		
2	Library coefficient:	•	Load )	
3	Boundary condition:	Thermal insulation	•	
5	Quantity	Value/Expression	Unit	Description
6 7	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
5	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
10	T <sub>inf</sub>	273.15	ĸ	External temperature
	Τ <sub>0</sub>	273.15	к	Temperature
	Radiation type:	None	0	
	E	٥	1	Surface emissivity
Group:	T <sub>amb</sub>	0	к	Ambient temperature
Select by group	l <sub>o</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

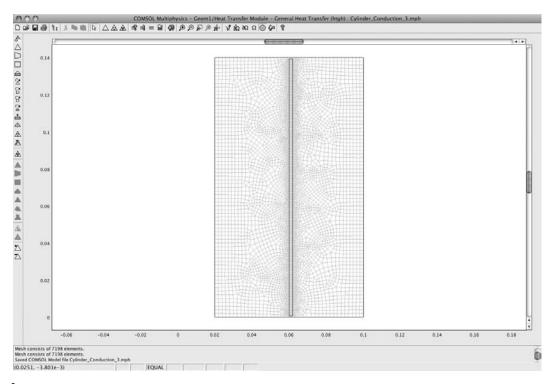
**FIGURE 5.47** 2D axisymmetric Cylinder\_Conduction\_3 model Boundary Settings (1, 4) edit window

Equation T = T <sub>0</sub>	Boundary Settings – C	ieneral Heat Transfer	(ntgn)	
Boundaries Groups	Boundary Cond	tion Highly Conducti	ve Layer	Element Color/Style
Boundary selection	-Boundary sources and			
2	Library coefficient:	•	Load)	
3 4	Boundary condition:	Temperature	•	
5		Value/Expression	Unit	Description
7	a <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
8	h	0	W/(m <sup>2</sup> K)	Heat transfer coefficient
10	Tinf	273.15	к	External temperature
	т <sub>о</sub>	T_0	к	Temperature
	Radiation type:	None	٥	
	ε	0	1	Surface emissivity
Group:	Tamb	0	К	Ambient temperature
	J <sub>0</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Select by group	Member of group(s):		1	
Interior boundaries	Member of group(s):		1	

FIGURE 5.48 2D axisymmetric Cylinder\_Conduction\_3 model Boundary Settings (2, 5, 10) edit window

Boundaries Groups	Boundary Cond	tion Highly Conducti	ve Layer	Element Color/Style
Boundary selection	Boundary sources and	l constraints		
1 2	Library coefficient:		Load	
3	Roundary condition:	Heat flux	0	
5	Quantity	Value/Expression	Unit	Description
6 7	q <sub>0</sub>	q_0	W/m <sup>2</sup>	Inward heat flux
8	h	0	W/(m <sup>2</sup> K)	Heat transfer coefficient
10	T <sub>inf</sub>	273.15	к	External temperature
	τ <sub>o</sub>	273.15	к	Temperature
	Radiation type:	None	•	
	٤	0	1	Surface emissivity
Group:	Tamb	0	к	Ambient temperature
Select by group	J <sub>0</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

I FIGURE 5.49 2D axisymmetric Cylinder\_Conduction\_3 model Boundary Settings (3) edit window



**FIGURE 5.50** 2D axisymmetric Cylinder\_Conduction\_3 model mesh

## **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters > Subdomain 1. Enter Maximum element size 0.002. Select Method > Quad. Click the Remesh button. Select Free Mesh Parameters > Subdomain 2. Enter Maximum element size 0.0005. Select Method > Quad. Click the Remesh button.

Click OK. See Figure 5.50.

## Solving the 2D Axisymmetric Cylinder\_Conduction\_3 Model

From the menu bar, select Solve > Solver Parameters > Parametric. Enter  $q_0$  in the Parameter name edit window. Enter 0:5e3:5e5 in the Parameter values edit window. See Figure 5.51. Click OK.

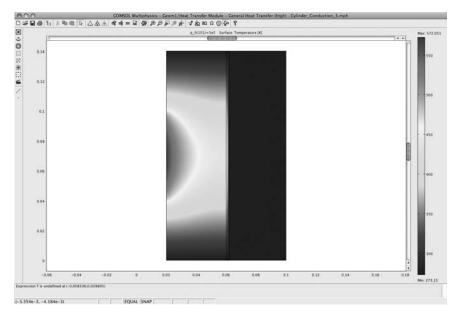
From the menu bar, select Solve > Solve Problem. See Figure 5.52.

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.53.

000	Solver Paramet	ers
Analysis	General Parametr	ic Stationary Adaptive Advanced
Stationary	Parameter Parameter name:	q_0
Adaptive mesh refinement	Parameter values: Linear system solver Linear system solver: Dira Preconditioner:	0:5e3:5e5
	Matrix symmetry: Aut	omatic
	(	elp Apply Cancel O

FIGURE 5.51 2D axisymmetric Cylinder\_Conduction\_3 model Solver Parameters edit window



I FIGURE 5.52 2D axisymmetric Cylinder\_Conduction\_3 model using the Parametric Solver (UMFPACK)

Principal Str	eamline Particle Tra	acing Max/Min	Deform Anima
Movie settings		Solutions to use	
File type:	AVI	Select via:	\$
Width (in pixels):	640	lo	-
Height (in pixels):	480	5000	U
		10000 15000	
Frames per second:	10	20000	
	Advanced	25000	
Charles ( Einenformetie		30000 35000	
Static / Eigenfunctio		40000	<b>A</b>
Cycle type:	Full harmonic	45000 Times:	
Number of frames:	11	Times:	

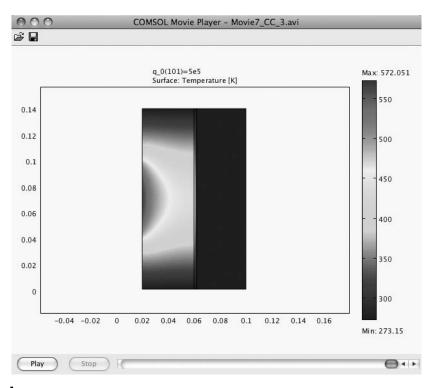
FIGURE 5.53 2D axisymmetric Cylinder\_Conduction\_3 model Plot Parameters window

Click the Start Animation button. See Figure 5.54.

Alternatively, you can play the file Movie5\_CC\_3.avi that was supplied with this book.

# 2D Axisymmetric Cylinder Conduction Models: Summary and Conclusions

The models presented in this section of Chapter 5 have introduced the following new concepts: two-dimensional axisymmetric modeling (axial symmetry [2D]), cylindrical coordinates, conductive media DC, Heat Transfer Module, heat conduction theory, opaque and transparent thermally conductive materials, and vacuum. Previously introduced concepts employed in these models include triangular mesh, free mesh parameters, subdomain mesh, maximum element size, and quadrilateral mesh (quad).



**FIGURE 5.54** 2D axisymmetric Cylinder\_Conduction\_3 model animation, final frame

A comparison of the calculated results for the three cylinder conduction models is shown in Table 5.15. As can be readily observed, the presence of a vacuum cavity significantly reduces the rate of heat flow through the model and raises the equilibrium temperature at the surface receiving the heat flux.

For simple heat transfer models, both the basic COMSOL Multiphysics software and the Heat Transfer Module yield the same result, as would be expected. For more complex models involving such conditions as a vacuum, the Heat Transfer Module is required.

 $\Delta T$ 

0

95.144

298.90 °C

572.051 K

Model Number	Module Used	Vacuum	T-max (K)	T-max (°C)
1p	Heat Transfer Module	No	476.907 K	203.76 °C
2	Basic Heat Transfer	No	476.907 K	203.76 °C

Yes

#### Table 5.15 Cylinder Conduction Modeling Results Summary

Heat Transfer Module

3

## 2D Axisymmetric Insulated Container Design

Sir James Dewar<sup>14</sup> invented the vacuum flask in 1892. The vacuum flask enabled him to store low-temperature liquified gases for longer periods of time. Being a research scientist, his primary concern was the study of the liquification process for gases and the study of the resulting liquids.

**NOTE** Prior to the mid-1900s, it was uncommon for research scientists to patent or commercialize new inventions, regardless of their potential commercial or economic impact. The usual process was to disclose new findings through letter publication to a learned society.<sup>15</sup>

The term "Thermos"<sup>16</sup> came into existence in 1904, when a German company was formed under the name Thermos GmbH to commercialize the vacuum flask technology. The vacuum flask (invented by Dewar) has come into widespread common usage by both scientists and nonscientists alike, under the name "thermos" or "thermos bottle." As such, the name "thermos" has, through common usage, become the generic name, in the United States and some other countries, for the vacuum flask or thermos bottle. There are, of course, other insulating materials in use that are not quite as efficient as the vacuum flask but nevertheless adequate. Thus some thermos bottles (vacuum flask containers) have no vacuum, but simply a low-thermal-conductivity solid (insulating material) in the place where the vacuum would normally exist.

## 2D Axisymmetric Thermos\_Container Model

This model is derived from the COMSOL thermos laminar flow and thermos laminar hcoeff models. Those models can be found in the Tutorial Models folder of the Heat Transfer Module Model Library. In this model, Thermos\_Container\_1, the walls of the flask (e.g., bottle, tank) are formed of stainless steel. In the 2D axisymmetric Thermos\_Container\_1 model, the selected thermal insulating solid is rigid urethane foam.<sup>17</sup> In the first variation on the 2D axisymmetric Thermos\_Container model, a vacuum cavity replaces the urethane foam. In the second variation on the 2D axisymmetric Thermos\_Container model, a glass<sup>18</sup> material replaces the stainless steel<sup>19</sup> wall material and the insulating vacuum cavity remains. These changes in the materials design of these models reflect some of the typical alterations and trade-offs that need to be made in the exploratory design phase of a new artifact (e.g., product, tool).

To start building the Thermos\_Container\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.55. Click OK.

0	Model Navigat	or
<ul> <li>☐ Transient</li> <li>☐ Bioheat Equat</li> <li>☐ Weakly Comp</li> <li>☐ K-∈ Turbulen</li> <li>☐ k-∞ Turbuler</li> </ul>	Model Library User Model Axial symmetry (2D) ysics dule Transfer ate analysis tion ressible Navier–Stokes ce Model	s Open Settings
🕨 🛅 k–ω Turbuler	nce Model mal Interaction	convection, and radiation. Steady-state analysis in 2D axial symmetry. Multiphysics

**FIGURE 5.55** 2D axisymmetric Thermos\_Container\_1 Model Navigator setup

### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.16; also see Figure 5.56. Click OK.

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

When building a model, it is usually best to choose names for modeler-defined parameters (e.g., constants, scalar functions) that are easily recalled and associated with the function that they provide in the model (e.g., k\_foam, rho\_foam).

Name	Expression	Description
k_foam	9e-2[W/(m*K)]	Thermal conductivity foam
rho_foam	6e2[kg/m^3]	Density foam
Cp_foam	1.4e3[J/(kg*K)]	Heat capacity foam
k_304ss	1.62e1[W/(m*K)]	Thermal conductivity 304ss
rho_304ss	8e6[kg/m^3]	Density 304ss
Cp_304ss	5.0e2[J/(kg*K)]	Heat capacity 304ss
p_0	1.0[atm]	Air pressure
T_0	2.7315e2[K]	Boundary temperature
T_L	9.5e1[degC]	Liquid temperature
L_wall	0.35[m]	Projected height of tank wall
L_top	0.15[m]	Width of top

Table 5.16 Constants Edit	Window
---------------------------	--------

## Building the 2D Axisymmetric Thermos Container

The actual sequence of steps required in the building of the 2D axisymmetric Thermos\_Container\_1 model is initially somewhat complex. However, once the model is built, the modeler can use the export and import functions to use the same physical model configuration and explore the influence of different materials and materials properties on the overall design behavior, as shown in the first and second variations of the 2D axisymmetric thermos container model.

Using the menu bar, select Draw > Specify Objects > Rectangle. Create each of the two rectangles indicated in Table 5.17.

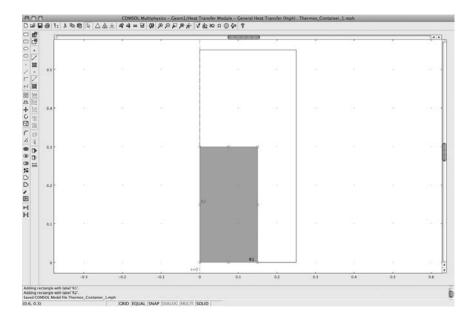
Name	Expression	Value	Description
k_foam	9e-2[W/(m*K)]	0.09[W/(m·K)]	Thermal conductivity foam
rho_foam	6e2[kg/m^3]	600[kg/m <sup>3</sup> ] Density foam	
Cp_foam	1.4e3[J/(kg*K)]	1400[J/(kg·K)]	Heat capacity foam
k_304ss	1.62e1[W/(m*K)]	16.2[W/(m·K)]	Thermal conductivity 304ss
rho_304ss	8e6[kg/m^3]	8e6[kg/m <sup>3</sup> ] Density 304ss	
Cp_304ss	5.0e2[J/(kg*K)]	500[J/(kg·K)]	Heat capacity 304ss
p_0	1.0[atm]	1.01325e5[Pa]	Air pressure
Т_О	2.7315e2[K]	273.15[K]	Boundary temperature
T_L	9.5e1[degC]	368.15[K]	Liquid temperature
L_wall	0.35[m]	0.35[m]	Projedted height of tank wall
L_top	0.15[m]	0.15[m]	Width of top

FIGURE 5.56 2D axisymmetric Thermos\_Container\_1 model Constants edit window

**Rectangle Edit Window** 

**Table 5.17** 

Object Number	Width	Height	Base	r	7
Rectangle 1	0.25	0.55	Corner	0	0
Rectangle 2	0.15	0.3	Corner	0	0



**FIGURE 5.57** 2D axisymmetric Thermos\_Container\_1 model rectangles R1 and R2

Click OK, and then click the Zoom Extents button. See Figure 5.57.

Using the menu bar, select Draw > Specify Objects > Ellipse. Create the ellipse indicated in Table 5.18. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Create the rectangle indicated in Table 5.19. Click OK.

Select Draw > Create Composite Object. Enter E1-R3 in the Set formula edit window. Verify that the Keep interior boundaries check box is unchecked. See Figure 5.58. Click OK.

Object Number	A-semiaxes	<b>B-semiaxes</b>	Base	r	z
Ellipse 1	0.15	0.05	Center	0	0.3

Table 5.18	Ellipse Edit Window
------------	---------------------

Object Number	Width	Height	Base	r	z
Rectangle	0.2	0.4	Corner	-0.2	0

Table 5.19 Rectangle Edit Window

Select Draw > Create Composite Object. Enter R2+CO1 in the Set formula edit window. Verify that the Keep interior boundaries check box is unchecked. See Figure 5.59.

Click OK. See Figure 5.60, which shows the profile of the outer tank.

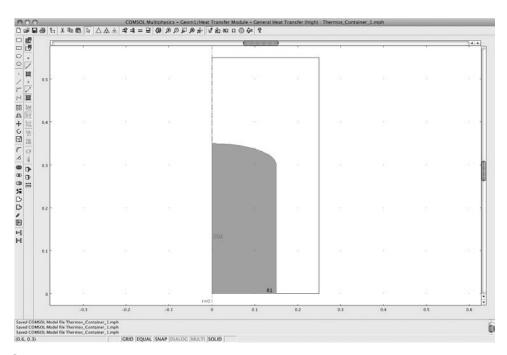
Create the inner structure of the insulated tank by following the steps in Table 5.20. Select the appropriate action from the menu bar using the Draw pull-down menu. See Figure 5.61.

	eate Composite Object
Object type Solids Curves Points	Shortcuts Union (Intersection) Select All Help
Object selection:	Set formula:
R1 R2 E1 R3	E1-R3  Keep interior boundaries  Repair Repair tolerance: 1.0E-4

FIGURE 5.58 2D axisymmetric Thermos\_Container\_1 model half-ellipse creation

<u>000</u> ci	reate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
R1 R2 CO1	R2+CO1
	Keep interior boundaries     Repair     Repair tolerance: 1.0E-4

FIGURE 5.59 2D axisymmetric Thermos\_Container\_1 model outer tank profile creation



**FIGURE 5.60** 2D axisymmetric Thermos\_Container\_1 model outer tank profile

Table 5.20	Tank Structure	<b>Creation Steps</b>
------------	----------------	-----------------------

Step	Object	Width/A	Height/B	Base	r	z
1	Rectangle	0.15-0.005	0.3–0.005	Corner	0	0.005
2	Ellipse	0.15-0.005	0.05-0.005	Center	0	0.3
3	Rectangle	0.2	0.4	Corner	-0.2	0
4	Create Compo	osite Object	Formula =	E1-R3	No Interior	Boundaries
5	Create Compo	osite Object	Formula =	R2+CO1	No Interior	Boundaries
6	Rectangle	0.15-0.025	0.3–0.025	Corner	0	0.025
7	Ellipse	0.15-0.025	0.05-0.025	Center	0	0.3
8	Rectangle	0.2	0.4	Corner	-0.2	0
9	Create Compo	osite Object	Formula =	E1-R3	No Interior	Boundaries
10	Create Compo	osite Object	Formula =	R2+CO1	No Interior	Boundaries
11	Rectangle	0.15-0.03	0.3–0.03	Corner	0	0.03
12	Ellipse	0.15-0.03	0.05-0.03	Center	0	0.3
13	Rectangle	0.2	0.4	Corner	-0.2	0
14	Create Compo	osite Object	Formula =	E1-R3	No Interior	Boundaries
15	Create Compo	osite Object	Formula =	R2+CO1	No Interior	Boundaries
16	Rectangle	0.15–0.03	0.29–0.03	Corner	0	0.03

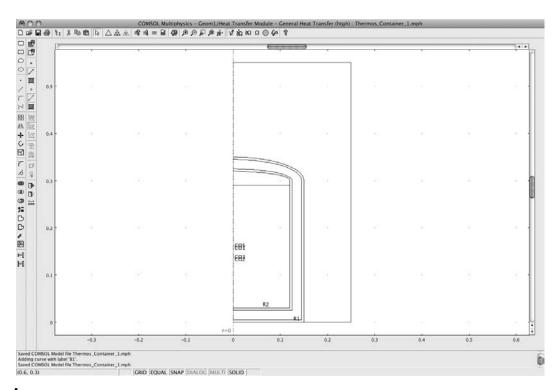


FIGURE 5.61 2D axisymmetric Thermos\_Container\_1 model tank components

The tank lid structure is defined by adding a line that separates the lid and the body of the tank. Select Draw > Specify Objects > Line. Enter 0.15-0.03 space 0.15 in the r edit window. Enter 0.3 space 0.3 in the z edit window. See Figures 5.62 and 5.63.

The next step is to combine the components into the final tank structure. Select Draw > Create Composite Object. Enter R1+CO2+CO3+CO4+CO5+R2. Important: This time, verify that the Keep interior boundaries check box is *checked*. See Figure 5.64.

00	0	Line
Coor	dinates	
r:	0.15-0.03 0.15	ОК
z:	0.3 0.3	Cancel
Style:	Polyline	\$ Apply
Name	B1	Help

FIGURE 5.62 2D axisymmetric Thermos\_Container\_1 model Line edit window

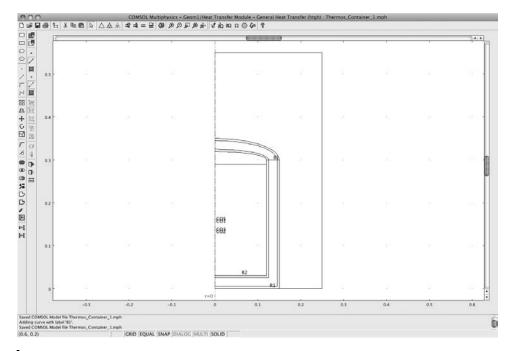


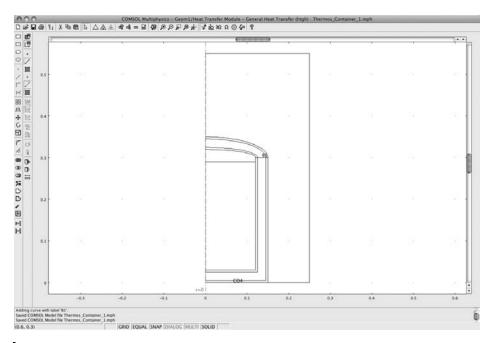
FIGURE 5.63 2D axisymmetric Thermos\_Container\_1 model tank components with lid line

Click OK. See Figure 5.65.

Now, to save time and effort on the next model, save the present insulated tank configuration. Select File > Export > Geometry Objects to File. Enter TC\_1\_Geometry in the Save As edit window. Select DXF file (\*.dxf) from the File Format pull-down list. See Figure 5.66. Click Save.

000 Ci	reate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula: R1+CO2+CO3+CO1+CO5+R2
CO2 CO3 CO1 CO5 R2	Keep interior boundaries Repair Repair 1.0E-4

**FIGURE 5.64** 2D axisymmetric Thermos\_Container\_1 model Create Composite Object edit window



**FIGURE 5.65** 2D axisymmetric Thermos\_Container\_1 model tank

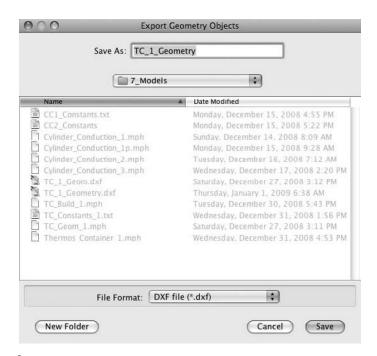


FIGURE 5.66 2D axisymmetric Thermos\_Container\_1 model tank Export Geometry Objects, Save As window

Subdomain	Operation	Name	Expression	Description
1, 3, 6, 8	Enter	k	k_304ss	Thermal conductivity
1, 3, 6, 8	Enter	ρ	rho_304ss	Density
1, 3, 6, 8	Enter	$C_{\rm P}$	Cp_304ss	Heat capacity

Table 5.21Subdomain Edit Window (1, 3, 6, 8)

Subdomain	Operation	Name	Expression	Description
2, 7	Enter	k	k_foam	Thermal conductivity
2, 7	Enter	ρ	rho_foam	Density
2, 7	Enter	$C_{P}$	Cp_foam	Heat capacity

### Table 5.23 Subdomain Edit Window (5, 9)

Subdomain	Operation	Name
5, 9	Load	Basic Materials Properties > Air, 1 atm

### Table 5.24 Subdomain Edit Window (4)

Subdomain	Operation	Name
4	Load	Basic Materials Properties > Water, liquid

# **Physics Subdomain Settings: General Heat Transfer**

Having established the geometry for the 2D axisymmetric Thermos\_Container\_1 model, the next step is to define the fundamental Physics conditions. In the Subdomain edit windows, load or enter the information shown in Tables 5.21 through 5.24. See also corresponding Figures 5.67 through 5.70.

Enter  $p_0$  in place of p in the density expression to yield  $rho(p_0[1/Pa],T[1/K])$  [kg/m^3] in the Density edit window.

uation · (-k⊽T) = Q = temperature			
Subdomains Groups	Conduction	on Convection Id	eal Gas Init Element Color
1 2 3 4 5 6 7 8 9	Library material:	\$           Value/Expression           k_304ss           400 0 0 400           rho_304ss           Cp_304ss           0           Opaque	Load           Unit         Description           W/(m·K)         Thermal conductivity           W/(m·K)         Thermal conductivity           kg/m³         Density           J/(kg·K)         Heat capacity at constant pressure           W/m³         Heat source
Group: +			

FIGURE 5.67 2D axisymmetric Thermos\_Container\_1 model Subdomain Settings (1, 3, 6, 8) edit window

quation · (−k⊽T) = Q = temperature	
Subdomains Groups Subdomain selection	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:
Select by group	

**FIGURE 5.68** 2D axisymmetric Thermos\_Container\_1 model Subdomain Settings (2, 7) edit window

0.0	Subdomain Settings – General Heat Transfer (htgh)
Equation $\nabla \cdot (-k\nabla T) = Q$ T = temperature Subdomains Groups Subdomain selection 1 2 3 4 5 6 7 8 0	Subdomain Settings - General Heat Transfer (htgh)
Group: 🛟	

I FIGURE 5.69 2D axisymmetric Thermos\_Container\_1 model Subdomain Settings (5, 9) edit window

quation '··(-k⊽T) = Q ' = temperature	
Subdomains Groups Subdomain selection	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Water, liquid       Load         Quantity       Value/Expression       Unit       Description         © k (isotropic)       kr[1/k])[W/(m*K)]       W/(m*K)       Thermal conductivity         P       rho(T[1/k])[kg/m^3]       kg/m³       Density         Cp       Cp(T[1/K])[J/(kg*K)]       J/(kg+K)       Heat capacity at constant pressure         Q       0       W/m³       Heat source         Opacity:       Opaque       Image: Constant pressure
Group: 🔹 🗘	

I FIGURE 5.70 2D axisymmetric Thermos\_Container\_1 model Subdomain Settings (4) edit window

Boundary	<b>Boundary Condition</b>	Value/Expression	Figure Number
2	Thermal insulation	_	5.71
8, 14, 21	Temperature	T_L	5.72

Table 5.25	Boundary Settings–General Heat Transfer Edit Window
------------	---

**NOTE** In this model, because of the high thermal conductivity of water, the temperature of the liquid is very close to uniform throughout. Thus uniformity can be assumed and subdomain 4 can be made inactive. The temperature of the liquid is incorporated into the boundary conditions. Also, because this model will calculate only the heat transfer, and not the detailed convection flow in the surrounding air, subdomain 9 can be made inactive. Convection losses are incorporated into the heat transfer coefficient boundary conditions. Incorporating both of these assumptions into the model significantly simplifies the model calculations.

Select subdomain 4. Uncheck the Active in this domain check box. Select subdomain 9. Uncheck the Active in this domain check box. Click OK.

### **Physics Boundary Settings: General Heat Transfer**

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.25. Click OK. See Figures 5.71 and 5.72.

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select the boundary condition.

Load the Library coefficient using the path Heat Transfer Coefficients > Air, Ext. Natural Convection as shown in Table 5.26.

Enter L\_wall in the Heat transfer coefficient (h) expression in place of the L\_htgh term for boundary 28:

 $h_ave(T[1/K],Tinf_htgh[1/K],L_wall[1/m])[W/(m^{2*}K)]$ 

Enter T\_0 in the External temperature  $(T_{inf})$  edit window for boundary 28. See Figure 5.73.

Boundary	Boundary Condition	Library Coefficient	Figure Number
28	Heat flux	Nat. Vertical wall, L=height	5.73
34	Heat flux	Nat. Horiz. plane, Upside, L=width	5.74

Equation - <b>n</b> ·(-kVT) = 0	Boundary Settings – G			
Boundaries Groups	Boundary Cond	tion Highly Conduction	ve Layer	Element Color/Style
Boundary selection	Boundary sources and Library coefficient: (	l constraints	Load)	
4	Boundary condition:	Thermal insulation	0	
6	Quantity	Value/Expression	Unit	Description
7	9 <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
9 10	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
11	T <sub>inf</sub>	273.15	к	External temperature
12	το	273.15	к	Temperature
14	Radiation type:	None	0	
16	<b>х</b> в	0	1	Surface emissivity
17	- T .	0	к	Ambient temperature
Group:	J <sub>0</sub>		W/m <sup>2</sup>	
Select by group	'0 Member of group(s):	epsilon_htgh*sigma_h	1	Surface radiosity expression

**FIGURE 5.71** 2D axisymmetric Thermos\_Container\_1 model Boundary Settings (2) edit window

Equation				
Γ = Τ <sub>0</sub>				
Boundaries Groups	Boundary Cond	ition Highly Conduction	ve Layer	Element Color/Style
Boundary selection	Boundary sources and	i constraints		
/	Library coefficient.		Load	
9	Boundary condition:	Temperature	0	
11	Quantity	Value/Expression	Unit	Description
12	9 <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
14	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
15 16	T <sub>inf</sub>	273.15	κ	External temperature
17 18	τ <sub>o</sub>	T_L	к	Temperature
19	Radiation type:	None	0	
21	a a	0	1	Surface emissivity
Group:	Tamb	0	к	Ambient temperature
Select by group	Jo	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

FIGURE 5.72 2D axisymmetric Thermos\_Container\_1 model Boundary Settings (8, 14, 21) edit window

Boundaries Groups	Boundary Cond	tion Highly Conductiv	ve Layer	Element Color/Style
Boundary selection	Boundary sources and	l constraints		
19	Library coefficient:	Nat. Vertical wall, L=h	eight	\$ Load
20	Paradan and Mara			
21	Boundary condition:	Heat flux	•	
23	Quantity	Value/Expression	Unit	Description
24	q <sub>n</sub>	0	W/m <sup>2</sup>	Inward heat flux
25	h		W/(m <sup>2</sup> ·K)	Heat transfer coefficient
26		h_ave(T[1/K],Tinf_ht		Heat transfer coefficient
28	T <sub>inf</sub>	273.15	к	External temperature
29	T <sub>O</sub>	273.15	к	Temperature
31	Radiation type:	None	\$	
32 33	ε	0	1	Surface emissivity
34	ε T		к	Contraction and the second
Group:	Tamb	0	~	Ambient temperature
Select by group	J <sub>0</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

FIGURE 5.73 2D axisymmetric Thermos\_Container\_1 model Boundary Settings (28) edit window

Enter L\_top in the Heat transfer coefficient (h) expression in place of the L\_htgh term for boundary 34:

```
h_ave(T[1/K],Tinf_htgh[1/K],L_top[1/m])[W/(m^2*K)]
```

Enter T\_0 in the External temperature  $(T_{inf})$  edit window for boundary 34. See Figure 5.74.

Click OK.

# **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters > Subdomain 1, 2, 3, 6, 7, 8. Enter Maximum element size 0.002. Select Method > Quad. Click the Remesh button. See Figure 5.75.

Click OK. See Figure 5.76.

# Solving the 2D Axisymmetric Thermos\_Container\_1 Model

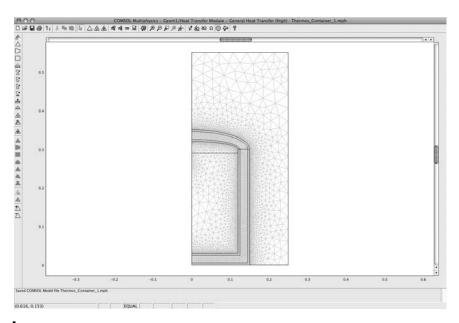
From the menu bar, select Solve > Solver Parameters > Parametric. Enter  $T_L$  in the Parameter name edit window. Enter 273.15:9.5:368.15 in the Parameter values edit window. See Figure 5.77. Click OK.

Boundaries Groups	Boundary Cond	ition Highly Conductiv	/e Layer	Element Color/Style
Boundary selection	Boundary sources and	d constraints		
19	Library coefficient:	Nat. Horiz. plane, Upsi	de, L=widtl	h 🗘 Load
20 21	Boundary condition:	Heat flux	•	
22	Quantity	Value/Expression	Unit	Description
23	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
25	- 27.			
26	h	h_ave(T[1/K],Tinf_ht	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
27	T <sub>inf</sub>	T_0	к	External temperature
29 30	т <sub>о</sub>	273.15	к	Temperature
31	Radiation type:	None	•	
33	ε	D	1	Surface emissivity
34 •	Tamb	0	к	Ambient temperature
Select by group	J <sub>O</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

**FIGURE 5.74** 2D axisymmetric Thermos\_Container\_1 model Boundary Settings (34) edit window

		_	Free Mesh Para	ameters	_	-
	Global	Subdomain	Boundary Po	int Advanced		ОК
1 2 3 4 5 6 7 8 9 9	ct by group t Remaining ect Meshed	Maxi	imain mesh parame imum element size: ient growth rate: iod:	Quad		Cancel Apply Help

I FIGURE 5.75 2D axisymmetric Thermos\_Container\_1 model Free Mesh Parameters edit window



**FIGURE 5.76** 2D axisymmetric Thermos\_Container\_1 model mesh

000	Solver Par	rameters	
Analysis:	General Pa	rametric Stationary	Adaptive Advanced
Stationary ‡	Parameter		
🗹 Auto select solver	Parameter name:	T_L	
Solver:			
Stationary	Parameter values:	273.15:	9.5:368.15
Time dependent Eigenvalue	Linear system solver		
Parametric	Linear system solver:	Direct (UMFPACK)	•
Stationary segregated	Linear system solver:	Direct (UMPPACK)	
Parametric segregated	Preconditioner:		(*)
-			Settings
Adaptive mesh refinement	Street Street		
	Matrix symmetry:	Automatic	÷
		DOWED	
		POWER	EN.IK
		(Help) (App	oly Cancel OK
		Creep CAPI	Cancel Ok

**FIGURE 5.77** 2D axisymmetric Thermos\_Container\_1 model Solver Parameters edit window

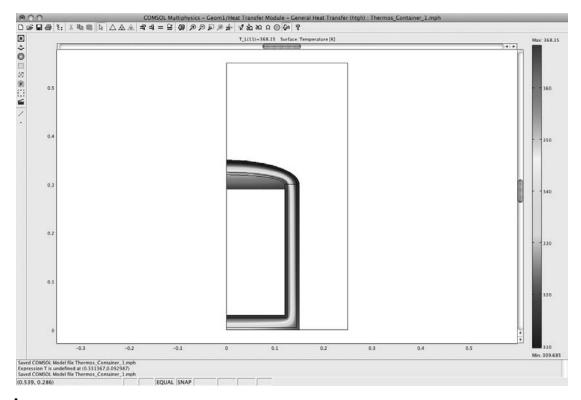


FIGURE 5.78 2D axisymmetric Thermos\_Container\_1 model using the Parametric Solver (UMFPACK)

From the menu bar, select Solve > Solve Problem. See Figure 5.78.

#### Postprocessing

Select Postprocessing > Plot Parameters > Surface. Select "°C [degC]" in the Unit pull-down list. See Figure 5.79.

Click OK. See Figure 5.80.

Given that our main interest in creating the 2D axisymmetric Thermos\_Container\_1 model is to examine the heat transfer, the next step is to determine the heat loss. Select Postprocessing > Boundary Integration. Select boundaries 28 and 34 (the wall and top of the tank, respectively). Select "Normal total heat flux" in the Predefined quantities pull-down list. Check the Compute surface integral (for axisymmetric modes) check box. See Figure 5.81.

Click OK. The result of the Boundary Integration (~82 W) is displayed as Value of surface integral: xx.xxxx [W], Expression: ntflux\_htgh, Boundaries: 28, 34 in the display window at the bottom of the COMSOL user interface. See Figure 5.82.

	Surface	_	Parameters Boundary	4	Principal
General	Surface	Contour	Boundary	Arrow	Principal
Surface	e plot				
_		Surface Data	Height Data		
Predefine	d quantities:	Temperature	R	Ran	ge
Expression	on:	т		M sm	ooth
Unit:		°c	1	a)	
Coloring a				2.	
Coloring:	Interpol	lated	Fill style:	Filled	
Surface col	lor				
Color	man:	jet 🛟	Colors: 1024	Col	or scale
Ocon					
O Unifo					
- C		Color)			
- C					
- C					
- C					
- C					
- C					
- C					
- C					
- C					
- C					

**FIGURE 5.79** 2D axisymmetric Thermos\_Container\_1 model Plot Parameters window

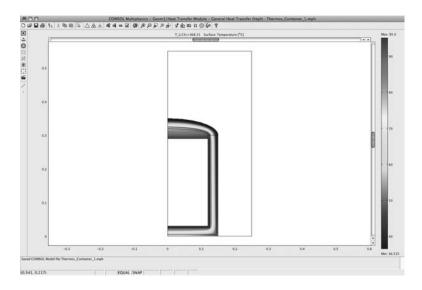
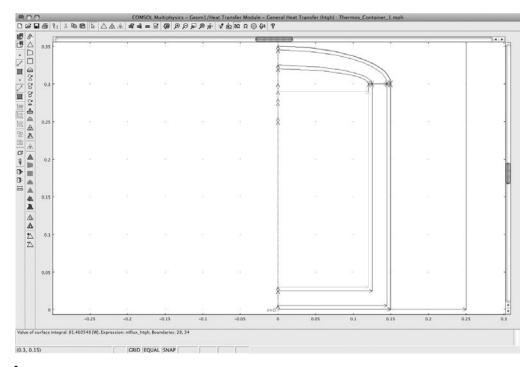


FIGURE 5.80 2D axisymmetric Thermos\_Container\_1 model surface temperature (°C)

000	Boundary Integ	ration		
Boundary selection:	Expression to integrate			
16	Predefined quantities:	Normal total he	at flux	*
17 18	Expression:	ntflux_htgh		
19	Expression.	ntilux_ntgn		
20	Unit of integral:	Ŵ		\$
21		<u></u>		
22	Compute surface i	ntegral (for axisym	metric modes	)
23				
24	Solution to use		19	
25		00015		
26	Parameter value:	368.15	*	
27	Time:			
28	T THE			
29	Solution at angle (phase	9: 0	degrees	
30		S. 19		
31	F			
	Frame:	Y		
32				
32 33	Integration order: 🗹	Auto 4	7	

FIGURE 5.81 2D axisymmetric Thermos\_Container\_1 model Boundary Integration edit window



**FIGURE 5.82** 2D axisymmetric Thermos\_Container\_1 model user interface display window

Principal St	reamline Particle Tr	acing Max/Min	Deform	Anima
Movie settings		Solutions to use		
File type:	AVI	Select via:	(	0
		1 Martine and a second		
Width (in pixels):	640	273.15 282.65		
Height (in pixels):	480	292.15		
Frames per second:	10	301.65 311.15		
	(Advanced)	320.65		
	Advanced	330.15		U
Static / Eigenfunctio	on animation	339.65 349.15		¥.
and a second second	Full harmonic	358.65		
Cycle type:				
Number of frames.	11	Times:		
Number of frames.	11 n	Times:		

FIGURE 5.83 2D axisymmetric Thermos\_Container\_1 model Plot Parameters window

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.83.

Click the Start Animation button. See Figure 5.84.

Alternatively, you can play the file Movie5\_TC\_1.avi that was supplied with this book.

# First Variation on the 2D Axisymmetric Thermos\_Container Model

Note In this model, Thermos\_Container\_2, the walls of the flask (e.g., bottle, tank) are formed of stainless steel. In the 2D axisymmetric Thermos\_Container\_1 model, the selected thermal insulating solid was rigid urethane foam. In this model, the first variation on the 2D axisymmetric Thermos\_Container model, a vacuum cavity replaces the urethane foam.

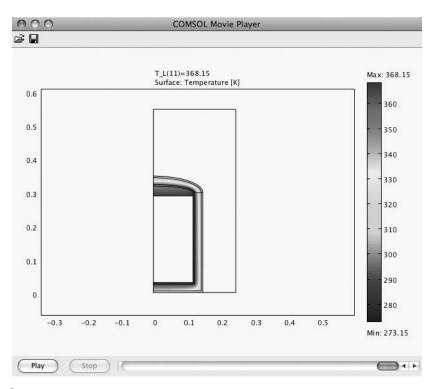


FIGURE 5.84 2D axisymmetric Thermos\_Container\_1 model animation, final frame

To start building the Thermos\_Container\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.85. Click OK.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.27; also see Figure 5.86. Click OK.

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

00	Model Naviga	ator
New	Model Library User Mod	els Open Settings
Application Modes COMSOL Multiphy AC/DC Module Heat Transfer Mo General Heat General Heat Transient Bioheat Equat	dule Fransfer ate analysis analysis	Description: Heat transfer by conduction,
<ul> <li>h -ε Turbulenc</li> <li>h -ω Turbulenc</li> <li>h -ω Turbulenc</li> <li>Electro-Therma</li> <li>Fluid-Therma</li> </ul>	ce Model nal Interaction	convection, and radiation. Steady-state analysis in 2D axial symmetry.
Application mode name: Element:	htgh Lagrange – T <sub>2</sub> J <sub>1</sub>	Multiphysics
Element:	Lagrange - 1 <sub>2</sub> J <sub>1</sub>	Multiphysics

FIGURE 5.85 2D axisymmetric Thermos\_Container\_2 Model Navigator setup

When building a model, it is usually best to choose a name for modelerdefined parameters (e.g., constants, scalar functions) that are easily recalled and associated with the function that they provide in the model (e.g., p\_vac, L\_wall).

Name	Expression	Value	Description
k_304ss	1.62e1[W/(m*K)]	16.2[W/(m·K)]	Thermal conductivity 304ss
rho_304ss	8e6[kg/m^3]	8e6[kg/m <sup>3</sup> ]	Density 304ss
Cp_304ss	5.0e2[J/(kg*K)]	500[J/(kg·K)]	Heat capacity 304ss
p_0	1.0[atm]	1.01325e5[Pa]	Air pressure
p_vac	1.33e-7[Pa]	(1.33e-7)[Pa]	Pressure in Vacuum
T_0	2.7315e2[K]	273.15[K]	Boundary temperature
T_L	9.5e1[degC]	368.15[K]	Liquid temperature
L_wall	0.35[m]	0.35[m]	Projected height of tank wall
L_top	0.15[m]	0.15[m]	Width of top
L_top	0.15[m]	0.15[m]	

FIGURE 5.86 2D axisymmetric Thermos\_Container\_2 model Constants edit window

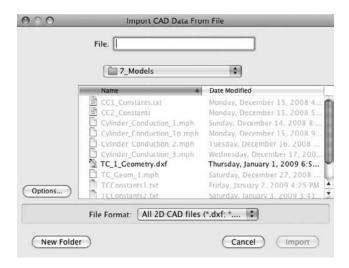
Name	Expression	Description
k_304ss	1.62e1[W/(m*K)]	Thermal conductivity 304ss
rho_304ss	8e6[kg/m^3]	Density 304ss
Cp_304ss	5.0e2[J/(kg*K)]	Heat capacity 304ss
p_0	1.0[atm]	Air pressure
p_vac	1.33e-7[Pa]	Pressure in vacuum
T_0	2.7315e2[K]	Boundary temperature
T_L	9.5e1[degC]	Liquid temperature
L_wall	0.35[m]	Projected height of tank wall
L_top	0.15[m]	Width of top

Select File > Save as. Enter Thermos\_Container\_2. Click the Save button.

#### **Importing the 2D Axisymmetric Thermos Container**

**NOTE** The actual sequence of steps required in the building of the 2D axisymmetric Thermos\_Container was presented in the discussion of the 2D axisymmetric Thermos\_Container\_1 model. Now the modeler can use the import function to utilize the same physical model configuration and explore the influence of different materials and materials properties on the overall model design behavior.

Using the menu bar, select File > Import > CAD Data From File. Select "TC\_1\_Geometry.dxf." See Figure 5.87. Click the Import button.



**FIGURE 5.87** 2D axisymmetric Thermos\_Container\_2 model import

Object Number	Width	Height	Base	r	z
Rectangle 1	0.25	0.55	Corner	0	0

NOTE Because the Geometry.dxf file contains only boundary information, the modeler needs to create a domain to which the boundary information can be applied.

Using the menu bar, select Draw > Specify Objects > Rectangle, as indicated in Table 5.28. Click OK. See Figure 5.88.

### **Physics Subdomain Settings: General Heat Transfer**

**Rectangle Edit Window** 

Table 5.28

Having established the geometry for the 2D axisymmetric Thermos\_Container\_2 model, the next step is to define the fundamental Physics conditions. Select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 5.29. See Figure 5.89.

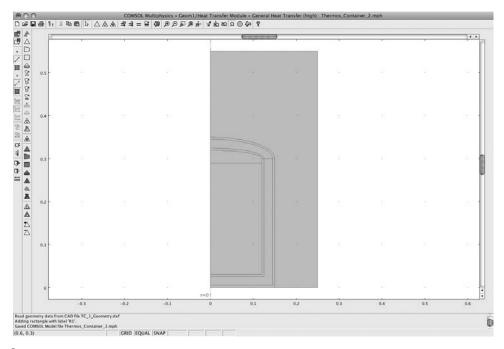


FIGURE 5.88 2D axisymmetric Thermos\_Container\_2 model import and rectangle R1

Subdomain	Operation	Name	Expression	Description
1, 3, 6, 8	Enter	k	k_304ss	Thermal conductivity
1, 3, 6, 8	Enter	ρ	rho_304ss	Density
1, 3, 6, 8	Enter	$C_{P}$	Cp_304ss	Heat capacity

In the Subdomain edit windows, enter the information shown in Table 5.30. Enter p\_vac in place of p in the density expression  $rho(p[1/Pa],T[1/K])[kg/m^3]$ , so that it reads  $rho(p_vac[1/Pa],T[1/K])[kg/m^3]$  in the Density edit window. Select "Transparent" from the Opacity pull-down list. See Figure 5.90.

In the Subdomain edit windows, enter the information shown in Table 5.31. Enter  $p_0$  in place of p in the density expression to yield  $rho(p_0[1/Pa],T[1/K])[kg/m^3]$  in the Density edit window. Select "Transparent" from the Opacity pull-down list. See Figure 5.91.

In the Subdomain edit windows, enter the information shown in Table 5.32. See Figure 5.92.

quation ?·(-k∇T) = Q = temperature			
Subdomains Groups Subdomain selection	Conduction Thermal properties and	nd heat sources/sinks	eal Gas Init Element Color
2 3 4 5 6 7 7 8 9	Library material: Quantity k (isotropic) k (anisotropic) p C <sub>p</sub> Q Opacity:	Value/Expression           k_304ss           400 0 0 400           rho_304ss           Cp_304ss           0           Opaque	Unit         Description           W/(m·K)         Thermal conductivity           W/(m·K)         Thermal conductivity           W/(m·K)         Thermal conductivity           kg/m³         Density           J/(kg·K)         Heat capacity at constant pressure           W/m³         Heat source
Group: *			

FIGURE 5.89 2D axisymmetric Thermos\_Container\_2 model Subdomain Settings (1, 3, 6, 8) edit window

Subdomain	Operation	Name
2, 7	Load	Basic Materials Properties > Air, 1 atm

# Table 5.30 Subdomain Edit Window

# Table 5.31 Subdomain Edit Window

Subdomain	Operation	Name
5, 9	Load	Basic Materials Properties > Air, 1 atm

# Table 5.32 Subdomain Edit Window

Subdomain	Operation	Name
4	Load	Basic Materials Properties > Water, liquid

quation · (-k⊽T) = Q = temperature		
Subdomains Groups Subdomain selection 1 2 3 4 5 6 7 8 9	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Air, 1 atm       Load         Quantity       Value/Expression       Unit       Description         Image: Sinks       K(T[1/K])[W/(m*K)]       W/(m·K)       Thermal conductivity         Image: Sinks       K(T[1/K])[U/(kg*K)]       V/(m·K)       Thermal conductivity         Image: Sinks       K(T[1/K])[U/(kg*K)]       V/(m·K)       Thermal conductivity         Image: Sinks       K(T[1/K])[U/(kg*K)]       J/(kg·K)       Heat capacity at constant presson         Image: Sinks       Image: Sinks       V/m³       Heat source         Image: Sinks       Image: Sinks       Image: Sinks       Image: Sinks         Image: Sinks       Image: Sinks       Image: Sinks       Image: Sinks         Image: Sinks       Image: Sinks       Image: Sinks       Image: Sinks       Image: Sinks         Image: Sinks <th>ssur</th>	ssur
Group:		

**FIGURE 5.90** 2D axisymmetric Thermos\_Container\_2 model Subdomain Settings (2, 7) edit window

·(-k⊽T) = Q = temperature	
Subdomains Groups Subdomain selection	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Air, 1 atm_1       Load         Quantity       Value/Expression       Unit       Description         Image: Sources (Sinks)       K(T[1/K])[W/(m*K)]       W/(m*K)       Thermal conductivity         Image: Sources (Sinks)       K(T[1/K])[W/(m*K)]       W/(m*K)       Thermal conductivity         Image: Source (Sinks)       K(Inisotropic)       400 0 0 400       W/(m*K)       Thermal conductivity         Image: Source (Sinks)       For (Sinks)       Kg/m³       Density       Cp.       Cp(T[1/K])[J/(Kg*K)]         Image: Source (Sinks)       Image: Source (Sinks)       W/m³       Heat source (Sinks)       Source (Sinks)

**FIGURE 5.91** 2D axisymmetric Thermos\_Container\_2 model Subdomain Settings (5, 9) edit window

·(-k∇T) = Q = temperature	
Subdomains Groups Subdomain selection	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Water, liquid ()       Load         Quantity       Value/Expression       Unit       Description         ()       k (isotropic)       k(T[1]/K])[W/(m*K)]       W/(m*K)       Thermal conductivity         ()       k (anisotropic)       400 0 0 400       W/(m*K)       Thermal conductivity         ()       rho(T[1]/K])[Kg/m^3]       kg/m <sup>3</sup> Density         ()       Cp       Cp(T[1]/K])[J]/(kg*K)]       J/(kg*K)         ()       0       W/m <sup>3</sup> Heat capacity at constant press         ()       0       W/m <sup>3</sup> Heat source         ()       Opacity:       Opaque       ()
Group: 🔹 🛟	

I FIGURE 5.92 2D axisymmetric Thermos\_Container\_2 model Subdomain Settings (4) edit window

Boundary	Boundary Condition	Value/Expression	Figure Number
2	Thermal insulation	_	5.93
8, 14, 21	Temperature	T_L	5.94

NOTE In this model, because of the high thermal conductivity of water, the temperature of the liquid is very close to uniform throughout. Thus uniformity can be assumed and subdomain 4 can be made inactive. The temperature of the liquid is incorporated into the boundary conditions. Also, because this model will calculate only the heat transfer, and not the detailed convection flow in the surrounding air, subdomain 9 can be made inactive. Convection losses are incorporated into the heat transfer coefficient boundary conditions. Incorporating both of these assumptions into the model significantly simplifies the model calculations.

Select subdomain 4. Uncheck the Active in this domain check box. Select subdomain 9. Uncheck the Active in this domain check box. Click OK.

### **Physics Boundary Settings: General Heat Transfer**

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.33. Click OK. See Figures 5.93 and 5.94.

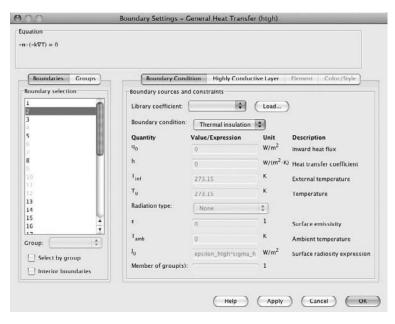


FIGURE 5.93 2D axisymmetric Thermos\_Container\_2 model Boundary Settings (2) edit window

Boundary Condi	tion Highly Conductiv	ve Layer	Element Color/Style
-Boundary sources and	l constraints		
Library coefficient: (	÷ (	Load	
Boundary condition:	Temperature	•	
Quantity		Unit	Description
q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
т.		- 121 - 18	
	273.15		External temperature
To	T_L	к	Temperature
Radiation type:	None	\$	
ε	0	1	Surface emissivity
т.,		к	
			Ambient temperature
	epsilon_htgh*sigma_h	W/m²	Surface radiosity expression
Member of group(s):		1	
	Boundary Condi Boundary sources and Library coefficient: Boundary condition: Quantity Q0 h T <sub>inf</sub> T <sub>0</sub> Radiation type:	Boundary Condition       Highly Conduction         Boundary sources and constraints       Library coefficient:         Library condition:       Temperature         Quantity       Value/Expression         Q0       0         h       0         Tinf       273.15         T0       T_L         Radiation type:       None         ε       0         Tamb       0         J0       epsilon_htgh*sigma_h	Boundary sources and constraints         Library coefficient:       Load         Boundary condition:       Temperature       Load         Boundary condition:       Temperature       Load         Boundary condition:       Temperature       Unit         Quantity       Value/Expression       Unit         q_0       0       W/m²       W/m²         h       0       W/m²       K         To       T_L       K       K         Radiation type:       None       I       I         z       0       K       J_0       epsilon_htgh*sigma_h       W/m²

FIGURE 5.94 2D axisymmetric Thermos\_Container\_2 model Boundary Settings (8, 14, 21) edit window

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select the boundary condition.

Load the Library coefficient using the path Heat Transfer Coefficients > Air, Ext. Natural Convection as shown in Table 5.34.

Enter L\_wall in the Heat transfer coefficient (h) expression in place of the L\_htgh term for boundary 28:

 $h_ave(T[1/K],Tinf_htgh[1/K],L_wall[1/m])[W/(m^{2*K})]$ 

# Table 5.34 Boundary Settings–General Heat Transfer Edit Window

Boundary	Boundary Condition	Library Coefficient	Figure Number
28	Heat flux	Nat. Vertical wall, L=height	5.95
34	Heat flux	Nat. Horiz. plane, Upside, L=width	5.96

quation n · (-k⊽T) = q <sub>0</sub> + h(T <sub>inf</sub> - T)				
1 ( - K + T) = 40 + 11( inf - 1)				
Boundaries Groups	Boundary Cond	ition Highly Conductiv	e Layer	Element Color/Style
Boundary selection	Boundary sources and	d constraints		
19	Library coefficient:	Nat. Vertical wall, L=h	eight	Load
20	Boundary condition:	Heat flux	ด	
22	Quantity	Value/Expression	Unit	Description
23	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
25	1.00	-		
26	h	h_ave(T[1/K],Tinf_htg	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
27	Tinf	T_0	к	External temperature
79 30	τ <sub>0</sub>	273.15	к	Temperature
31 32	Radiation type:	None	•	
33	£	0	1	Surface emissivity
34 The second se	T <sub>amb</sub>	0	к	Ambient temperature
Select by group	Jo	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

**FIGURE 5.95** 2D axisymmetric Thermos\_Container\_2 model Boundary Settings (28) edit window

Boundaries Groups	Boundary Cond	ition Highly Conductiv	ve Layer	Element Color/Style
Boundary selection	Boundary sources and	d constraints		
19	Library coefficient.	Nat. Horiz. plane, Upsi	de, L-widt	h 🗘 Load
20 21	Boundary condition:	Heat flux	6	
22	Quantity	Value/Expression	Unit	Description
24	9 <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
25	h	-		
26	n	h_ave(T[1/K],Tinf_hts	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
27	T <sub>inf</sub>	T_0	к	External temperature
29	To	273.15	к	Temperature
31	Radiation type:	None	•	
33	£	0	1	Surface emissivity
34	Tamb	0	к	Ambient temperature
Group:	Jo			
Select by group		epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

**FIGURE 5.96** 2D axisymmetric Thermos\_Container\_2 model Boundary Settings (34) edit window

000			Free Mesh	Paramet	ers	
	Global	Subdomain	Boundary	Point	Advanced	ОК
Select	by group Remaining t Meshed	Maxin		size: 0 e:	Quad	Cancel Apply Help

**FIGURE 5.97** 2D axisymmetric Thermos\_Container\_2 model Free Mesh Parameters edit window

Enter T\_0 in the External temperature  $(T_{inf})$  edit window for boundary 28. See Figure 5.95.

Enter L\_top in the Heat transfer coefficient (h) expression in place of the L\_htgh term for boundary 34:

 $h_ave(T[1/K],Tinf_htgh[1/K],L_top[1/m])[W/(m^2*K)]$ 

Enter T\_0 in the External temperature  $(T_{inf})$  edit window for boundary 34. See Figure 5.96.

Click OK.

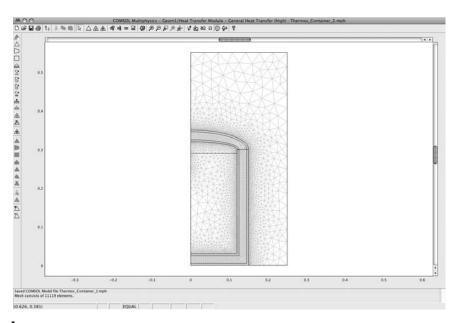
#### **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters > Subdomain 1, 2, 3, 6, 7, 8. Enter Maximum element size 0.002. Select Method > Quad. Click the Remesh button. See Figure 5.97.

Click OK. See Figure 5.98.

#### Solving the 2D Axisymmetric Thermos\_Container\_2 Model

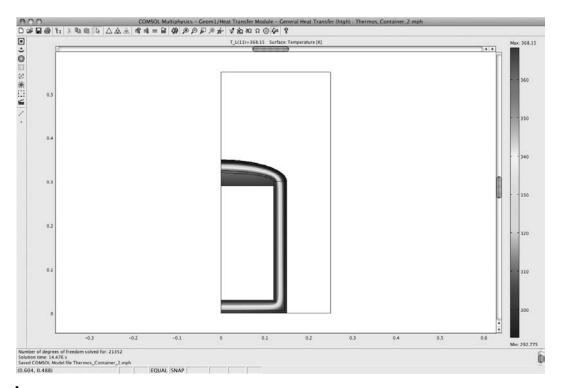
From the menu bar, select Solve > Solver Parameters > Parametric. Enter  $T_L$  in the Parameter name edit window. Enter 273.15:9.5:368.15 in the Parameter values edit window. See Figure 5.99. Click OK.



**FIGURE 5.98** 2D axisymmetric Thermos\_Container\_2 model mesh

900	Solver Pa	rameters		_	
Analysis:	General Pa	trametric	Stationary	Adaptive	Advanced
Stationary 🛟	Parameter				
Auto select solver	Parameter name:		T_L		
iolver:					
Stationary Time dependent	Parameter values:		273.15:9	.5:368.15	
Eigenvalue	Linear system solver				
Parametric	Linear system solver:	Direct (I	UMFPACK)		12
Stationary segregated Parametric segregated					
	Preconditioner:			\$	
	Matrix symmetry:	Automa			
		Help		ly) (c	ancel OK

FIGURE 5.99 2D axisymmetric Thermos\_Container\_2 model Solver Parameters edit window



**FIGURE 5.100** 2D axisymmetric Thermos\_Container\_2 model using the Parametric Solver (UMFPACK)

From the menu bar, select Solve > Solve Problem. See Figure 5.100.

#### Postprocessing

Select Postprocessing > Plot Parameters > Surface. Select "°C [degC]" in the Unit pull-down list. See Figure 5.101.

Click OK. See Figure 5.102.

Given that our main interest in creating the 2D axisymmetric Thermos\_Container\_2 model is to examine the heat transfer, the next step is to determine the heat loss. Select Postprocessing > Boundary Integration. Select boundaries 28 and 34 (the wall and top of the tank, respectively). Select "Normal total heat flux" in the Predefined quantities pull-down list. Check the Compute surface integral (for axisymmetric modes) check box. See Figure 5.103.

Click OK. The result of the Boundary Integration (~37 W) is displayed as Value of surface integral: xx.xxxx [W], Expression: ntflux\_htgh, Boundaries: 28, 34 in the display window at the bottom of the COMSOL user interface. The amount of energy lost is approximately 45% of that lost using the urethane foam insulation (~82 W) in the 2D axisymmetric Thermos\_Container\_1 model. See Figure 5.104.

General	Surface	Contour	Boundary	Arrow	Principal	
Surface	plot					
		Surface Data	Height Data			
	37					
Predefined	d quantities:	Temperature		Rang	ge	
Expressio	n:	т		Sm Sm	ooth	
Unit:		°C		1		
Coloring an						
Coloring:	Interpo	lated	Fill style:	Filled	\$	
1999 1999 1999 1997	· · · · · · · · · · · · · · · · · · ·					
Surface colo	ır					
		jet 🗘	Colors: 1024	<b>1</b> Col	or scale	
Surface colo	nap:	jet 🔹	Colors: 1024	🗹 Col	or scale	
Surface colo	nap:		Colors: 1024	Col	or scale	
Surface colo	nap:		Colors: 1024	<b>I</b> Col	or scale	
Surface colo	nap:		Colors: 1024	Col	or scale	
Surface colo	nap:		Colors: 1024	Col	or scale	
Surface colo	nap:		Colors: 1024	Col	or scale	
Surface colo	nap:		Colors: 1024	Coi	or scale	
Surface colo	nap:		Colors: 1024	<b>S</b> col	or scale	
Surface colo	nap:		Colors: 1024	ि <b>लि</b> со	or scale	
Surface colo	nap:		Colors: 1024	ि <b>लि</b> Col	or scale	

**FIGURE 5.101** 2D axisymmetric Thermos\_Container\_2 model Plot Parameters window

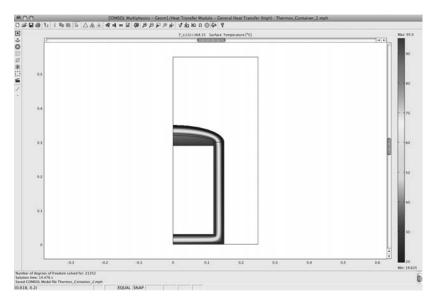
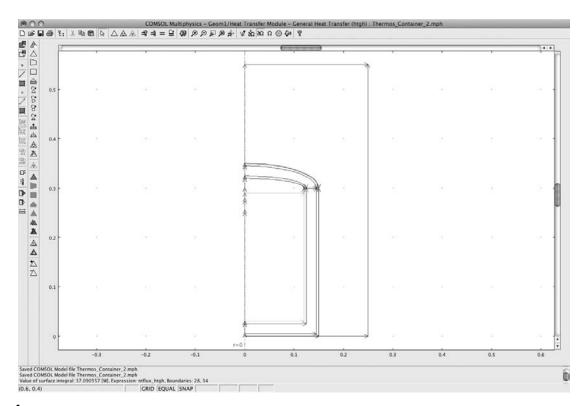


FIGURE 5.102 2D axisymmetric Thermos\_Container\_2 model surface temperature (°C)

Boundary Integ	ration	
Expression to integrate		
Predefined quantities:	Normal total heat f	ux 🗘
Expression:	ntflux htah	
	<u></u>	*
Compute surface i	ntegral (for axisymme	tric modes)
Solution to use		
Parameter value:	368.15	\$
Time:		
Solution at angle (phase	e): 0	degrees
Frame:	¥	
Integration order: 🗹 A	Auto 4	
	Expression to integrate Predefined quantities: Expression: Unit of integral: Compute surface i Solution to use Parameter value: Time: Solution at angle (phase Frame:	Expression: ntflux_htgh Unit of integral: W Compute surface integral (for axisymme Solution to use Parameter value: 368.15 Time: Solution at angle (phase): 0 Frame: ;

**FIGURE 5.103** 2D axisymmetric Thermos\_Container\_2 model Boundary Integration edit window



**FIGURE 5.104** 2D axisymmetric Thermos\_Container\_2 model user interface display window

Principal Str	eamline Particle Tra	icing Max/Min	Deform Anima
Movie settings		Solutions to use	
File type.	AVI \$	Select via:	<b>\$</b>
		land the second s	
Width (in pixels):	640	273.15 282.65	1
Height (in pixels):	480	292.15	
Frames per second:	10	301.65	1
riance per second.		311.15 320.65	
	Advanced	320.65	U
Static / Eigenfunctio	n animation	339.65	
		349.15 358.65	<b></b>
Cycle type:	Full harmonic	1358,65	
Reverse direction	11	Times:	
Reverse direction	1	Times:	
Reverse direction	1	Times:	
Reverse direction	1	Times:	
Reverse direction	1	Times:	
Reverse direction	1	Times:	Start Animation

FIGURE 5.105 2D axisymmetric Thermos\_Container\_2 model Plot Parameters window

# **Postprocessing Animation**

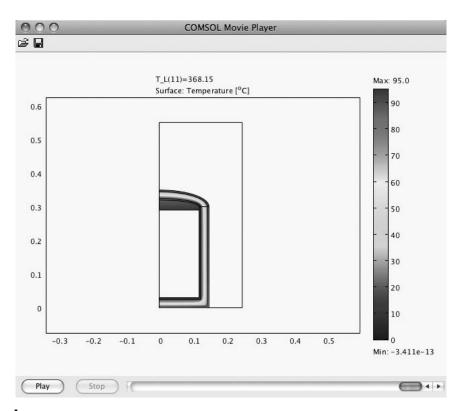
Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.105.

Click the Start Animation button. See Figure 5.106.

Alternatively, you can play the file Movie5\_TC\_2.avi that was supplied with this book.

# Second Variation on the 2D Axisymmetric Thermos\_Container Model

NOTE In this model, the second variation on the 2D axisymmetric Thermos\_ Container model, a glass material replaces the stainless steel walls and a vacuum cavity replaces the urethane foam.



**FIGURE 5.106** 2D axisymmetric Thermos\_Container\_2 model animation, final frame

To start building the Thermos\_Container\_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > General Heat Transfer > Steady-state analysis. See Figure 5.107. Click OK.

#### Constants

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 5.35; also see Figure 5.108. Click OK.

New	Model Naviga Model Library User Mode	
<ul> <li>☐ Transient</li> <li>Bioheat Equat</li> <li>☐ Weakly Comp</li> <li>☐ k-∈ Turbulent</li> <li>☐ k-ω Turbulert</li> </ul>	dule Transfer ate analysis ion ressible Navier–Stokes ce Model mal Interaction	Description: Heat transfer by conduction, convection, and radiation. Steady-state analysis in 2D axial symmetry.
Dependent variables: Application mode name.	T J htgh { Lagrange - T <sub>2</sub> J <sub>1</sub>   \$	Multiphysics

I FIGURE 5.107 2D axisymmetric Thermos\_Container\_3 Model Navigator setup

Table 5.35 Constants Edit Window

Name	Expression	Description
p_0	1.0[atm]	Air pressure
p_vac	1.33e-7[Pa]	Pressure in vacuum
T_0	2.7315e2[K]	Boundary temperature
T_L	9.5e1[degC]	Liquid temperature
L_wall	0.35[m]	Projected height of tank wall
L_top	0.15[m]	Width of top

000	2	Consta	Constants		
Name	Expression	Value	Description		
p_0	1.0[atm]	1.01325e5[Pa]	Air pressure		
p_vac	1.33e-7[Pa]	(1.33e-7)[Pa]	Pressure in Vacuum		
0_1	2./315e2[K]	2/3.15[K]	Boundary temperature		
r_L	9.5e1[degC]	368.15[K]	Liquid temperature		
wall	0.35[m]	0.35[m]	Projected height of tank wall		
L_top	0.15[m]	0.15[m]	Width of top		
c		Help	Apply Cancel OK		

**FIGURE 5.108** 2D axisymmetric Thermos\_Container\_3 model Constants edit window

	7_Models	\$
	Name 🔺	Date Modified
Options	<ul> <li>CC1_Constants.txt</li> <li>CC2_Constants</li> <li>Cylinder_Conduction_1.mph</li> <li>Cylinder_Conduction_2.mph</li> <li>Cylinder_Conduction_3.mph</li> <li>Cylinder_Conduction_3.mph</li> <li>TC_1_Geometry.dxf</li> <li>TC_Geom_1.mph</li> <li>TCConstants1.txt</li> <li>TCCConstants2.txt</li> </ul>	Monday, December 15, 2008 4 Monday, December 15, 2008 5 Sunday, December 14, 2008 8 Monday, December 15, 2008 9 Tuesday, December 16, 2008 Wednesday, December 16, 2008 <b>Thursday, January 1, 2009 6:5</b> Saturday, December 27, 2008 Friday, January 2, 2009 4:25 PM Saturday, January 3, 2009 3:41

**FIGURE 5.109** 2D axisymmetric Thermos\_Container\_2 model import

When building a model, it is usually best to choose a name for modeler-defined parameters (e.g., constants, scalar functions) that are easily recalled and associated with the function that they provide in the model (e.g., p\_vac, L\_wall).

Select File > Save as. Enter Thermos\_Container\_3. Click the Save button.

#### **Importing the 2D Axisymmetric Thermos Container**

**NOTE** The actual sequence of steps required in the building of the 2D axisymmetric Thermos\_Container was presented in the discussion of the 2D axisymmetric Thermos\_Container\_1 model. Now the modeler can use the import function to utilize the same physical model configuration and explore the influence of different materials and materials properties on the overall model design behavior.

Using the menu bar, select File > Import > CAD Data From File. Select " $TC_1$ \_Geometry.dxf." See Figure 5.109. Click the Import button.

**NOTE** Because the Geometry.dxf file contains only boundary information, the modeler needs to create a domain to which the boundary information can be applied.

Using the menu bar, select Draw > Specify Objects > Rectangle, as indicated in Table 5.36.

Click OK. See Figure 5.110.

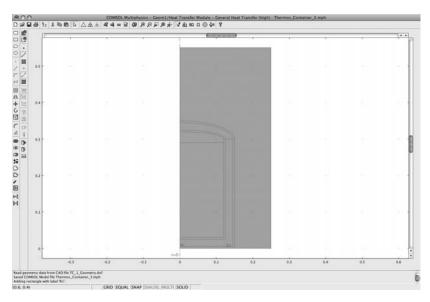


FIGURE 5.110 2D axisymmetric Thermos\_Container\_3 model import and rectangle R1

# **Physics Subdomain Settings: General Heat Transfer**

Having established the geometry for the 2D axisymmetric Thermos\_Container\_3 model, the next step is to define the fundamental Physics conditions. Select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 5.37. See Figure 5.111.

In the Subdomain edit windows, enter the information shown in Table 5.38.

# Table 5.36 Rectangle Edit Window

Object Number	Width	Height	Base	r	z
Rectangle 1	0.25	0.55	Corner	0	0

# Table 5.37 Subdomain Edit Window

Subdomain	Operation	Name
1, 3, 6, 8	Load	Basic Materials Properties > Silica Glass

# Table 5.38 Subdomain Edit Window

Subdomain	Operation	Name
2, 7	Load	Basic Materials Properties > Air, 1 atm

000	Subdomain Settings – General Heat Transfer (htgh)
Equation $\nabla \cdot (-k\nabla T) = Q$ T = temperature	
Subdomains Groups Subdomain selection	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Silica Glass       Load         Quantity       Value/Expression       Unit       Description         Image: Silica Glass       Unit       Unit       Unit         Image: Silica Glass
Group: 🔹 🗘	Opacity: Opaque

FIGURE 5.111 2D axisymmetric Thermos\_Container\_3 model Subdomain Settings (1, 3, 6, 8) edit window

Enter p\_vac in place of p in the density expression rho(p[1/Pa],T[1/K])[kg/m^3], so that it reads rho(p\_vac[1/Pa],T[1/K])[kg/m^3] in the Density edit window. Select "Transparent" from the Opacity pull-down list. See Figure 5.112.

In the Subdomain edit windows, enter the information shown in Table 5.39.

Enter  $p_0$  in place of p in the density expression to yield rho( $p_0[1/Pa]$ , T[1/K])[kg/m^3] in the Density edit window. Select "Transparent" from the Opacity pull-down list. See Figure 5.113.

In the Subdomain edit windows, enter the information shown in Table 5.40. See Figure 5.114.

Subdomain	Operation	Name				
5, 9	Load	Basic Materials Properties > Air, 1 atm				

#### Table 5.39 Subdomain Edit Window

uuation · (-k⊽T) = Q = temperature	
Subdomains Groups Subdomain selection	Conduction       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Air, 1 atm       Load         Quantity       Value/Expression       Unit       Description         Image: Rel conductive in the intervence of the inter
Group: 🔹 🛟	

FIGURE 5.112 2D axisymmetric Thermos\_Container\_3 model Subdomain Settings (2, 7) edit window

·(-k⊽T) = Q = temperature	
Subdomains Groups Subdomain selection 1 2 3 4 5 6 7 8 0 Group:	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Air, 1 atm_1       Image: Color       Load         Quantity       Value/Expression       Unit       Description         Image: k (isotropic)       k(T[1/K])[W/(m*K)]       W/(m:K)       Thermal conductivity         k (anisotropic)       400 0 0 400       W/(m:K)       Thermal conductivity         p       rho(p.0[1/Pa],T[1/K])       kg/m³       Density         Cp       Cp(T[1/K])[J/(kg*K)]       J/(kg+K)       Heat capacity at constant press         Q       0       W/m³       Heat source         Opacity:       Transparent       Image: Color       Image: Color
Select by group	

FIGURE 5.113 2D axisymmetric Thermos\_Container\_2 model Subdomain Settings (5, 9) edit window

Cult damain	Onenation	News
Subdomain	Operation	Name
4	Load	Basic Materials Properties > Water, liquid

NOTE In this model, because of the high thermal conductivity of water, the temperature of the liquid is very close to uniform throughout. Thus uniformity can be assumed and subdomain 4 can be made inactive. The temperature of the liquid is incorporated into the boundary conditions. Also, because this model will calculate only the heat transfer, and not the detailed convection flow in the surrounding air, subdomain 9 can be made inactive. Convection losses are incorporated into the heat transfer coefficient boundary conditions. Incorporating both of these assumptions into the model significantly simplifies the model calculations.

Select Subdomain 4. Uncheck the Active in this domain check box. Select Subdomain 9. Uncheck the Active in this domain check box. Click OK.

0.0	Subdomain Settings – General Heat Transfer (htgh)
quation · (−k⊽T) = Q = temperature	
Subdomains Groups Subdomain selection	Conduction       Convection       Ideal Gas       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Water, liquid       Load         Quantity       Value/Expression       Unit       Description         Image: box is isotropic       k(fsotropic)       k(T[1/K)][W/(m*K)]       Thermal conductivity         Image: box isotropic       400 0 0 400       W/(m*K)       Thermal conductivity         Image: box isotropic       400 0 0 400       W/(m*K)       Thermal conductivity         Image: box isotropic       400 0 0 400       W/(m*K)       Thermal conductivity         Image: box isotropic       400 0 0 400       W/(m*K)       Thermal conductivity         Image: box isotropic       400 0 0 400       W/(m*K)       Thermal conductivity         Image: box isotropic       400 0 0 400       W/(m*K)       Thermal conductivity         Image: box isotropic       400 0 0 400       W/(m*K)       Thermal conductivity         Image: box isotropic       400 0 0 400       W/(m*K)       Heat capacity at constant pressure         Image: box isotropic       Image: box isotropic       J/(kg·K)       Heat source         Image: box isotropic       Image: box isotropic       Image: box isotropic       Image
Select by group	

FIGURE 5.114 2D axisymmetric Thermos\_Container\_3 model Subdomain Settings (4) edit window

Boundary	Boundary Condition	Value/Expression	Figure Number
2	Thermal insulation	_	5.115
8, 14, 21	Temperature	T_L	5.116

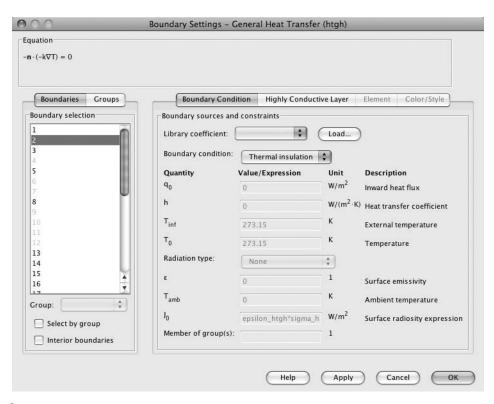
Table 5.41 Boundary Settings–General Heat Transfer Edit Window

# **Physics Boundary Settings: General Heat Transfer**

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 5.41. Click OK. See Figures 5.115 and 5.116.

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select the Boundary condition.

Load the Library coefficient using the path Heat Transfer Coefficients > Air, Ext. Natural Convection as shown in Table 5.42.



**FIGURE 5.115** 2D axisymmetric Thermos\_Container\_3 model Boundary Settings (2) edit window

= T <sub>0</sub>				
Boundaries Groups	Boundary Cond	ition Highly Conducti	e Layer	Element Color/Style
Boundary selection	Boundary sources and	d constraints		
13	Library coefficient:		Load	
15 16	Boundary condition:	Temperature		
17	Quantity	Value/Expression	Unit	Description
18	a <sup>0</sup>	0	W/m <sup>2</sup>	Inward heat flux
20	h	0	W/(m <sup>2</sup> K)	Heat transfer coefficient
22	Tinf	273.15	к	External temperature
23 24	то	T_L	к	Temperature
25	Radiation type:		•	
27	ε	0	1	Surface emissivity
1.6	Tamb	0	к	Ambient temperature
Group:	J	epsilon htgh*sigma h	W/m <sup>2</sup>	
Select by group	Member of group(s):	epsnon_ntgn_sigma_n	1	Surface radiosity expression
Interior boundaries	member of group(s).		1. T	

FIGURE 5.116 2D axisymmetric Thermos\_Container\_3 model Boundary Settings (8, 14, 21) edit window

Enter L\_wall in the Heat transfer coefficient (h) expression in place of the L\_htgh term for boundary 28:

```
h_ave(T[1/K],Tinf_htgh[1/K],L_wall[1/m])[W/(m^2*K)]
```

Enter T\_0 in the External temperature  $(T_{inf})$  edit window for boundary 28. See Figure 5.117.

Enter L\_top in the Heat transfer coefficient (h) expression in place of the L\_htgh term for boundary 34:

```
h_ave(T[1/K],Tinf_htgh[1/K],L_top[1/m])[W/(m^2*K)]
```

Enter T\_0 in the External temperature  $(T_{inf})$  edit window for boundary 34. See Figure 5.118.

Click OK.

# Table 5.42 Boundary Settings–General Heat Transfer Edit Window

Boundary	<b>Boundary Condition</b>	Library Coefficient	Figure Number
28	Heat flux	Nat. Vertical wall, L=height	5.117
34	Heat flux	Nat. Horiz. plane, Upside, L=width	5.118

$\mathbf{h} \cdot (-k \nabla T) = \mathbf{q}_0 + \mathbf{h} (T_{inf} - T)$				
Boundaries Groups	Boundary Cond	ition Highly Conductiv	re Layer	Element Color/Style
Boundary selection	-Boundary sources and	d constraints		
19	Library coefficient:	Nat. Vertical wall, L=h	eight_1 🗘	Load
20. 21	Boundary condition:	Heat flux		
22	Quantity	Value/Expression	Unit	Description
24	a <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
25	h	h_ave(T[1/K],Tinf_htg	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
27	Tint	T_0	к	External temperature
29. 30	то	273.15	к	Temperature
31	Radiation type:	None	٥	
33 34 <del>4</del>	ε	0	1	Surface emissivity
Group:	Tamb	0	к	Ambient temperature
Select by group	J <sub>O</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

FIGURE 5.117 2D axisymmetric Thermos\_Container\_3 model Boundary Settings (28) edit window

uation				
$(-k\nabla T) = q_0 + h(T_{inf} - T)$				
Boundaries Groups	Boundary Cond	ition Highly Conduction	ve Layer	Element Color/Style
Boundary selection	Boundary sources and	d constraints		
19	Library coefficient:	Nat. Horiz. plane, Upsi	de, L=widt	Load
20	0-2500000000000000000000000000000000000			
21	Boundary condition:	Heat flux	•	
23	Quantity	Value/Expression	Unit	Description
2.4	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
25	h	h_ave(T[1/K],Tinf_hte	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
27		n_ave(1[1/K],11n1_nti		Meat transfer coefficient
28	Tint	T_0	к	External temperature
29 30	τ <sub>0</sub>	273.15	к	Temperature
31	Radiation type	None	0	
33	ε	0	1	Surface emissivity
34 🔹	Tamb	0	к	Ambient temperature
Group:	Jo	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Select by group		epsilon_ntgn-sigma_n		Surface radiosity expression
Interior boundaries	Member of group(s):		1	

FIGURE 5.118 2D axisymmetric Thermos\_Container\_3 model Boundary Settings (34) edit window

Free Mesh Parameters	
Subdomain Boundary Point Advanced	ОК
Subdomain mesh parameters Maximum element size: 0.002 Element growth rate: Method: Quad \$	Cancel Apply Help
	Subdomain mesh parameters Maximum element size: 0.002 Element growth rate:

**FIGURE 5.119** 2D axisymmetric Thermos\_Container\_3 model Free Mesh Parameters edit window

# **Mesh Generation**

From the menu bar, select Mesh > Free Mesh Parameters > Subdomain 1, 2, 3, 6, 7, 8. Enter Maximum element size 0.002. Select Method > Quad. Click the Remesh button. See Figure 5.119.

Click OK. See Figure 5.120.

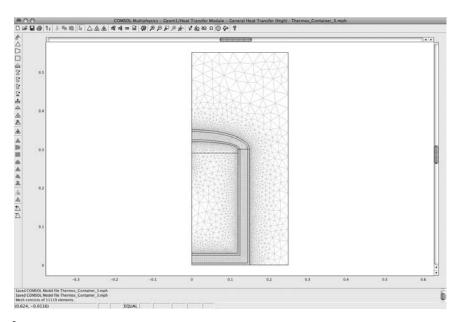
# Solving the 2D Axisymmetric Thermos\_Container\_3 Model

From the menu bar, select Solve > Solver Parameters > Parametric. Enter T\_L in the Parameter name edit window. Enter 273.15:9.5:368.15 in the Parameter values edit window. See Figure 5.121. Click OK.

From the menu bar, select Solve > Solve Problem. See Figure 5.122.

### Postprocessing

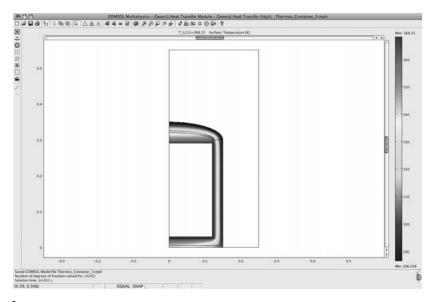
Select Postprocessing > Plot Parameters > Surface. Select "°C [degC]" in the Unit pull-down list. See Figure 5.123.

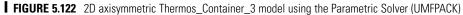


**FIGURE 5.120** 2D axisymmetric Thermos\_Container\_3 model mesh

000	Solver Par	rameters
Analysis	General Par	arametric Stationary Adaptive Advanced
Stationary 🗘	Parameter	
Auto select solver	Parameter name:	T_L
Stationary	Parameter values:	273.15:9.5:368.15
Time dependent Eigenvalue	Linear system solver	
Parametric Stationary segregated	Linear system solver:	Direct (UMFPACK)
Parametric segregated	Preconditioner:	÷
		(Settings)
Adaptive mesh refinement		
	Matrix symmetry:	Automatic
		(Help Apply Cancel OK

**FIGURE 5.121** 2D axisymmetric Thermos\_Container\_3 model Solver Parameters edit window





Coloring: Interpolated 🐳 Fill style: Filled 📢	Surface Data       Height Data         Predefined quantities:       Temperature       Image: Coloring:         Expression:       T       Image: Coloring:         Duit:       Image: Coloring:       Image: Coloring:         Coloring:       Interpolated       Fill style:         Fill coloring:       Image: Colors:       1024         Image: Color color       Image: Color scale	General Surface	Contour	Boundary	A	rrow	Principal
Surface Data       Height Data         Predefined quantities:       Temperature         Expression:       T         Unit:       °C         oloring and fill         coloring:       Interpolated         Fill style:       Filled         wrface color         © Colorrmap:       jet         Colors:       1024         © Colorrs:       1024	Surface Data       Height Data         Predefined quantities:       Temperature       Image: Temperature         Expression:       T       Image: Temperature         Expression:       T       Image: Temperature         Unit:       Image: Temperature       Image: Temperature         coloring and fill       Image: Temperature       Image: Temperature         coloring:       Interpolated       Fill style:         Image: Temperature       Image: Temperature       Image: Temperature         coloring:       Interpolated       Fill style:         Image: Temperature       Image: Temperature       Image: Temperature         Image: Temperature       Image: Temperature       Image: Temperature         coloring:       Image: Temperature       Image: Temperature         Image: Temperature       Image: Temperature       Image: Temperature         coloring:       Image: Temperature       Image: Temperature         Image: Temperature       Image: Temperature       Temperature         coloring:       Temperature       Temperature         Image: Temperature       Image: Temperature       Temperature         coloring: Temperature       Temperature       Temperature         Image: Temperature       Temperature <t< td=""><td>Surface plot</td><td>-</td><td></td><td></td><td></td><td></td></t<>	Surface plot	-				
Expression: T Smooth Unit: Coloring and fill Coloring: Interpolated Fill style: Filled Fill style: Filled Color scale Colorimap: jet Colors: 1024 V Color scale	Expression: T Smooth Unit: Coloring and fill Coloring: Interpolated Fill style: Filled Fill style: Filled Color scale Colorimap: iet Colors: 1024 Color scale	-	Surface Data	Height Data			
Unit: OC  Coloring and fill Coloring: Interpolated  Fill style: Filled  Color  Color  Color  Colors: 1024 Color scale	Unit: Coloring and fill Coloring: Interpolated Fill style: Filled urface color Colormap: It Colors: 1024 Color scale	Predefined quantities:	Temperature		•	Range	)
Coloring and fill Coloring: Interpolated Surface color Colormap: jet Colors: 1024 Color scale	Coloring and fill Coloring: Interpolated Fill style: Filled wrface color Colormap: iet Colors: 1024 Color scale	Expression:	т		_	Smo	oth
Coloring: Interpolated Fill style: Filled Surface color Colormap: jet Colors: 1024 Color scale	Coloring: Interpolated Fill style: Filled urface color O Colormap: jet Colors: 1024 Color scale	Unit:	°c		•		
			jet 🛟	Colors: 102	4	🗹 Colo	r scale
		Uniform color: (	Color				
		Uniform color: (	Color				
		Uniform color: (	Color				
		Uniform color: (	Color	•			



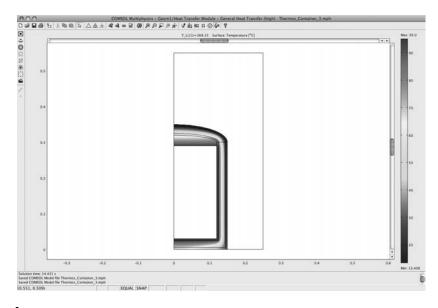


FIGURE 5.124 2D axisymmetric Thermos\_Container\_3 model surface temperature (°C)

Click OK. See Figure 5.124.

Given that our main interest in creating the 2D axisymmetric Thermos\_ Container\_3 model is to examine the heat transfer, the next step is to determine the heat loss. Select Postprocessing > Boundary Integration. Select boundaries 28 and 34 (the wall and top of the tank, respectively). Select "Normal total heat flux" in the Predefined quantities pull-down list. Check the Compute surface integral (for axisymmetric modes) check box. See Figure 5.125.

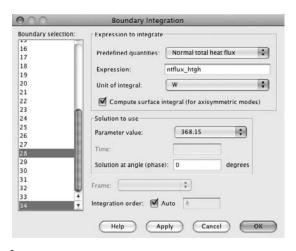


FIGURE 5.125 2D axisymmetric Thermos\_Container\_3 model Boundary Integration edit window

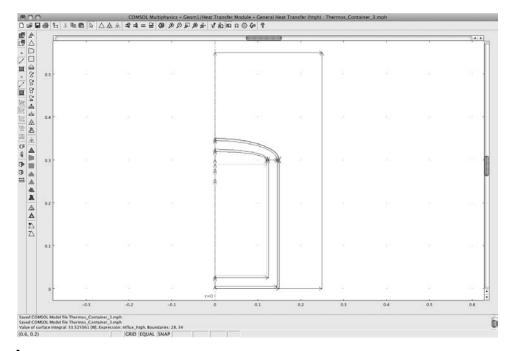


FIGURE 5.126 2D axisymmetric Thermos\_Container\_3 model user interface display window

Click OK. The result of the Boundary Integration (~34 W) is displayed as Value of surface integral: xx.xxxx [W], Expression: ntflux\_htgh, Boundaries: 28, 34 in the display window at the bottom of the COMSOL user interface. The amount of energy lost is approximately 41% of that lost using the urethane foam insulation in the 2D axisymmetric Thermos\_Container\_1 model (~82 W). See Figure 5.126.

## **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 5.127.

Click the Start Animation button. See Figure 5.128.

Alternatively, you can play the file Movie5\_TC\_3.avi that was supplied with this book.

# 2D Axisymmetric Thermos\_Container Models: Summary and Conclusions

The models presented in this section of Chapter 5 have introduced the following concepts: two-dimensional axisymmetric modeling (Axial symmetry [2D]), cylindrical coordinates, conductive media DC, Heat Transfer Module, heat conduction theory, opaque and transparent thermally conductive materials, export and import of CAD

Principal Str	eamline Particle Trac	ing Max/Min	Deform	Animat
Movie settings		Solutions to use		
				-
File type:	AVI	Select via:		\$
Width (in pixels):	640	273.15		
		282.65		
Height (in pixels):	480	292.15		
Frames per second:	10	301.65 311.15		
	( Advanced )	320.65		
	Advanced	330.15		U
Static / Eigenfunctio	n animation	339.65		¥ .
Cycle type:	Full harmonic	349.15 358.65		Ŧ
		Times:		
Number of frames:	11			
Reverse direction           Use camera setti	ngs from main window			
			(Start Ar	imation

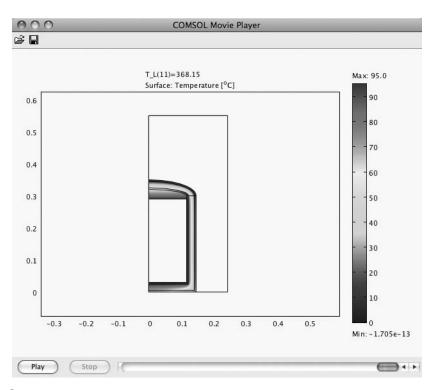
FIGURE 5.127 2D axisymmetric Thermos\_Container\_3 model Plot Parameters window

drawings (.dxf files), heat transfer coefficients, and vacuum. Previously introduced concepts employed in this section include the triangular mesh, free mesh parameters, subdomain mesh, maximum element size, and quadrilateral mesh (quad).

A comparison of the calculated results for the three thermos container models is shown in Table 5.43. As can be readily observed, the presence of a vacuum cavity significantly reduces the rate of heat flow through the model and the associated heat loss.

Model Number	Materials Used	Vacuum	Heat Loss	∆ <b>W</b> (%)
1	304ss, urethane foam	No	~82 W	_
2	304ss	Yes	~37 W	~45%
3	Silica glass	Yes	~34 W	~41%

#### Table 5.43 Thermos Container Modeling Results Summary



**FIGURE 5.128** 2D axisymmetric Thermos\_Container\_2 model animation, final frame

# References

- 1. http://en.wikipedia.org/wiki/History\_of\_thermodynamics
- 2. http://en.wikipedia.org/wiki/William\_Thomson,\_1st\_Baron\_Kelvin
- 3. http://en.wikipedia.org/wiki/James\_Prescott\_Joule
- 4. http://en.wikipedia.org/wiki/Ludwig\_Boltzmann
- 5. http://en.wikipedia.org/wiki/James\_Clerk\_Maxwell
- 6. http://en.wikipedia.org/wiki/Max\_Planck
- 7. http://en.wikipedia.org/wiki/Issac\_newton
- http://en.wikipedia.org/wiki/Newton%27s\_Law\_of\_Cooling#Newton.27s\_law\_ of\_cooling
- 9. http://en.wikipedia.org/wiki/Joseph\_Fourier
- 10. http://en.wikipedia.org/wiki/Fourier%27s\_Law
- 11. A. D. Cameron, J. A. Casey, and G. B. Simpson, NAFEMS Benchmark Tests for Thermal Analysis (Summary), NAFEMS Ltd., 1986.

- 12. J. R. Davis, ed., *Metals Handbook Desk Edition*, second edition, ASM International, 1998.
- 13. http://en.wikipedia.org/wiki/Niobium
- 14. http://en.wikipedia.org/wiki/Sir\_James\_Dewar
- 15. http://en.wikipedia.org/wiki/Royal\_Society
- 16. http://en.wikipedia.org/wiki/Thermos
- 17. ASM International, *Engineered Materials Desk Edition*, "Thermal Analysis and Properties of Polymers," Table 22, Thermal and related properties of amino resins (urethane foam).
- 18. http://www.matweb.com/ (glass)
- 19. http://www.matweb.com/ (304 stainless steel)

# Exercises

- 1. Build, mesh, and solve the COMSOL 2D axisymmetric cylinder conduction model problem presented in this chapter.
- 2. Build, mesh, and solve the first variation of the 2D axisymmetric cylinder conduction model problem presented in this chapter.
- 3. Build, mesh, and solve the second variation of the 2D axisymmetric cylinder conduction model problem presented in this chapter.
- 4. Build, mesh, and solve the 2D axisymmetric Thermos\_Container model presented in this chapter.
- 5. Build, mesh, and solve the first variation of the 2D axisymmetric Thermos\_Container model presented in this chapter.
- 6. Build, mesh, and solve the second variation of the 2D axisymmetric Thermos\_ Container model presented in this chapter.
- 7. Explore other variations of the arguments in the COMSOL 2D axisymmetric cylinder conduction models.
- 8. Explore other variations of the arguments in the 2D axisymmetric Thermos\_ Container models.
- 9. Explore how an increase in the pressure modifies the behavior of the COMSOL 2D axisymmetric cylinder conduction model.
- 10. Explore how changes in the tank geometry affect the heat loss in the 2D axisymmetric Thermos\_Container model.

# 6

# 2D Simple Mixed-Mode Modeling

# In This Chapter

2D Mixed-Mode Guidelines for New COMSOL® Multiphysics® Modelers 2D Mixed-Mode Modeling Considerations 2D Coordinate System 2D Axisymmetric Coordinate System Joule Heating and Heat Conduction Theory Heat Conduction Theory 2D Resistive Heating Modeling 2D Resistive Heating Model First Variation on the 2D Resistive Heating Model Second Variation on the 2D Resistive Heating Model, Including Alumina Isolation 2D Resistive Heating Models: Summary and Conclusions 2D Inductive Heating Considerations 2D Axisymmetric Coordinate System 2D Axisymmetric Inductive Heating Model First Variation on the 2D Axisymmetric Inductive Heating Model Second Variation on the 2D Axisymmetric Inductive Heating Model 2D Axisymmetric Inductive Heating Models: Summary and Conclusions

# 2D Mixed-Mode Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers

# 2D Mixed-Mode Modeling Considerations

It is assumed, at this point, that the reader has been exposed, at least briefly, to the information contained in Chapters 4 and 5. In this chapter, the basic material from Chapters 4 and 5 is utilized and somewhat expanded. In the earlier chapters, models were built and then solved using a quasi-static approach. In this chapter, the transient (time-dependent) method of solution is introduced. Transient models are intrinsically more difficult than quasi-static models. Transient models require a firmer understanding of the underlying physics and a more complete characterization of the materials employed in the model.

NOTE In transient or time-dependent (e.g., dynamic, unsteady) models, at least one of the dependent variables changes as a function of time.

These 2D models implicitly assume, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables that are of interest are homogeneous, isotropic, and constant unless otherwise specified (e.g., time dependent) throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model.

In the two models presented in this chapter, the resistive heating model and the inductive heating model, heat is generated within the body of the modeled materials through the same mechanism, Ohm's law<sup>1</sup> (i.e., Joule heating<sup>2</sup>), by two fundamentally different, but similar methods. In the resistive heating models, heat is generated by the flow of direct current (DC)<sup>3</sup> through the body in the models, resulting in Joule heating. As the body heats, the temperature rises. Because the resistivity depends on the temperature, the resistivity (conductivity) changes and consequently the amount of heat generated within the body changes, and so on.

In the inductive heating model, eddy currents (alternating currents [AC]<sup>4</sup>) are induced within the material of the modeled body. Heat is generated by the flow of the induced alternating current within the body, generating Joule heating. As the body heat increases, the temperature rises. Similarly, because the resistivity depends on the temperature, the resistivity (conductivity) changes and consequently the amount of heat generated within the body changes, the temperature rises, and so on.

As mentioned in previous chapters, it is always preferable for the modeler to be able to accurately anticipate the expected behavior (results) of the model and to understand how those results should be presented. Never assume that the default values that are initially present when the model is first created will suit the needs of a new model. Always verify that the values employed in the model are the correct ones needed for that model. Calculated solution values that significantly deviate from the expected values or from comparison values measured in experimentally derived realistic models are probably indicative of one or more modeling errors either in the original model design, in the earlier model analysis, or in the understanding of the underlying physics, or are simply due to human error.

# 2D Coordinate System

Two different 2D coordinate systems are employed in the models that are built in this chapter. In the first set of models (resistive heating), the basic 2D coordinate system plus time is employed. The second set of models (inductive heating) employs the 2D axisymmetric coordinate system plus time. Each of the coordinate systems was chosen

to facilitate the modeler building the least difficult model necessary to achieve a reasonably accurate demonstration of the principles involved and achieve a good first approximation result.

Note Because it is completely impossible to accommodate all variable factors into any scientific or engineering problem larger than the two-body problem,<sup>5</sup> each scientific or engineering calculation yields an approximate result. A good first approximation result is derived from a calculation that yields an answer that allows the modeler to determine the degree of feasibility of an adequate solution to the problem in question, within the limits of tolerable variance (error). All of the nonmodeling parameters need to be estimated either by the modeler, his or her power structure, or his or her accountant.

The purpose of the models presented here is to demonstrate the application of the chosen modeling techniques to applied physical prototypes, using measured materials properties for commercially available materials. These first approximation result models can be modified and used by the modeler to build other exploratory candidate models to determine the feasibility of similar devices as part of a more complex development or analysis project.

In a steady-state solution to a 2D model, parameters can vary only as a function of position in space (x) and space (y) coordinates. Such a 2D model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (x) and space (y) and in time (t); see Figure 6.1.

The transient solution model is essentially a sequential collection of (quasi-static) solutions, except that one or more of the dependent variables (f(T, t)) has changed with time. The space coordinates (x) and (y) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values  $(x_{\min} \le x \le x_{\max})$  and  $(y_{\min} \le y \le y_{\max})$ . The time coordinate (t) represents the range of values  $(t_{\min} \le t \le t_{\max})$  from the beginning of the observation period  $(t_{\min})$  to the end of the observation period  $(t_{\max})$ .

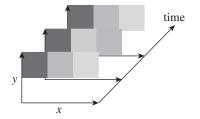
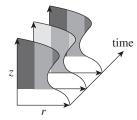


FIGURE 6.1 2D coordinate system, plus time



324

FIGURE 6.2 2D axisymmetric coordinate system, plus time

# 2D Axisymmetric Coordinate System

In the steady-state solution to any 2D axisymmetric model, parameters can vary only as a function of the radial position in space (r) and the axial position space (z) coordinates. Such a model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (r) and space (z) and in time (t); see Figure 6.2.

The transient solution model is essentially a sequential collection of steady-state (quasi-static) solutions. The space coordinates (r) and (z) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values ( $r_{\min} \le r \le r_{\max}$ ) and ( $z_{\min} \le z \le z_{\max}$ ). The time coordinate (t) represents the range of values ( $t_{\min} \le t \le t_{\max}$ ) from the beginning of the observation period ( $t_{\min}$ ) to the end of the observation period ( $t_{\max}$ ).

# **Joule Heating and Heat Conduction Theory**

Joule heating techniques are extremely important in device design considerations. Joule heating is applied to tasks as varied as heating houses (AC) and baking potatoes (microwave AC). It accounts for some of the most widely utilized technologies employed for research, design, and application in engineering and physics. Most modern products or processes require an understanding of Joule heating techniques either during development or during the use of the product or process (e.g., automobiles, plate glass fabrication, plastic extrusion, plastic products, houses, baked potatoes, ice cream).

NOTE Heating and heat transfer concerns have existed since the beginning of prehistory. There have been many contributors to our present understanding of the interaction of electric currents and solids. In this particular area, however, two scientists made especially notable contributions: Georg Ohm<sup>6</sup> and James Prescott Joule.<sup>7</sup> Ohm discovered Ohm's law:<sup>8</sup>

$$I = \frac{V}{R} \tag{6.1}$$

where

I =current in amperes (A)

V = voltage (electromotive force) in volts (V)

R = resistance in ohms

Joule discovered Joule's law:9

$$Q = I^2 \cdot R \cdot t \tag{6.2}$$

where Q = heat generated in joules (J) I = current in amperes (A) R = resistance in ohms t = time in seconds (S)

The first example presented in this chapter, the resistive heating model, explores the 2D electro-thermal interaction modeling of Joule heating using transient analysis. The model is solved for a material that is both electrically and thermally conductive. This model is implemented using the COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Electro-Thermal Application Mode.

In the first variation on the resistive heating model, the new model is built to explore a common configurational change and is solved using the same COMSOL Multiphysics Application Mode. In the second variation on this model, a model is built that incorporates materials modifications in addition to the configurational changes; it is solved using the COMSOL Multiphysics AC/DC Electro-Thermal Application Mode. The second variation also explores the influence of a low-pressure gas/vacuum environment on the model's properties. The calculated modeling results are then compared.

The second example, the induced heating model, explores the use of induced AC eddy currents to create Joule heating in a 2D axisymmetric model. The first and second variations on the induced heating model explore the effects of materials and parametric changes.

# **Heat Conduction Theory**

Heat conduction is a naturally occurring process that is readily observed in many aspects of modern life (e.g., refrigerators, freezers, microwave ovens, thermal ovens, engines). The heat transfer process allows both linear and rotational work to be done in the generation of electricity and the movement of vehicles. The initial understanding of transient heat transfer was developed by Newton<sup>10</sup> and started with Newton's law of cooling;<sup>11</sup>

$$\frac{dQ}{dt} = h^* A^* (T_{\rm S} - T_{\rm E}) \tag{6.3}$$

where 
$$\frac{dQ}{dt}$$
 = incremental energy lost in joules per unit time (J/s)  
A = energy transmission surface area (m<sup>2</sup>)  
h = heat transfer coefficient [W/(m<sup>2\*</sup>K]  
T<sub>s</sub> = surface temperature of the object losing heat (K)  
T<sub>E</sub> = temperature of the environment gaining heat (K)

Subsequent work by Jean Baptiste Joseph Fourier,<sup>12</sup> based on Newton's law of cooling, developed the law for steady-state heat conduction (known as Fourier's law<sup>13</sup>). Fourier's law is expressed here in differential form:

$$q = -k\nabla T \tag{6.4}$$

where

q = heat flux in watts per square meter (W/m<sup>2</sup>)

k = thermal conductivity of the material [W/(m\*K]

 $\nabla T$  = temperature gradient (K/m)

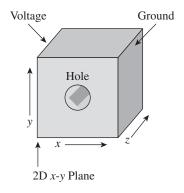
# 2D Resistive Heating Modeling

# **2D Resistive Heating Model**

The following numerical solution model (Resistive\_Heating\_1) is derived from a model that was originally developed by COMSOL as a Multiphysics demonstration model for distribution with the Multiphysics software in the basic Multiphysics Model Library. This model introduces the coupling of two important basic Application Modes: Joule Heating in the Conductive Media DC Application Mode and the Heat Transfer by Conduction Application Mode. The coupling of these two modes in this model demonstrates the interactions normally found in typical engineering materials.

It is important for the new modeler to personally build each model presented in this text. There is no substitute in the path to understanding of the modeling process for the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and subsequently correct errors, there by adding to his or her experience and fund of modeling knowledge. Even building the simplest model will expand the modeler's store of knowledge.

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either add heat or remove heat to achieve or maintain a desired temperature. Figure 6.3 shows a 3D rendition of the 2D resistive heating geometry, as will be modeled in this section.



**FIGURE 6.3** 3D rendition of the 2D resistive heating model

To start building the Resistive\_Heating\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (default setting) from the Space dimension pull-down list. Select COMSOL Multiphysics > Electro-Thermal Interaction > Joule Heating > Transient analysis. See Figure 6.4. Click OK.

0 0	Model Navigat	or
New Space dimension:	Model Library User Model 2D Control Contro Control Control Control Control Control Con	
	r-state analysis ient analysis I Interaction	Description: A predefined combination of application modes for Joule heating (resistive heating). Combines Conductive Media DC with heat transfe by conduction for modeling of electro-thermal applications such as
Dependent variables: Application mode name:	T V ht dc	Joule heating. Transient analysis.
Element:	Lagrange - Quadra	Multiphysics

**FIGURE 6.4** 2D Resistive\_Heating\_1 Model Navigator setup

Name	Expression	Description
r_Cu	1.754e-8[ohm*m]	Resistivity of copper at T_ref
T_ref	20[degC]	Reference temperature
alpha_Cu	3.9e-3[1/K]	Temperature coefficient copper
V_0	1e-1[V]	Electric potential (voltage)
T_air	300[K]	Air temperature
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity copper
rho_Cu	8.96e3[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity copper

#### Table 6.1 Constants Edit Window

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.1; also see Figure 6.5. Click OK.

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.0 and a height of 1.0. Select "Base: Center" and set x equal to 0 and y equal to 0 in the Rectangle edit window. See Figure 6.6.

Name	Expression	Value	Description
r Cu T_ref alpha_Cu V_0 T_air t_air k_Cu k_Cu Cp_Cu Cp_Cu	1.754e-8[ohm*m] 20[degC] 3.9e-3[1/K] 1e-1[V] 300[K] 3.94e2[W/(m*K)] 8.96e3[kg/m^3] 3.8e2[J/(kg*K)]	(1.754e-8)[Ω·m] 293.15[K] 0.0039[1/K] 0.1[V] 300[K] 394[W/(m·K)] 8960[kg/m <sup>3</sup> ] 380[J/(kg·K)]	Resistivity of copper at T ref Reference temperature Temperature coefficient Electric potential (voltage) Air temperature Thermal conductivity copper Density copper Heat capacity copper
C = 1		Help	Apply Cancel OK

**FIGURE 6.5** 2D Resistive\_Heating\_1 model Constants edit window

Size		Rotat	ion angle	
Width:	1.0	α:	0	(degrees
Height:	1.0			
Base:	Center	\$ Style:	Solid	¢
x:	0	Name:	R1	
	1		4	

**FIGURE 6.6** 2D Resistive\_Heating\_1 model Rectangle edit window

Click OK, and then click the Zoom Extents button. See Figure 6.7.

Using the menu bar, select Draw > Specify Objects > Circle. In the Circle edit window, enter a radius of 0.1 and a base of "Center." Set x equal to 0 and y equal to 0. See Figure 6.8.

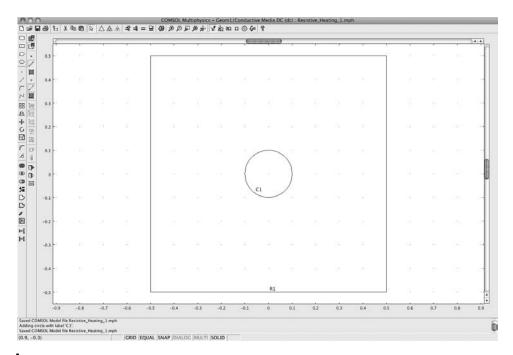
Click OK. See Figure 6.9.

Size		Rotation angle
Radius:	0.1	α: 0 (degrees
Positior	1	
Base:	Center	Style: Solid
x:	0	Name: C1
y:	0	

FIGURE 6.8 2D Resistive\_Heating\_1 model Circle edit window

The rectangle is the 2D representation of a cube in cross section. The circle is added to the 2D geometry to allow the creation of a hole through the cube, as shown in Figure 6.3.

Using the menu bar, select Draw > Create Composite Object. In the Set formula edit window, enter R1-C1. See Figure 6.10.



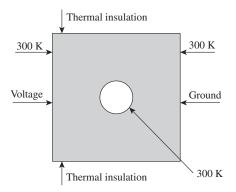
**FIGURE 6.9** 2D Resistive\_Heating\_1 model rectangle and circle

000 0	reate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: R1 C1	Set formula:          R1-C1         Image: Comparison of the state of the
	Repair tolerance: 1.0E-4

I FIGURE 6.10 2D Resistive\_Heating\_1 model Create Composite Object edit window

Click OK. See Figure 6.11.

This model introduces the coupling of two basic Application Modes: Joule Heating in the Conductive Media DC Application Mode and the Heat Transfer by Conduction Application Mode. The Physics subdomain and boundary settings will



**FIGURE 6.12** 2D Resistive\_Heating\_1 model boundary conditions overview

need to be specified in each mode separately. Figure 6.12 shows an overview of the boundary conditions for the combination of both modes.

First, however, the subdomain settings values need to be specified.

#### Physics Subdomain Settings: Heat Transfer by Conduction (ht)

Having established the geometry for the 2D Resistive\_Heating\_1 model of a block with a hole, the next step is to define the fundamental Physics conditions. Using the menu bar, select Multiphysics > Heat Transfer by Conduction (ht).

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). In the Subdomain edit windows, enter the information shown in Table 6.2. See Figure 6.13.

Note  $\checkmark$  For transient calculations, all of the physical property values are required for the conduction calculation. If *C*p and rho are set to zero, the implication is that the material is a perfect vacuum, which is logically inconsistent with the stated value of *k*.

Select the Init tab. Enter T\_ref in the Initial value edit window. See Figure 6.14. Click OK.

Name	Expression	Description
k (isotropic)	k_Cu	Thermal conductivity
ρ	rho_Cu	Density
$C_{P}$	<i>C</i> p_Cu	Heat capacity

#### Table 6.2 Subdomain Edit Window

$_{s}\rho C_{p}\partial T/\partial t - \nabla \cdot (k\nabla T) = Q + h_{tr}$	$ans(T_{ext}-T) + C_{trans}(T_{ambtrans})$	$^{4} - T^{4}$ ), T = temperature		
Subdomains Groups		Physics Init	Element	Color
Subdomain selection	Thermal properties a	nd heat sources/sinks		
1 (default)	Library material.	:	Load	
	Quantity	Value/Expression	Unit	Description
	δ <sub>ts</sub>	1	1	Time-scaling coefficient
	le (isotropic)	k_Cu	W/(m⋅K)	Thermal conductivity
	🔿 k (anisotropic)	400 0 0 400	W/(m·K)	Thermal conductivity
	ρ	rho_Cu	kg/m <sup>3</sup>	Density
	C <sub>p</sub>	Cp_Cu	J/(kg·K)	Heat capacity at constant pressure
	Q	Q_dc	W/m <sup>3</sup>	Heat source
	h <sub>trans</sub>	0	W/(m <sup>3</sup> ·K)	Convective heat transfer coefficient
Group: default	Text	0	к	External temperature
Select by group	C <sub>trans</sub>	0	W/(m <sup>3</sup> ⋅K <sup>4</sup> )	User-defined constant
Active in this domain	Tambtrans	0	К	Ambient temperature

**FIGURE 6.13** 2D Resistive\_Heating\_1 model Subdomain Settings edit window

Subdomains Groups		Physics	Init Element Color	]
Subdomain selection	T(t <sub>0</sub> ) T_ref		K Temperature	
Group: default				
Group: default				

**FIGURE 6.14** 2D Resistive\_Heating\_1 model Subdomain Settings, Init edit window

Boundary	<b>Boundary Condition</b>	Value/Expression	Figure Number
1, 4–8	Temperature	T_air	6.15
2, 3	Thermal insulation	—	6.16

## Table 6.3 Boundary Settings–Heat Transfer by Conduction (ht) Edit Window

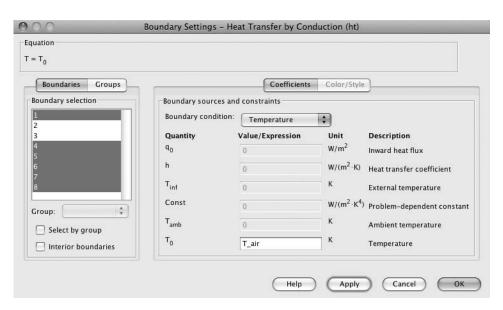
# **Physics Boundary Settings: Heat Transfer by Conduction (ht)**

Using the menu bar, select Physics > Boundary Setting. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.3. Click OK. See Figures 6.15 and 6.16.

# Physics Subdomain Settings: Conductive Media DC (dc)

Using the menu bar, in the Model Navigator menu, select Multiphysics > Conductive Media DC (dc). Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). Select "Linear temperature relation" from the Conductivity relation pull-down list. In the Subdomain edit windows, enter the information as shown in Table 6.4. See Figure 6.17.

NOTE At this point in the model, the generation of heat is coupled to the resistivity through the temperature change.



**FIGURE 6.15** 2D Resistive\_Heating\_1 model Boundary Settings (1, 4–8) edit window

FIGURE 6.16 2D Resistive\_Heating\_1 model Boundary Settings (2, 3) edit window

Select the Init tab. Enter  $V_0^{(1-x[1/m])}$  in the  $V(t_0)$  edit window. See Figure 6.18. Click OK.

NOTE The initial conditions assume a linear voltage drop across the body of the model. This is reflected in the initialization equation  $(V(t_0) = V_0*(1-x[1/m]))$ .

# Physics Boundary Settings: Conductive Media DC (dc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.5. Click OK. See Figures 6.19, 6.20, and 6.21.

#### Table 6.4 Subdomain Settings–Conductive Media DC (dc) Edit Window

Name	Expression	Description
$ ho_0$	r_Cu	Resistivity at reference temperature
α	alpha_Cu	Temperature coefficient
T <sub>0</sub>	T_ref	Reference temperature

# 336 Chapter 6 2D Simple Mixed-Mode Modeling

Subdomains Groups		Physics Init E	lement C	olor
Subdomain selection	Material properties and	d sources		
1 (default)	Library material:	÷ (	oad	
	Quantity	Value/Expression	Unit	Description
	Je	0 0	A/m <sup>2</sup>	External current density
	Qj	0	A/m <sup>3</sup>	Current source
	d	1	m	Thickness
	Conductivity relation:	Linear temperature r	elation 💲	
	ρ <sub>0</sub>	r_Cu	Ω·m	Resistivity at reference temp
Group: default	α	alpha_Cu	1/K	Temperature coefficient
	т	Т	к	Temperature
Select by group	т	T_ref	к	Reference temperature

I FIGURE 6.17 2D Resistive\_Heating\_1 model Subdomain Settings edit window

Initial value	Physics Init	Element Color	
Initial value	Physics Init	Element Color	
$V(t_0) V_0^{*}(1-x[1/m])$		V Electric potential	

I FIGURE 6.18 2D Resistive\_Heating\_1 model Subdomain Settings Init edit window

Boundary	Boundary Condition	Value/Expression	Figure Number
1	Electric potential	V_0	6.19
2, 3, 5–8	Electric insulation	—	6.20
4	Ground	_	6.21

# Table 6.5 Boundary Settings–Conductive Media DC (dc) Edit Window

Roundaries Groups	Conditions Color/Style
Boundary selection	Boundary sources and constraints Library material:
Group: 🔹	

FIGURE 6.19 2D Resistive\_Heating\_1 model Boundary Settings (1) edit window

Boundaries Groups	Conditions Color/Style
oundary selection	Boundary sources and constraints Library material:
Select by group Interior boundaries	

FIGURE 6.20 2D Resistive\_Heating\_1 model Boundary Settings (2, 3, 5–8) edit window

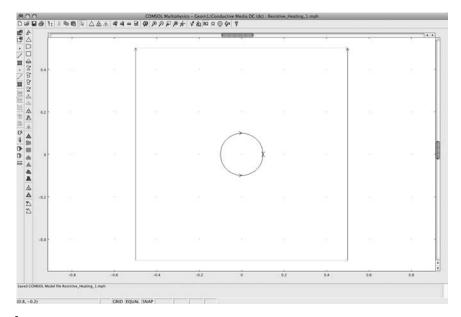
uation = 0	
Boundaries Groups Boundary selection 1 2 3 4 5 6 6 7 8 • • • • • • • • • • • • •	Conditions Culor/Style Boundary sources and constraints Library material: Load Boundary condition: Ground

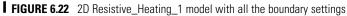
FIGURE 6.21 2D Resistive\_Heating\_1 model Boundary Settings (4) edit window

Figure 6.22 shows the 2D Resistive\_Heating\_1 model with all the boundary settings.

# **Mesh Generation**

On the toolbar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 4300 elements. See Figure 6.23.





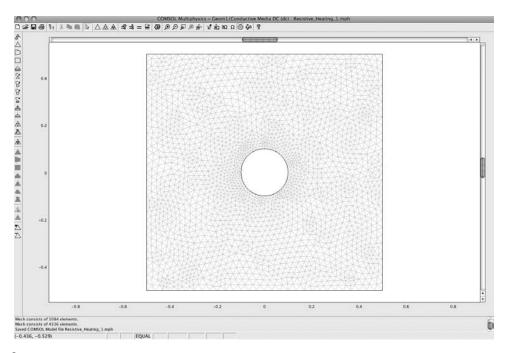


FIGURE 6.23 2D Resistive\_Heating\_1 model mesh window

# Solving the 2D Resistive\_Heating\_1 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter 0:50:2000 in the Times edit window. See Figure 6.24. Click OK.

### Time-Dependent Solving of the 2D Resistive\_Heating\_1 Model

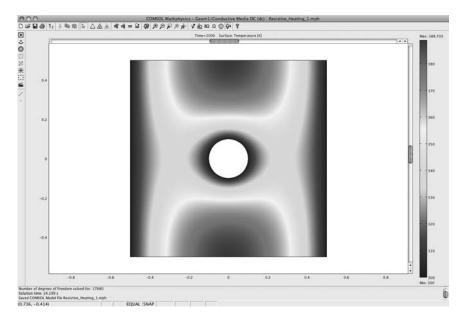
Select Solve > Solve Problem. See Figure 6.25.

### Postprocessing and Visualization

The default plot shows the temperature distribution in Kelvin. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot

000	Solver Paramete	ers
Analysis: Transient 🛊	General	Time Stepping Advanced
Auto select solver	Time stepping Times:	0:50:2000
Stationary Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated Carametric segregated Carametric segregated	Relative tolerance: Absolute tolerance:	0.01
	Preconditioner:	ct (UMFPACK)
	Matrix symmetry: Auto	omatic 🗘
	(He	elp Apply Cancel Ol

**FIGURE 6.24** 2D Resistive\_Heating\_1 model Solver Parameters edit window



**FIGURE 6.25** 2D Resistive\_Heating\_1 model solution

General Surface	Contour	Boundary	Arrow	Principal	1
	Contour	boundary			
Surface plot					
	Surface Data	Height Data			
Predefined quantities:	Temperature	\$	Range	D	
Expression:	Т		🗹 Smooth		
Unit:	°c	\$			
Coloring and fill					
Coloring: Interpo	lated	Fill style:	illed	\$	
Surface color					
O Colormap:	jet 🗘 (	olors: 1024	Color se	ale	
O Uniform color:			0		
O dillorin color.	Color				

**FIGURE 6.26** 2D Resistive\_Heating\_1 model Plot Parameters edit window

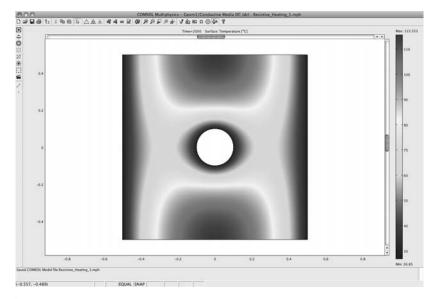
Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pulldown list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.26.

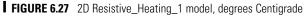
Click OK. See Figure 6.27.

It is relatively simple to demonstrate the heat flux. Select Postprocessing > Plot Parameters > Arrow. Check the Arrow plot check box. Select Heat Transfer by Conduction (ht) > Heat flux from the Predefined quantities pull-down list. Click the Color button and select a color such as "black." Click OK. See Figure 6.28.

Click OK. See Figure 6.29.

Next, because this is a transient analysis model, the modeler can test how close the solution is to the steady-state value. Select Postprocessing > Cross-Section Plot Parameters > General > Point plot. Verify that all of the Solutions to use are selected. See Figure 6.30.





Arrow plot	Plot arrows on. Subdomains 🛟
Subdoma	in Data Boundary Data Height Data
Predefined quantities:	Heat flux
x component:	fluxx_ht
y component:	fluxy_ht
Unit:	W/m <sup>2</sup>
x points:	of points Vector with coordinates
Arrow type. Arrow	rtional Color



				T	imir = 200	00 Sur	face: Te	mperate	ure (*Ci	Arrov	c Heat fi	28						Max
1				_		1	-	-	•								5414	
	- 1		+	-	1		1	14			4	1	-	-				
0.4			-	•	1	12				4		-	-+	~			- 11	8
			-	-	*	e		3	3	×.	•	-	1				- 11	
			-	×.		+	4	4	ł	÷				-			- 11	
0.2		-	-		7	4	1	4	1	i.			-	-			- 11	P
		60 ar	•		- 1	4	1		4	4	1		•	-			- 11	-
				8		2	r			8	κ.	*	2	-			1	
0			•		-						۴.		-	-				-
		-	-	1	1	1	-	-		1			1	-			- 11	
						1	1	Ť	+	*			1	-			- 11	
-0.2										,			-				- 11	1
					-	-	2		-	1			_				- 11	1
-0.4			-		1	10				10	-		-				- 11	
			-		4						-	-	_	_			- 11	
																	L.	
-0.8	-0.6	-0.4	6		-0.2			0			0.2			0.4	0.6	0.8		Min

**FIGURE 6.29** 2D Resistive\_Heating\_1 model, temperature and heat flux

Plot type O Line/Extrusion plot O Doint plot	
Solutions to use	Solution at angle (phase):
0 50 100 150 200 250 300 350 400 450 Times:	Frame:
lot in: New figure 🗘 🗌 Keep cur Display cross-section in main axes Title/Axis	Color

Gen	eral Line/Extrusion	Point
Point plot		
y-axis data		
Predefined quantities:	Temperature	\$
Expression:	т	
Unit:	0°C	\$
Coordinates		
x: 0		12
y: 0.4		
x-axis data Auto Expression	0	
н	elp Apply	Cancel OK

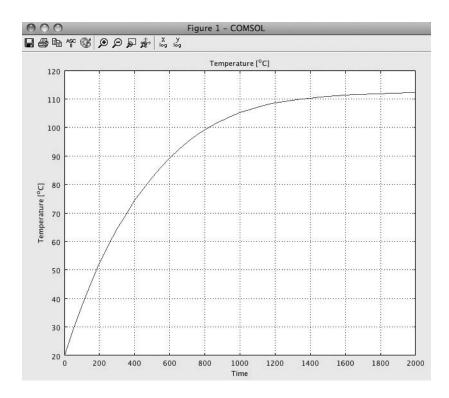
FIGURE 6.31 2D Resistive\_Heating\_1 model, Cross-Section Plot Parameters, Point edit window

Click the Point tab. Select "°C" from the Unit pull-down list. Enter x = 0, y = 0.4 in the Coordinates edit windows. See Figure 6.31.

Click OK. Figure 6.32 shows the temperature versus time plot for the point x = 0, y = 0.4. It is easily seen that the temperature is close to the steady-state value (the curve approaches the horizontal, small  $\Delta T$ ) at the end of the modeling calculation.

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.33.



**FIGURE 6.32** 2D Resistive\_Heating\_1 model, temperature versus time at x = 0, y = 0.4

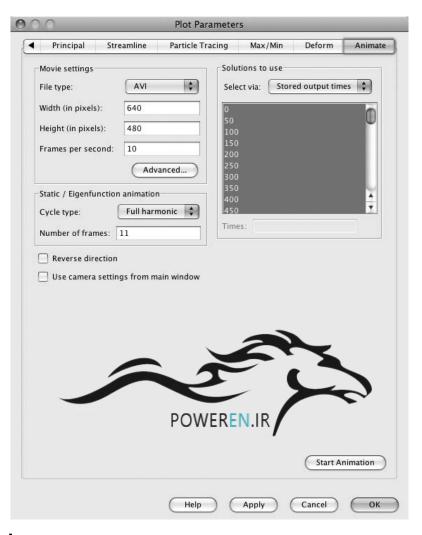
Click the Start Animation button. See Figure 6.34.

Alternatively, you can play the file Movie6\_RH\_1.avi that was supplied with this book.

# First Variation on the 2D Resistive Heating Model

The following numerical solution model (Resistive\_Heating\_2) is derived from the model Resistive\_Heating\_1. In this model, geometric and materials composition changes are introduced, such as might be used in a general industrial application. It is a multielement heating unit with Nichrome (a nickel–chromium alloy) heating bars and copper connecting bars.

**NOTE** The Resistive\_Heating\_2 model demonstrates materials and a configuration as might be employed in heat sealers, soldering heads, packaging equipment, and printed circuit board processing equipment.



**FIGURE 6.33** 2D Resistive\_Heating\_1 model, Plot Parameters window

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either add heat or remove heat so as to achieve or maintain a desired temperature. Figure 6.35 shows a 3D rendition of the 2D resistive heating geometry, as will be modeled here.

To start building the Resistive\_Heating\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from

**FIGURE 6.34** 2D Resistive\_Heating\_1 model animation, final frame

the Space dimension pull-down list. Select COMSOL Multiphysics > Electro-Thermal Interaction > Joule Heating > Transient analysis. See Figure 6.36. Click OK.

# Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.6; see also Figure 6.37. Click OK.

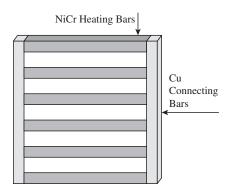


FIGURE 6.35 3D rendition of the 2D Resistive\_Heating\_2 model (not to scale)

0 0	Model Navigat	or
New	Model Library User Model	ls Open Settings
Space dimension:	2D 🗘	Ro
▼ 📄 Joule Heat	nal Interaction ing /-state analysis ient analysis	Description:
AC/DC Module     Heat Transfer Mo     MEMS Module     GRF Module     GRF Module     Structural Mecha		A predefined combination of application modes for Joule heating (resistive heating). Combines Conductive Media DC with heat transfe by conduction for modeling of electro-thermal applications such as Joule heating.
Dependent variables: Application mode name:	T V ht dc	Transient analysis.
Flement:	Lagrange - Quadra 🛟	Multiphysics

**FIGURE 6.36** 2D Resistive\_Heating\_2 Model Navigator setup

## Table 6.6Constants Edit Window

Name	Expression	Description
V_0	1 [V]	Electric potential (voltage)
T_ref	20[degC]	Reference temperature
T_air	300[K]	Air temperature
r_Cu	1.754e-8[ohm*m]	Resistivity Cu at T_0
alpha_Cu	3.9e-3[1/K]	Temperature coefficient Cu
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity Cu
rho_Cu	8.96e3[kg/m^3]	Density Cu
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity Cu
r_NiCr	1.08e-6[ohm*m]	Resistivity NiCr at T_0
alpha_NiCr	1.7e-3[1/K]	Temperature coefficient NiCr
k_NiCr	1.13e1[W/(m*K)]	Thermal conductivity NiCr
rho_NiCr	8.4e3[kg/m^3]	Density NiCr
Cp_NiCr	4.5e2[J/(kg*K)]	Heat capacity NiCr

Name	Expression	Value	Description
V_0	1e-1[V]	0.1[V]	Electric potential (voltage)
T_ref	20[degC]	293.15[K]	Reference temperature
T_air	300[K]	300[K]	Air temperature
r_Cu	1.754e-8[ohm*m]	(1.754e-8)[Ω·m]	Resistivity Cu at T_0
alpha_Cu	3.9e-3[1/K]	0.0039[1/K]	Temperature coefficient Cu
k_Cu	3.94e2[W/(m*K)]	394[W/(m·K)]	Thermal conductivity Cu
rho_Cu	8.96e3[kg/m^3]	8960[kg/m <sup>3</sup> ]	Density Cu
Cp_Cu	3.8e2[J/(kg*K)]	380[J/(kg·K)]	Heat capacity Cu
r_NiCr	1.08e-6[ohm*m]	(1.08e-6)[Ω·m]	Resistivity NiCr at T_0
alpha_NiCr	1.7e-3[1/K]	0.0017[1/K]	Temperature coefficient NiCr
k_NiCr	1.13e1[W/(m*K)]	11.3[W/(m·K)]	Thermal conductivity NiCr
rho_NiCr	8.4e3[kg/m^3]	8400[kg/m <sup>3</sup> ]	Density NiCr
Cp_NiCr	4.5e2[J/(kg*K)]	450[J/(kg·K)]	Heat capacity NiCr
(			) + >
🖼 🔛		Help A	pply Cancel OK

FIGURE 6.37 2D Resistive\_Heating\_2 model Constants edit window

**NOTE** In building this model, the calculational parameters (e.g., constants, scalar expressions) have been consolidated into a convenient location (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

Using the menu bar, select Draw > Specify Objects > Rectangle. Create each of the rectangles indicated in Table 6.7. See Figure 6.38.

Using the menu bar, select Draw > Create Composite Object. Enter: R1+R7+R8-R2-R3-R4-R5-R6. See Figure 6.39.

R Number	Width	Height	Base	x	y
1	1.0	1.1	Center	0	0
2	0.9	0.1	Corner	-0.45	0.35
3	0.9	0.1	Corner	-0.45	0.15
4	0.9	0.1	Corner	-0.45	-0.05
5	0.9	0.1	Corner	-0.45	-0.25
6	0.9	0.1	Corner	-0.45	-0.45
7	0.05	1.1	Corner	-0.50	-0.55
8	0.05	1.1	Corner	-0.45	-0.55

Table 6.7	<b>Rectangle Edit Window</b>
-----------	------------------------------

**FIGURE 6.38** 2D Resistive\_Heating\_2 model created rectangles

Click OK, and then click the Zoom Extents button. See Figure 6.40.

**INOTE** In building this model, the same geometry has been built that will be used in the next model. To save the modeler some time, select File > Export > Geometry Objects to File. Enter RH2\_Geometry in the Save As edit window. Click the Save button.

0.000	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
R1	R1+R7+R8-R2-R3-R4-R5-R6
R2	
R3	Keep interior boundaries
R4	Repair
NSC-	Repair Repair tolerance: 1.0E-4

FIGURE 6.39 2D Resistive\_Heating\_2 model Create Composite Object edit window

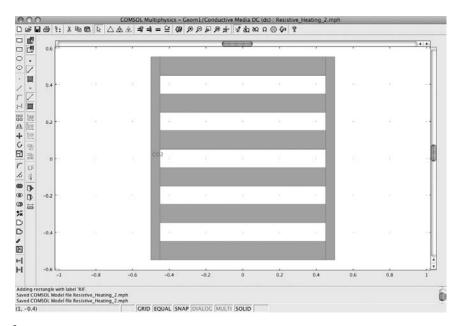


FIGURE 6.40 2D Resistive\_Heating\_2 model heater bar assembly

This model introduces the coupling of two basic Application Modes: Joule Heating in the Conductive Media DC Application Mode and the Heat Transfer by Conduction Application Mode. The Physics subdomain and boundary settings will need to be specified in each mode separately. Figure 6.41 shows an overview of the boundary conditions for the combination of both modes.

First, however, the subdomain settings values need to be specified.

Subdomain Number	Name	Expression	Description
1, 8	k (isotropic)	k_Cu	Thermal conductivity
	ρ	rho_Cu	Density
	$C_{P}$	Cp_Cu	Heat capacity

 Table 6.8
 Subdomain Settings Edit Window

## Physics Subdomain Settings: Heat Transfer by Conduction (ht)

Having established the geometry for the 2D Resistive\_Heating\_2 model of a heater bar assembly, the next step is to define the fundamental Physics conditions. Using the menu bar, select Multiphysics > Heat Transfer by Conduction (ht). Using the menu bar, select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 6.8. Click the Apply button. See Figure 6.42.

In the Subdomain edit windows, enter the information shown in Table 6.9. Click the Apply button. See Figure 6.43.

, p	$rans(T_{ext}-T) + C_{trans}(T_{ambtrans})$			
Subdomains Groups	1	Physics Init	Flement	Color
Subdomain selection	Thermal properties a	nd heat sources/sinks		
1 2 (default)	Library material:	:	Load	
3 (default)	Quantity	Value/Expression	Unit	Description
4 (default) 5 (default)	δ <sub>ts</sub>	1	1	Time-scaling coefficient
6 (default)	k (isotropic)	k_Cu	W/(m·K)	Thermal conductivity
7 (default) 8	() k (anisotropic)	400 0 0 400	W/(m·K)	I hermal conductivity
	P	rho_Cu	kg/m <sup>3</sup>	Density
	-	- Anna - Tanan	J/(kg·K)	•
	C <sub>p</sub>	Cp_Cu		Heat capacity at constant pressure
	Q	Q_dc	W/m <sup>3</sup>	Heat source
	h <sub>trans</sub>	0	W/(m <sup>3</sup> ·K)	Convective heat transfer coefficient
Group:	T <sub>ext</sub>	0	к	External temperature
	C <sub>trans</sub>	0	W/(m <sup>3</sup> ·K <sup>4</sup> )	User-defined constant
Select by group	Tambtrans	0	ĸ	Ambient temperature

**FIGURE 6.42** 2D Resistive\_Heating\_2 model Subdomain Settings (1, 8) edit window

Subdomain Number	Name	Expression	Description
2–7	k (isotropic)	k_NiCr	Thermal conductivity
	ρ	rho_NiCr	Density
	$C_{P}$	Cp_NiCr	Heat capacity

#### Table 6.9 Subdomain Settings Edit Window

**NOTE** For transient calculations, all of the physical property values are required for the conduction calculation. In this case, the properties of both copper (Cu) and Nichrome (NiCr) are required.

Select the Init tab. Select subdomains 1–8. Enter T\_ref in the Initial value edit window. See Figure 6.44. Click OK.

# Physics Boundary Settings: Heat Transfer by Conduction (ht)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.10. See Figures 6.45 and 6.46.

Subdomains Groups		Physics Init	Element	Color
Subdomain selection	Thermal properties a Library material:	nd heat sources/sinks	Load	
2 (default) 4 (default) 5 (default) 6 (default) 7 (default) 8	Quantity <sup>8</sup> ts (a) k (isotropic) (b) k (anisotropic) p C <sub>p</sub> Q h <sub>trans</sub>	Value/Expression	Unit 1 W/(m-K) W/(m-K) kg/m <sup>3</sup> J/(kg·K) W/m <sup>3</sup> ·K)	Description Time-scaling coefficient Thermal conductivity Thermal conductivity Density Heat capacity at constant pressure Heat source Convective heat transfer coefficient
Group: default	T <sub>ext</sub> C <sub>trans</sub> T <sub>ambtrans</sub>	0	Κ W/(m <sup>3</sup> ·K <sup>4</sup> ) K	External temperature User-defined constant Ambient temperature

FIGURE 6.43 2D Resistive\_Heating\_2 model Subdomain Settings (2–7) edit window

000	Subdomain Settings - H	eat Transfer by Con	duction (ht)		
Equation $\delta_{ts}\rho C_p \partial T/\partial t - \nabla \cdot (k \nabla T) = Q + h_{tra}$	ns(T <sub>ext</sub> -T) + C <sub>trans</sub> (T <sub>ambtrans</sub> <sup>4</sup> -	T <sup>4</sup> ), T= temperature			
Subdomains Groups		Physics Init	Flement Color		
Subdomain selection 1 2 (default) 3 (default) 4 (default) 5 (default) 6 (default) 7 (default) 8	T(t <sub>0</sub> ) T_ref		K Temperature		
Croup:		Help	Apply	Cancel	ОК

**FIGURE 6.44** 2D Resistive\_Heating\_2 model Subdomain Settings, Init edit window

Click OK. Figure 6.47 shows the final combined Heat Transfer by Conduction (ht) boundary settings.

# Physics Subdomain Settings: Conductive Media DC (dc)

Using the menu bar, select Multiphysics > Conductive Media DC (dc). Using the menu bar, select Physics > Subdomain Settings. Select Subdomains 1–8 in the Subdomain

	Boundary	Value/		Figure
Boundary	Condition	Expression	Click Apply	Number
1, 40	Temperature	T_air	Yes	6.45
2, 3, 5–7,	Thermal insulation	—	Yes	6.46
9–11, 13–15,				
17–19, 21–23,				
25, 26, 28, 29,				
31, 33, 35, 37, 39				

= T <sub>0</sub>				
Boundaries Groups		Coefficients	Color/Style	]
Boundary selection	Boundary sources an	d constraints		
32	Boundary condition:	Temperature	•	
34	Quantity	Value/Expression	Unit	Description
35 36	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
37	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
38 39	T <sub>inf</sub>	0	к	External temperature
40	Const	0	W/(m <sup>2</sup> ·K <sup>4</sup> )	Problem-dependent constan
Group:	T <sub>amb</sub>	0	— к	Ambient temperature
Select by group	τ <sub>o</sub>	T_air	к	Temperature

I FIGURE 6.45 2D Resistive\_Heating\_2 model Boundary Settings (1, 40) edit window

Boundaries Groups		Coefficients	Color/Style	]
Boundary selection	Boundary sources an	d constraints		-
32 33	Boundary condition:	Thermal insulation	\$	
34	Quantity	Value/Expression	Unit	Description
35	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
36 37	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
38 39	T <sub>inf</sub>	0	к	External temperature
40	Const	0	W/(m <sup>2</sup> ·K <sup>4</sup> )	Problem-dependent constan
Group:	T <sub>amb</sub>	0	к	Ambient temperature
Select by group Interior boundaries	то	0	к	Temperature

**FIGURE 6.46** 2D Resistive\_Heating\_2 model Boundary Settings (2, 3...) edit window

FIGURE 6.47 2D Resistive\_Heating\_2 model Combined Heat Transfer by Conduction (ht) boundary settings

selection window (all of the subdomains). Enter 0.01 in the Thickness (d) edit window. Select "Linear temperature relation" from the Conductivity relation pull-down list. Click the Apply button.

In the Subdomain edit windows, enter the information shown in Table 6.11. Click the Apply button. See Figure 6.48.

In the Subdomain edit windows, enter the information shown in Table 6.12. Click the Apply button. See Figure 6.49.

NOTE At this point in the model, the generation of heat is coupled to the resistivity in each different material (Cu, NiCr) through the temperature change.

Subdomain Number	Name	Expression	Description
1, 8	$ ho_0$	r_Cu	Resistivity at reference temperature
	α	alpha_Cu	Temperature coefficient
	T <sub>0</sub>	T_ref	Reference temperature

$\nabla \cdot \mathbf{d}(\sigma \nabla V - \mathbf{J}^{\mathbf{e}}) = \mathbf{d}\mathbf{Q}_{j}, \sigma = 1/(\rho_{0})$	$(1 + \alpha(T - T_0)))$			
Subdomains Groups		Physics Init El	ement (	olor ]
Subdomain selection	Material properties and	sources		
1 2 (default)	Library material:	:	oad	
3 (default)	Quantity	Value/Expression	Unit	Description
4 (default) 5 (default)	Je	0 0	A/m <sup>2</sup>	External current density
6 (default) 7 (default)	Qj	0	A/m <sup>3</sup>	Current source
8	d	0.01	m	Thickness
	Conductivity relation: Linear temperature relation			
	ρ <sub>0</sub>	r_Cu	Ω·m	Resistivity at reference tem
Group:	α	alpha_Cu	1/K	Temperature coefficient
Select by group	т	Т	к	Temperature
Active in this domain	то	T_ref	к	Reference temperature

**FIGURE 6.48** 2D Resistive\_Heating\_2 model Subdomain Settings (1, 8) edit window

Subdomains Groups		Physics Init E	lement (	Color
Subdomain selection	Material properties an	Material properties and sources		
1 2 (default)	Library material:	•	.oad)	
3 (default)	Quantity	Value/Expression	Unit	Description
4 (default) 5 (default)	Je	0 0	A/m <sup>2</sup>	External current density
6 (default) 7 (default)	Qj	0	A/m <sup>3</sup>	Current source
8	d	0.01	m	Thickness
	Conductivity relation:	Conductivity relation: Linear temperature relation		
	ρ <sub>0</sub>	r_NiCr	Ω·m	Resistivity at reference temp
Group: default	α	alpha_NiCr	1/К	Temperature coefficient
	т	т	к	Temperature
Select by group	т <sub>о</sub>	T_ref	к	Reference temperature

**FIGURE 6.49** 2D Resistive\_Heating\_2 model Subdomain Settings (2–7) edit window

Name	Expression	Description
$ ho_0$	r_NiCr	Resistivity at reference temperature
α	alpha_NiCr	Temperature coefficient
T <sub>0</sub>	T_ref	Reference temperature
	ρ <sub>0</sub> α	ρ <sub>0</sub> r_NiCr α alpha_NiCr

 Table 6.12
 Subdomain Settings–Conductive Media DC (dc) Edit Window

Select the Init tab. Select subdomains 1–8 in the Subdomain selection window (all of the subdomains). Enter  $V_0*(1 - x[1/m])$  in the  $V(t_0)$  edit window. See Figure 6.50. Click OK.

NOTE The initial conditions assume a linear voltage drop across the body of the model. This is reflected in the initialization equation  $(V(t_0) = V_0 * (1 - x[1/m]))$ .

## Physics Boundary Settings: Conductive Media DC (dc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.13. See Figures 6.51, 6.52, and 6.53.

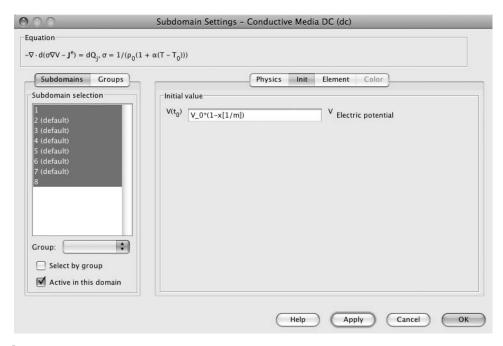


FIGURE 6.50 2D Resistive\_Heating\_2 model Subdomain Settings, Init edit window

Boundary	Boundary Condition	Value/ Expression	Figure Number
1	Electric potential	V_0	6.51
2, 3, 5–7,	Electric insulation	—	6.52
9–11, 13–15,			
17–19, 21–23,			
25, 26, 28, 29,			
31, 33, 35, 37, 39			
40	Ground	—	6.53

# Table 6.13 Boundary Settings–Conductive Media DC (dc) Edit Window

= V <sub>0</sub>	
Boundaries Croups Boundary selection	Conditions     Color/Style       Boundary sources and constraints     Library material       Library material
Interior boundaries	

**FIGURE 6.51** 2D Resistive\_Heating\_2 model Boundary Settings (1) edit window

uation J = 0	
Boundaries Croups Boundary selection 33 34 35 36 37 30 30 40 40 5 5 80 40 5 5 80 40 5 5 80 40 5 9 5 9 6 1 9 7 1 9 9 40 1 9 9 1 9 9 1 9 1 9 1 9 1 9 1 9 1 9 1	Conditions Color/Style Boundary sources and constraints Library material:

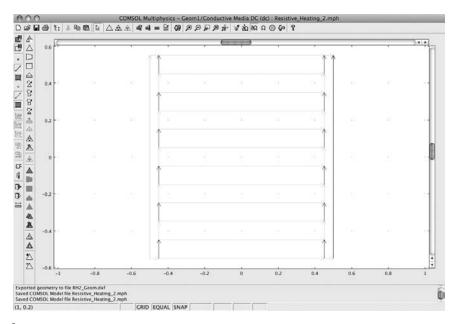
**FIGURE 6.52** 2D Resistive\_Heating\_2 model Boundary Settings (2, 3, 5–8...) edit window

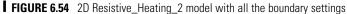
**FIGURE 6.53** 2D Resistive\_Heating\_2 model Boundary Settings (40) edit window

Click OK. Figure 6.54 shows the 2D Resistive\_Heating\_2 model with all the boundary settings.

#### **Mesh Generation**

On the toolbar, click the Initialize Mesh button once. Click the Refine Mesh button once. This results in a mesh of approximately 2200 elements. See Figure 6.55.





**FIGURE 6.55** 2D Resistive\_Heating\_2 model mesh window

# Solving the 2D Resistive\_Heating\_2 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter 0:50:2000 in the Times edit window. See Figure 6.56. Click OK.

# Time-Dependent Solving of the 2D Resistive\_Heating\_2 Model

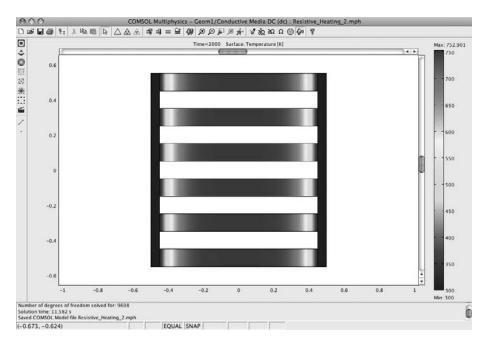
Select Solve > Solve Problem. See Figure 6.57.

# Postprocessing and Visualization

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot

900	Solver Par	ameters			
Analysis:	Ge	neral Time St	epping Ad	Ivanced	
Transient         Image: Constraint of the second	Time stepping Times: Relative tolerance: Absolute tolerance: Allow complex nu Linear system solver Linear system solver Preconditioner:	mbers Direct (UMFPA	0:50:200 0.01 0.0010 CK)	00	
	Matrix symmetry:	Automatic			
		Help	Apply	Cancel (	ок

**FIGURE 6.56** 2D Resistive\_Heating\_2 model Solver Parameters edit window



**FIGURE 6.57** 2D Resistive\_Heating\_2 model solution

	Surface	Contour	Boundary	Arrow	Principal
General	Surface	Contour	boundary	Arrow	Principal
Surface	plot				
		Surface Data	leight Data		
	1				
Predefined	quantities:	Temperature	\$	Range	
Expressio	n-	т		Smoot	h .
			1720	1 177 00 1929	0
Unit:		°C	\$		
Coloring an	d fill				
Coloring:	Interpol	ated 🗘	Fill style:	Filled	\$
Surface cold	or				
Colorn		jet 🗘 C	olors: 1024	Michael	
	-	Jet V	01015. 1024	Color s	scale
	m color:	Color			
O Unifor					
O Unifor					
O Unifor					
O Unifor					
O Unifor					
O Unifor					
() Unifor					
() Unifor					
O Unifor					
O Unifor					
O Unifor					
OUnifor					
OUnifor					
OUnifor		Help	Арріу	Can	

FIGURE 6.58 2D Resistive\_Heating\_2 model Plot Parameters edit window

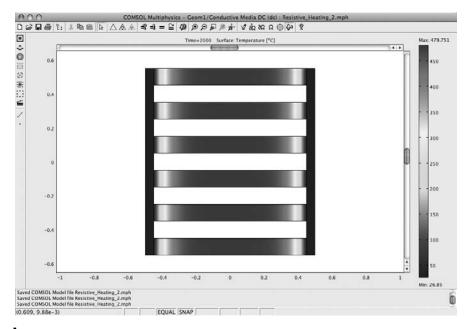
Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pull-down list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.58.

Click OK. See Figure 6.59.

Next, because this is a transient analysis model, the modeler can test how close the solution is to the steady-state value. Select Postprocessing > Cross-Section Plot Parameters > General > Point plot. Verify that all of the Solutions to use are selected. See Figure 6.60.

Click the Point tab. Enter x = 0, y = 0.1 in the Coordinates edit windows. See Figure 6.61.

Click OK. Figure 6.62 shows the temperature versus time plot for the point x = 0, y = 0.1. It is easily seen that the temperature is not close to the steady-state value (the curve is still rising, an almost linear  $\Delta T$ ) at the end of the modeling calculation.



**FIGURE 6.59** 2D Resistive\_Heating\_2 model, degrees centigrade

O Line/Extrusion plot O Point plo	ot
	Solution at angle (phase):
Select via: Stored output times	0 degrees
0	Frame:
so 🗸	¢
100 150	
200	
250	
300	
350	
400 450	
Times	
ot in: New figure 🗘 🗌 Keep c	urrent plot
	Color
Display cross-section in main axes	
d Display cross-section in main axes	

I FIGURE 6.60 2D Resistive\_Heating\_2 model, Cross-Section Plot Parameters, General edit window

Predefined quantities:	Temperature	•
Expression:	T	
Unit: Coordinates	K	
x: 0		
y: 0.1		
x-axis data Auto C Expression.		

FIGURE 6.61 2D Resistive\_Heating\_2 model, Cross-Section Plot Parameters, Point edit window

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.63.

Click the Start Animation button. See Figure 6.64.

Alternatively, you can play the file Movie6\_RH\_2.avi that was supplied with this book.

# Second Variation on the 2D Resistive Heating Model, Including Alumina Isolation

The following numerical solution model (Resistive\_Heating\_3) is derived from the model Resistive\_Heating\_2. In this model, geometric and materials composition changes are introduced, such as might be used in a general industrial application. This model introduces Alumina as the thermal and electrical insulator and employs the



FIGURE 6.63 2D Resistive\_Heating\_2 model, Plot Parameters window

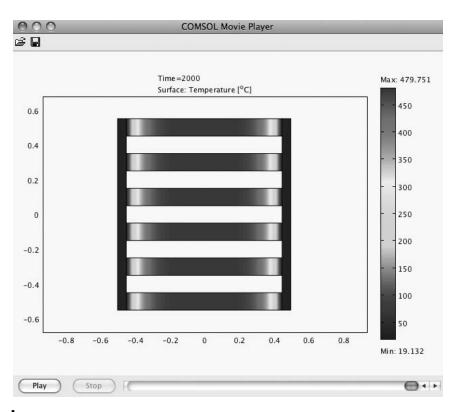


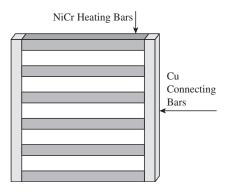
FIGURE 6.64 2D Resistive\_Heating\_2 model animation, final frame

Heat Transfer Module. It is a multielement heating unit with Nichrome (a nickelchromium alloy) heating bars and copper connecting bars.

NOTE The Resistive\_Heating\_3 model demonstrates materials and a configuration as might be employed in vacuum heat sealers, soldering heads, packaging equipment, and other vacuum processing equipment.

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either add heat or remove heat so as to achieve or maintain a desired temperature. Figure 6.65 shows a 3D rendition of the 2D resistive heating geometry, as will be modeled here.

To start building the Resistive\_Heating\_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select Heat Transfer Module > Electro-Thermal Interaction > Joule Heating > Transient analysis. See Figure 6.66. Click OK.





## Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information as shown in Table 6.14; also see Figure 6.67. Click OK.

NOTE In building this model, the calculational parameters (e.g., constants, scalar expressions) have been consolidated into a convenient location (e.g., a Constants File, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the

0 0	Model Naviga	ator
Space dimension:	Model Library User Model	)
<ul> <li>▶ ☐ k-€ Turbulenc</li> <li>▶ ☐ k-₩ Turbulenc</li> <li>▼   Electro-Therr</li> <li>▼ ☐ Joule Heat</li> <li>☐ Steady</li> </ul>	dule Transfer ion ressible Navier-Stokes e Model ce Model nal Interaction	Heat Transfer Module
Dependent variables	TJV	Transient analysis.
Application mode name: Element:	htgh emdc Lagrange - V <sub>2</sub> T <sub>2</sub> J <sub>1</sub>	Multiphysics

FIGURE 6.66 2D Resistive\_Heating\_3 Model Navigator setup

Name	Expression	Description
V_0	1[V]	Electric potential (voltage)
T_ref	20[degC]	Reference temperature
T_air	300[K]	Air temperature
r_Cu	1.754e-8[ohm*m]	Resistivity Cu at T_0
alpha_Cu	3.9e-3[1/K]	Temperature coefficient Cu
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity Cu
rho_Cu	8.96e3[kg/m^3]	Density Cu
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity Cu
r_NiCr	1.08e-6[ohm*m]	Resistivity NiCr at T_0
alpha_NiCr	1.7e-3[1/K]	Temperature coefficient NiCr
k_NiCr	1.13e1[W/(m*K)]	Thermal conductivity NiCr
rho_NiCr	8.4e3[kg/m^3]	Density NiCr
Cp_NiCr	4.5e2[J/(kg*K)]	Heat capacity NiCr

#### Table 6.14Constants Edit Window

settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

At this point, three alternate paths can taken. If the geometry was built and exported by building the Resistive\_Heating\_2 model, then the RH2\_Geometry.dxf file can be imported. If not, then the modeler can use the file that comes with the book. However, if the geometry has never been built, then follow the instructions given here. If one of the import paths is taken, then jump to the next Note.

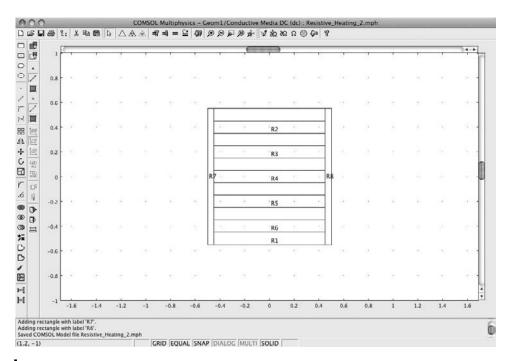
FIGURE 6.67 2D Resistive\_Heating\_3 model Constants edit window

R Number	Width	Height	Base	x	y
1	1.0	1.1	Center	0	0
2	0.9	0.1	Corner	-0.45	0.35
3	0.9	0.1	Corner	-0.45	0.15
4	0.9	0.1	Corner	-0.45	-0.05
5	0.9	0.1	Corner	-0.45	-0.25
6	0.9	0.1	Corner	-0.45	-0.45
7	0.05	1.1	Corner	-0.50	-0.55
8	0.05	1.1	Corner	0.45	-0.55

Using the menu bar, select Draw > Specify Objects > Rectangle. Create each of the rectangles indicated in Table 6.15. See Figure 6.68.

Using the menu bar, select Draw > Create Composite Object. Enter R1+R7+R82-R2-R3-R4-R5-R6. See Figure 6.69.

Click OK, and then click the Zoom Extents button.



**FIGURE 6.68** 2D Resistive\_Heating\_3 model created rectangles

000	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection	Set formula:
R1	R1+R7+R8-R2-R3-R4-R5-R6
R2 R3 R4 R5 R6	Keep interior boundaries

**FIGURE 6.69** 2D Resistive\_Heating\_3 model Create Composite Object edit window

NOTE	For imported file users, jump to here.	
NOIL	i of imported file abors, jump to here.	

Using the menu bar, select File > Import > CAD Data From File > RH2\_ Geometry.dxf. Using the menu bar, select Draw > Specify Objects > Rectangle. Create the rectangle indicated in Table 6.16.

Click the Zoom Extents button. See Figure 6.70.

This model introduces the coupling of two basic Application Modes: Joule Heating in the Conductive Media DC Application Mode and the Heat Transfer by Conduction Application Mode. The Physics subdomain and boundary settings will need to be specified in each mode separately. Figure 6.71 shows an overview of the boundary conditions for the combination of both modes.

First, however, the subdomain settings values need to be specified.

#### Physics Subdomain Settings: General Heat Transfer (htgh)

Having established the geometry for the 2D Resistive\_Heating\_3 model of a heater bar assembly, the next step is to define the fundamental Physics conditions. Using the menu bar, select Multiphysics > General Heat Transfer (htgh). Using the menu bar, select Physics > Subdomain Settings. Select Subdomains (3, 5, 7, 9, 11) > Load >

Table 6.16	Rectangle I	Edit Window
------------	-------------	-------------

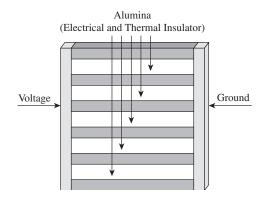
R Number	Width	Height	Base	x	y
1	1.0	1.1	Center	0	0

**FIGURE 6.70** 2D Resistive\_Heating\_3 model heater bar assembly

Basic Materials Properties > Alumina. Click OK, and then click the Apply button. See Figure 6.72.

In the Subdomain edit windows, enter the information shown in Table 6.17. Click the Apply button. See Figure 6.73.

In the Subdomain edit windows, enter the information shown in Table 6.18. Click the Apply button. See Figure 6.74.





spC <sub>p</sub> ∂T/∂t + ∇·(-k∇T) = Q = temperature Subdomains Groups Subdomain selection 1 2 (default) 3 4 (default) 5 6 (default) 9 10 (default) 11 12 (default) 13 Group: ♥♥♥	Conduction       Convection       Ideal Gas       Out-of-Plane       Init       Element       Color         Thermal properties and heat sources/sinks         Library material:       Alumina       Load         Quantity       Value/Expression       Unit       Description         δts       1       Time-scaling coefficient         • k (isotropic)       27[W/(m*K)]       W/(m·K)       Thermal conductivity         • k (anisotropic)       400 0 0 400       W/(m·K)       Thermal conductivity         • β       3900[kg/m^3]       kg/m <sup>3</sup> Uensity         C <sub>p</sub> 900[J/(kg*K)]       J/(kg·K)       Heat capacity at constant press         Q       Q_emdc       W/m <sup>3</sup> Heat source         Opacity:       Opaque       Image: State	
Group:		

FIGURE 6.72 2D Resistive\_Heating\_3 model Subdomain Settings (3, 5, 7, 9, 11) edit window

# Table 6.17 Subdomain Settings Edit Window

Subdomain Number	Name	Location	Description
1, 13	k (isotropic)	k_Cu	Thermal conductivity
	ρ	rho_Cu	Density
	CP	Cp_Cu	Heat capacity

# Table 6.18 Subdomain Settings Edit Window

Subdomain Number	Name	Expression	Description
2, 4, 6, 8, 10, 12	k (isotropic)	k_NiCr	Thermal conductivity
	ρ	rho_NiCr	Density
	$C_{P}$	Cp_NiCr	Heat capacity

= temperature Subdomains Groups	Conduction Co	nvection Ideal Gas	Out-of-Plane Init Element Color
Subdomain selection  2 (default) 3 4 (default) 5 6 (default) 7 8 (default) 9 10 (default) 11 12 (default) 13	Thermal properties an Library material: Quantity $\delta_{tc}$ O k (isotropic) O k (anisotropic) p $C_p$ Q Opacity:	Value/Expression  K_Cu  400 0 0 400  rho_Cu  Cp_Cu  Q_emdc  Opaque	Load       Description         1       Time-scaling coefficient         W/(m:K)       Thermal conductivity         W/(m:K)       Thermal conductivity         W/(m:K)       Thermal conductivity         kg/m3       Density         J/(kg·K)       Heat capacity at constant pressure         W/m3       Heat source
Group: 🔹			

**FIGURE 6.73** 2D Resistive\_Heating\_3 model Subdomain Settings (1, 13) edit window

Subdomains Groups	Physics	s Infinite Elements	Init Ele	ment Color
Subdomain selection	Material properties and	sources		
1 2 (default)	Library material:		oad	
3	Quantity	Value/Expression	Unit	Description
4 (default) 5	Je	0 0	A/m <sup>2</sup>	External current density
G (default)	Qj	0	A/m <sup>3</sup>	Current source
8 (default)	d	0.01	m	Thickness
9 10 (default)	Conductivity relation:	Linear temperature r	alation 🗘	
11 12 (default)	Po	r_NiCr	Ω·m	Resistivity at reference temp
13 Troup: default	α	alpha_NiCr	1/K	Temperature coefficient
	т	Т	к	Temperature
Select by group	τ <sub>0</sub>	T_ref	к	Reference temperature

**FIGURE 6.74** 2D Resistive\_Heating\_3 model Subdomain Settings (2, 4, 6, 8, 10, 12) edit window

quation $_{ts}\rho C_{\rho}\partial T/\partial t + \nabla \cdot (-k\nabla T) = Q$ = temperature	
Subdomains Groups	Conduction Convection Ideal Gas Out-of-Plane Init Element Color
Subdomain selection	Initial value
1 2 (default) 3 4 (default) 5 6 (default) 7 8 (default) 9 10 (default) 11 12 (default) 12 (default) 13	T(t <sub>0</sub> ) T_ref

**FIGURE 6.75** 2D Resistive\_Heating\_3 model Subdomain Settings, Init edit window

NOTE For transient calculations, all of the physical property values are required for the conduction calculation. In this case, the properties of copper (Cu), Nichrome (NiCr), and Alumina  $(Al_2O_3)$  are required.

Select the Init tab. Select subdomains 1–13. Enter T\_ref in the Initial value edit window. See Figure 6.75. Click OK.

# Physics Boundary Settings: General Heat Transfer (htgh)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 6.19. See Figures 6.76 and 6.77.

Boundary	Boundary Condition	Value/ Expression	Click Apply	Figure Number
1, 40	Temperature	T_air	Yes	6.76
2, 3, 5, 26, 28, 39	Thermal insulation	_	No	6.77

# Table 6.19 Boundary Settings–Heat Transfer by Conduction (ht) Edit Window

= T <sub>0</sub>				
Boundaries Groups	Boundary Cond	ition Highly Conducti	ve Layer	Element Color/Style ]
Boundary selection	Boundary sources and	d constraints		
1	Library coefficient:	•	Load	
2 3 4	Boundary condition:	Temperature	•	
5	Quantity	Value/Expression	Unit	Description
6 <b>U</b>	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
8	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
10	Tinf	273.15	к	External temperature
11 12	τ <sub>0</sub>	T_air	к	Temperature
13 14	Radiation type:	None	•	
15	٤	0	1	Surface emissivity
Group:	Tamb	0	к	Ambient temperature
Select by group	J <sub>0</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

**FIGURE 6.76** 2D Resistive\_Heating\_3 model Boundary Settings (1, 40) edit window

quation n · (-kVT) = 0				
$1 \cdot (-\mathbf{K}\mathbf{V}1) = 0$				
Boundaries Groups	Boundary Cond	ition Highly Conducti	ve Layer	Element Color/Style
Boundary selection	Boundary sources and	l constraints		
25	Library coefficient:	•	Load	
26	Boundary condition:			
28		Thermal insulation	-	
2.9		Value/Expression	Unit	Description
30	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
32	h	0	$W/(m^2 \cdot K)$	Heat transfer coefficient
33 34	Τ <sub>inf</sub>	273.15	к	External temperature
35	то	273.15	к	Temperature
36 37	Radiation type:	None	•	
38	ε		1	5 ST 15 ST
	•	0		Surface emissivity
Group:	Tamb	0	к	Ambient temperature
Select by group	Jo	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

I FIGURE 6.77 2D Resistive\_Heating\_3 model Boundary Settings (2, 3, 5, 26, 28, 39) edit window

## Physics Subdomain Settings: Conductive Media DC (emdc)

Using the menu bar, select Multiphysics > Conductive Media DC (dc). Using the menu bar, select Physics > Subdomain Settings. Select subdomains 1–13 in the Subdomain selection window (all of the subdomains). Enter 0.01 in the Thickness (d) edit window. Select "Linear temperature relation" from the Conductivity relation pull-down list. Click the Apply button.

Select Subdomains (3, 5, 7, 9, 11) > Load > Basic Materials Properties > Alumina. Select "Conductivity" from the Conductivity relation pull-down list. Enter 0.001 in the Electric conductivity edit window. Click the Apply button.

In the Subdomain edit windows, enter the information shown in Table 6.20. Click the Apply button. See Figure 6.78.

$\nabla \cdot d(\sigma \nabla V - \mathbf{J}^{e}) = dQ_{j}, \sigma = 1/(\rho)$	$_{0}(1 + \alpha(T - T_{0})))$			
Subdomains Groups	Physic	s Infinite Elements	Init Ele	ment Color
Subdomain selection	Material properties and	l sources		
1 2 (default)	l ibrary material		oad)	
3	Quantity	Value/Expression	Unit	Description
4 (default) 5	Je	0 0	A/m <sup>2</sup>	External current density
6 (default) 7	Qj	0	A/m <sup>3</sup>	Current source
δ (default)	d	0.01	m	Thickness
9 10 (default)	Conductivity relation:	Conductivity relation:		
11 12 (default)	Ρ <sub>0</sub>	r_Cu	Ω·m	Resistivity at reference temp
Group:	α	alpha_Cu	1/K	Temperature coefficient
Select by group	т	Т	к	Temperature
Active in this domain	Τ <sub>0</sub>	T_ref	к	Reference temperature

FIGURE 6.78 2D Resistive\_Heating\_3 model Subdomain Settings (1, 13) edit window

#### Table 6.20 Subdomain Settings–Conductive Media DC (emdc) Edit Window

Name	Expression	Description
$\rho_0$	r_Cu	Resistivity at reference temperature
α	alpha_Cu	Temperature coefficient
T <sub>0</sub>	T_ref	Reference temperature
1	ρ <sub>0</sub> α	ρ <sub>0</sub> r_Cu α alpha_Cu

Subdomain Number	Name	Expression	Description
2, 4, 6, 8, 10, 12	$\rho_0$	r_NiCr	Resistivity at reference temperature
	α	alpha_NiCr	Temperature coefficient
	T <sub>0</sub>	T_ref	Reference temperature

Table 6.21	Subdomain Settings–Conductive Media DC (emdc) Edit Window

In the Subdomain edit windows, enter the information shown in Table 6.21. Click the Apply button. See Figure 6.79.

At this point in the model, the generation of heat is coupled to the resistivity in each different material (Cu, NiCr) through the temperature change.

Select the Init tab. Select subdomains 1–13 in the Subdomain selection window (all of the subdomains). Enter  $V_0*(1 - x[1/m])$  in the  $V(t_0)$  edit window. See Figure 6.80. Click OK.

Subdomains Groups	Physics	s Infinite Elements	Init Eler	nent Color
Subdomain selection	Material properties and	sources		
1 2 (default)	Library material:	÷ (1	oad	
3	Quantity	Value/Expression	Unit	Description
4 (default) 5	Je	0 0	A/m <sup>2</sup>	External current density
6 (default) 7	Qj	0	A/m <sup>3</sup>	Current source
8 (default)	d	0.01	m	Thickness
9 10 (default)	Conductivity relation:	Linear temperature r	elation 💲	
11 12 (default)	ρ <sub>0</sub>	r_NiCr	Ω·m	Resistivity at reference temp
13 Troup: default	α	alpha_NiCr	1/K	Temperature coefficient
	т	Т	к	Temperature
Select by group	то	T_ref	к	Reference temperature

FIGURE 6.79 2D Resistive\_Heating\_3 model Subdomain Settings (2, 4, 6, 8, 10, 12) edit window

379

Subdomains Groups	Physics Infinite	Elements Init Element Color
2 (default) 4 (default) 5 (default) 5 (default) 7 3 (default) 10 (default) 11 12 (default) 13	V(t <sub>0</sub> ) V_0*(1-x[1/m])	V Electric potential
roup: 🔹 🛟		

FIGURE 6.80 2D Resistive\_Heating\_3 model Subdomain Settings Init edit window

NOTE The initial conditions assume a linear voltage drop across the body of the model. This is reflected in the initialization equation  $(V(t_0) = V_0^*(1 - x[1/m]))$ .

## Physics Boundary Settings: Conductive Media DC (emdc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, Select or enter the given boundary condition and value as shown in Table 6.22. Check the Interior boundaries check box. See Figures 6.81, 6.82, and 6.83.

Boundary	<b>Boundary Condition</b>	Value/Expression	Figure Number
1	Electric potential	V_0	6.81
2, 3, 5,	Electric insulation	_	6.82
26, 28, 39			
40	Ground	—	6.83

× ۷ <sub>0</sub>	
Boundaries Groups oundary selection	Conditions       Port       Color/Style         Boundary sources and constraints       Library material: <ul> <li>Load</li> <li>Boundary condition:</li> <li>Electric potential</li> <li>Quantity</li> <li>Value/Expression</li> <li>Unit</li> <li>Description</li> <li>V<sub>0</sub></li> <li>V_0</li> <li>V_0</li> <li>V Electric potential</li> </ul>

FIGURE 6.81 2D Resistive\_Heating\_3 model Boundary Settings (1) edit window

Click OK. See Figure 6.84.

#### **Mesh Generation**

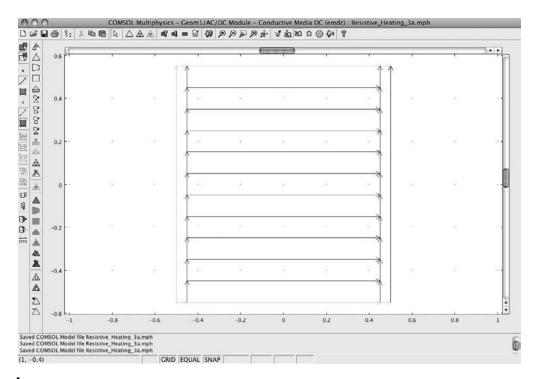
On the toolbar, click Mesh > Free Mesh Parameters. Select the Subdomain tab. Select subdomains 1-13. Enter 0.02 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 6.85.

Boundaries Groups	Conditions Port Color/Style
oundary selection	Boundary sources and constraints
n A	Library material:
14. 15	
16	Boundary condition: Electric insulation
17	
9	
10 1	
roup:	
Select by group	
Interior boundaries	

**I FIGURE 6.82** 2D Resistive\_Heating\_3 model Boundary Settings (2, 3, 5, 26, 28, 39) edit window

= 0	
Boundaries Groups Boundary selection 33 34 35 36 37 38 39 40 Croup: Select by group Interior boundaries	Conditions     Port     Color/Style       Boundary sources and constraints     Library material:     Load       Boundary condition:     Ground     Image: Color and a state of the state o

FIGURE 6.83 2D Resistive\_Heating\_3 model Boundary Settings (40) edit window



**FIGURE 6.84** 2D Resistive\_Heating\_3 model with all the boundary settings

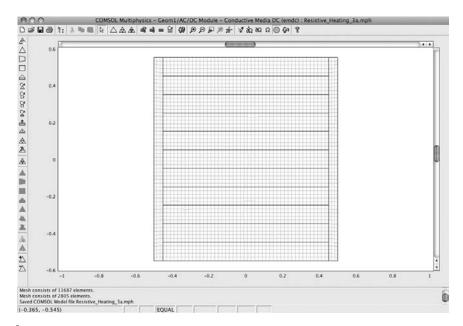
0	Free Mesh Parameters	_
Global	Subdomain Boundary Point Advanced	ОК
Subdomain selection 1 2 3 4 5 6 7 8 9 10 11 12 13 Select by group Select Remaining Select Meshed	Subdomain mesh parameters Maximum element size: 0.02 Element growth rate: Method: Quad 🗘	Canc Appl Help
	Remesh Mesh Selected	

FIGURE 6.85 2D Resistive\_Heating\_3 model Free Mesh Parameters window

Click the Remesh button. Click OK. See Figure 6.86.

#### Solving the 2D Resistive\_Heating\_3 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.



**FIGURE 6.86** 2D Resistive\_Heating\_3 model mesh

Analysis:	Gene	ral Time Stepping Advanced
Transient  Auto select solver olver: Stationary Time dependent Eigenvalue Parametric Stationary segregated		0:50:2000 0.01 0.0010
Adaptive mesh refinement	Preconditioner:	Direct (UMFPACK)
	Matrix symmetry:	Automatic
	(	Help Apply Cancel OK

**FIGURE 6.87** 2D Resistive\_Heating\_3 model Solver Parameters edit window

**NOTE** The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter 0:50:2000 in the Times edit window. See Figure 6.87. Click OK.

#### Time-Dependent Solving of the 2D Resistive\_Heating\_3 Model

Select Solve > Solve Problem. See Figure 6.88.

#### Postprocessing and Visualization

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot

**FIGURE 6.88** 2D Resistive\_Heating\_3 model solution

Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pull-down list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.89.

Click OK. See Figure 6.90.

Next, because this is a transient analysis model, the modeler can test how close the solution is to the steady-state value. Select Postprocessing > Cross-Section Plot Parameters > General > Point plot. Verify that all of the Solutions to use are selected. See Figure 6.91.

Click the Point tab. Enter x = 0, y = 0.1 in the Coordinates edit windows. See Figure 6.92.

Click OK. Figure 6.93 shows the temperature versus time plot for the point x = 0, y = 0.1. It is easily seen that the temperature is somewhat close to the steady-state value (the curve is still rising, at a decreasing  $\Delta T$ ) at the end of the modeling calculation.

It is interesting to see how the heat flux moves in this array. Select Postprocessing > Plot Parameters > Arrow. Check the Arrow plot check box. Select "Total heat flux."

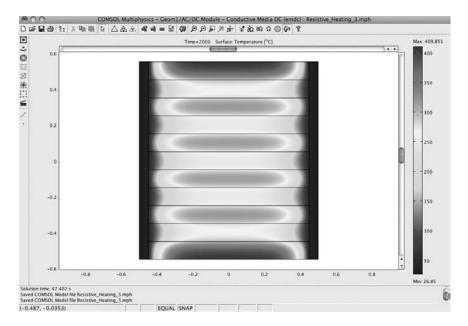
	rface Contour Boundary Arrow P	rincipal
Surface plot		
	Surface Data Height Data	
Predefined quanti	tities: Temperature Range	
Expression:	T Smooth	
Unit:	°c \$	
Coloring and fill		
Coloring:	nterpolated 🗘 Fill style: Filled 🗘	)
Surface color		
Colormap:	jet Colors: 1024 🗹 Color scale	2
O Uniform color	r: Color	

**FIGURE 6.89** 2D Resistive\_Heating\_3 model Plot Parameters edit window

Click the Color button and select a color (black). Click OK, and then click OK again. See Figure 6.94.

## **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.95.



**FIGURE 6.90** 2D Resistive\_Heating\_3 model, degrees Centigrade

Plot type O Line/Extrusion plot  O Point plot	
Solutions to use	Solution at angle (phase):
Select via: Stored output times 💠	0 degrees
0	Frame:
50	( )
100	
150	
200	
250	
300	
350 🤤	
400	
450	
Times:	
lot in: New figure 🛟 📋 Keep curr	rent plot
Display cross-section in main axes	Color
Title/Axis	

I FIGURE 6.91 2D Resistive\_Heating\_3 model, Cross-Section Plot Parameters, General edit window

Point plot /-axis data	
Predefined quantities:	Temperature
Expression:	Т
Jnit	К
Coordinates	
s; 0	
r: 0.1	
Auto     Expression.	

**FIGURE 6.92** 2D Resistive\_Heating\_3 model, Cross-Section Plot Parameters, Point edit window

**FIGURE 6.94** 2D Resistive\_Heating\_3 model, heat flux (proportional arrows)

Click the Start Animation button. See Figure 6.96.

Alternatively, you can play the file Movie6\_RH\_3.avi that was supplied with this book.

#### 2D Resistive Heating Models: Summary and Conclusions

The models presented in this section of Chapter 6 have introduced the following new concepts: Ohm's law, Joule heating, mixed-mode modeling, mixed-materials modeling, transient analysis, and the good first approximation. Previously introduced concepts employed include the triangular mesh, free mesh parameters, subdomain mesh, maximum element size, and quadrilateral mesh (quad).

The three resistive heating models are more illustrative of the mixed-mode modeling concept than they are directly amenable to the comparison of calculated values. They present different examples of the diversity of applied scientific and engineering model designs that can be explored using electro-thermal coupling and transient analysis. These models also demonstrate the significant power of relatively simple physical principles, such as Ohm's law and Joule's law.

	mline Particle Tr	-		Anima
Movie settings		Solutions to us	e	
File type:	AVI 🛟	Select via:	stored output tim	nes 🛟
Width (in pixels):	640	0		Ō
Height (in pixels):	480	50		U
		100 150		
Frames per second:	10	200		
	Advanced	250 300		
Static / Eigenfunction a	inimation	350		
	Full harmonic 🛟	400 450		Ŧ
		Times:		
Number of frames: 1	1			
Number of frames: 1 Reverse direction Use camera setting:	1 s from main window			
Reverse direction	1			
Reverse direction	1			
Reverse direction	1			
Reverse direction	1			
Reverse direction	1			
Reverse direction	1			
Reverse direction	1		(Start A	nimation

**FIGURE 6.95** 2D Resistive\_Heating\_3 model, Plot Parameters window

# 2D Inductive Heating Considerations

## 2D Axisymmetric Coordinate System

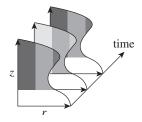
In this part of Chapter 6, the concepts of interest are most easily explored using the 2D axisymmetric coordinate system. Reviewing briefly the 2D axisymmetric coordinate system basics, parameters in steady-state solutions to any 2D axisymmetric model can vary only as a function of the radial position in space (r) and the axial position in space (z) coordinates. Such models represent the parametric condition of the model in a time-independent mode (quasi-static). In transient solution models, as

FIGURE 6.96 2D Resistive\_Heating\_3 model animation, final frame

presented in this section, parameters can vary both by position in space (r) and space (z) and by time (t); see Figure 6.97.

The transient solution model is essentially a sequential collection of steady-state (quasi-static) solutions, except that the condition has been added that at least one of the dependent variables changes as a function of time.

In transient or time-dependent (e.g., dynamic, unsteady) models, at least one of the dependent variables changes as a function of time. The change in the dependent variable is a direct result of the coupling inherent in the physical properties of the materials involved in the model.



**FIGURE 6.97** 2D axisymmetric coordinate system, plus time

For example, the resistance of a material typically changes  $(\pm)$  as a function of temperature. When heat is generated in a material through current flow (Joule's law), the temperature of the material changes. Hence, as the temperature changes over time, the resistance changes.

The space coordinates (r) and (z) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values  $(r_{\min} \le r \le r_{\max})$  and  $(z_{\min} \le z \le z_{\max})$ . The time coordinate (t) represents the range of values  $(t_{\min} \le t \le t_{\max})$  from the beginning of the observation period  $(t_{\min})$  to the end of the observation period  $(t_{\max})$ .

Joule heating techniques are extremely important in device design considerations. Joule heating is applied to tasks as varied as heating houses (AC) and baking potatoes (microwave AC). It accounts for some of the most widely utilized technologies employed for research, design, and application in engineering and physics. Most modern products or processes require an understanding of Joule heating techniques either during development or during the use of the product or process (e.g., automobiles, plate glass fabrication, plastic extrusion, plastic products, houses, baked potatoes, ice cream).

NOTE Heating and heat transfer concerns have existed since the beginning of prehistory. There have been many contributors to the present understanding of the interaction of electric currents and solids. Three scientists especially stand out in this regard: Georg Ohm, James Prescott Joule, and Leon Foucault.

Ohm published his discovery of Ohm's law in 1827:<sup>14</sup>

$$I = \frac{V}{R} \tag{6.5}$$

where

*I* = current in amperes (A) *V* = voltage (electromotive force) in volts (V)

R = resistance in ohms

Joule discovered Joule's law in 1843:<sup>15,16</sup>

$$Q = I^2 \cdot R \cdot t \tag{6.6}$$

where

Q = heat generated in joules (J)

I =current in amperes (A)

R = resistance in ohms

t = time in seconds (S)

In 1851, Foucault<sup>17</sup> discovered eddy currents<sup>18</sup> (also called Foucault currents). Eddy currents result when a conductor is in the presence of a changing magnetic field.

$$(j\omega\sigma - \omega^{2}\varepsilon)A + \nabla \times (\mu^{-1}\nabla \times A) = J^{e}$$
(6.7)

where A = magnetic vector potential

 $\omega$  = angular frequency

 $\sigma =$ conductivity

 $\epsilon$  = permittivity

$$\mu$$
 = permeability

 $J^{e}$  = current density due to an external source<sup>19</sup>

The induced eddy currents interact with the resistance of the conductor (Ohm's law) through Joule's law, causing heat. The net effect of these interactions is induction heating.<sup>20</sup> In this model, the equations are as follows:

$$j\omega\sigma(T)A + \nabla \times (\mu^{-1}\nabla \times A) = 0$$
(6.8)

where A = magnetic vector potential  $\omega =$  angular frequency  $\sigma(T) =$  conductivity  $\mu =$  permeability

and

$$\rho C_{\rm p} \frac{\partial T}{\partial t} - \nabla \cdot k \nabla T = Q(T, A)$$
(6.9)

where  $\rho = \text{density}$   $C_p = \text{specific heat capacity}$  T = temperature t = time k = thermal conductivityA = magnetic vector potential

and

$$\sigma(T) = (\rho_{\rm ref}(1 + \alpha(T - T_{\rm ref}))^{-1}$$
(6.10)

where  $\sigma(T)$  = electrical conductivity  $\rho_{ref}$  = resistivity at the reference temperature T = temperature  $T_{ref}$  = reference temperature  $\alpha$  = thermal coefficient of the resistivity and

$$Q(T) = \frac{1}{2} \sigma(T) |E_{\rm p}|^2$$
(6.11)

where

Q(T) = heat generated per period for a sinusoidal wave function  $\sigma(T)$  = conductivity at the present temperature T = temperature  $E_{\rm p}$  = electric field, peak value

The first example presented in this section, the Inductive\_Heating\_1 model, explores 2D axisymmetric electro-thermal interaction modeling of Joule heating using transient analysis. The model is solved for a material that is both electrically and thermally conductive. The model is implemented using the COMSOL AC/DC Module Electro-Thermal Application Mode. This model demonstrates the principle of induction heating.

In the first variation on the inductive heating model, the new model is built to explore a common configurational change and is solved using the same COMSOL Multiphysics AC/DC Module Application Mode. In the second variation on this model, a model is built that incorporates materials modifications in addition to the configurational changes and is again solved using the COMSOL Multiphysics AC/DC Electro-Thermal Application Mode. The second variation also explores the influence of an insulating environment on the model's properties. The calculated modeling results are then compared.

## 2D Axisymmetric Inductive Heating Model

The following numerical solution model (Inductive\_Heating\_1) is derived from a model that was originally developed by COMSOL as a Multiphysics demonstration model for distribution with the Multiphysics software in the basic Multiphysics Model Library. This model continues the introduction of the coupling of two important basic physical materials properties: Joule heating and heat transfer. The coupling of these two properties in this model demonstrates one of the interactions normally found in typical engineering materials.

The new modeler should personally build each model. There is no better method to obtain a rapid understanding of the modeling process than to employ the process of gaining the hands-on experience of actually building, meshing, solving, and postprocessing a model. Many times the inexperienced modeler will make and correct errors, thereby adding to his or her experience and fund of modeling knowledge. Even building the simplest model will expand the modeler's store of knowledge.

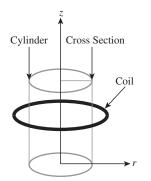


FIGURE 6.98 3D rendition of the 2D axisymmetric Inductive\_Heating\_1 model

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either adds heat or removes heat so as to achieve or maintain a desired temperature. Figure 6.98 shows a 3D rendition of the 2D inductive heating geometry, as will be modeled here.

To start building the Inductive\_Heating\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select AC/DC Module > Electro-Thermal Interaction > Azimuthal Induction Heating > Transient analysis. See Figure 6.99. Click OK.



FIGURE 6.99 2D axisymmetric Inductive\_Heating\_1 Model Navigator setup

#### Table 6.23 Axis Edit Window

Name	Value
r min	-0.05
r max	0.5
z min	-0.3
z max	0.3

#### Table 6.24 Grid Edit Windows

Name	Value
r spacing	0.05
Extra r	0.03
z spacing	0.05
Extra z	-0.01 0.01

**NOTE** There are two mode names in the Application mode name window (htgh emqa). The names in the window show which applications are coupled in this application: General Heat Transfer (htgh) and Azimuthal Induction Currents, Vector Potential (emqa).

### **Options and Settings**

Using the menu bar, select Options > Axes/Grid Settings. In the Axis edit windows, enter the information shown in Table 6.23; see Figure 6.100.

Select the Grid tab. Uncheck the Auto check box. In the Grid edit windows, enter the information shown in Table 6.24; See Figure 6.101. Click OK.

Axis	s equal		
r z limi	ts	phi limits	1
r min:	-0.05	Auto	
r max:	0.5	phi min:	-1
z min:	-0.3	phi max:	1
z max:	0.3		

FIGURE 6.100 2D axisymmetric Inductive\_Heating\_1 model Axis edit window

Auto	Visible 🗹 Labels			
r-z grid		phi grid		
r spacing:	0.05	Auto 🗹		
Extra r:	0.03	phi spacing:	0.2	
z spacing:	0.05	Extra phi:		
Extra z:	-0.01 0.01			

**FIGURE 6.101** 2D axisymmetric Inductive\_Heating\_1 model Grid edit window

Note The "extra" r and z grid points are defined points on the grid that are used to aid the modeler in creating the designs needed to build this model. The model can also be built on a standard grid by using the Specify Object commands.

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.25; also see Figure 6.102. Click OK.

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.01. Select "Base: Center" and set r equal to 0.05 and z equal to 0 in the Circle edit window. See Figure 6.103. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.2 and a height of 0.5. Select "Base: Corner" and set r equal to 0 and z equal to -0.25 in the Rectangle edit window. See Figure 6.104. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.03 and a height of 0.1. Select "Base: Corner" and set r equal to 0 and z equal to -0.05 in the Rectangle edit window. See Figure 6.105.

Name	Expression	Description
I_0	1e3[A]	Coil current
d_0	2e-2[m]	Coil diameter
c_0	pi*d_0	Coil circumference
Js_0	I_0/c_0	Coil surface current density
T_ref	20[degC]	Reference temperature
r_Cu	1.67e-8[ohm*m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	Resistivity coefficient copper
rho_air	1.293[kg/m^3]	Density air STP
Cp_air	1.01e3[J/(kg*K)]	Heat capacity air
k_air	2.6e-2[W/(m*K)]	Thermal conductivity air
rho_Cu	8.96e3[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity copper

### Table 6.25Constants Edit Window

Name	Expression	Value	Description
1_0	1e3[A]	1000[A]	Coll current
d_0	2e-2[m]	0.02[m]	Coil diameter
r_0	pi*d_0	0.062832[m]	Coil circumference
Js_0	I_0/c_0	15915.494309[A/m]	Coil surface current density
I_ref	20[degC]	293.15[K]	Reference temperature
r Cu	1.67e-8[ohm*m]	(1.67e-8)[Ω·m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	0.0039[1/K]	Resistivity coefficient copper
rho_air	1.293[kg/m^3]	1.293[kg/m <sup>3</sup> ]	Density air STP
Cp_air	1.01e3[J/(kg*K)]	1010[J/(kg·K)]	Heat capacity air
k air	2.6e-2[W/(m*K)]	0.026[W/(m·K)]	Thermal conductivity air
rho_Cu	8.96e3[kg/m^3]	8960[kg/m <sup>3</sup> ]	Density copper
Cµ_Cu	3.8e2[J/(kg*K)]	380[J/(kg·K)]	Heat capacity copper
k Cu	3.94e2[W/(m*K)]	394[W/(m·K)]	Thermal conductivity copper

**FIGURE 6.102** 2D axisymmetric Inductive\_Heating\_1 model Constants edit window

Size		Rotat	ion angle	
Radius:	0.01	α:	0	(degrees
Position				
Base:	Center	\$ Style:	Solid	<b>.</b>
r:	0.05	Name:	C1	
z	0			

**FIGURE 6.103** 2D axisymmetric Inductive\_Heating\_1 model Circle edit window

) 0 0		Rectangle
Size		Rotation angle
Width:	0.2	α: 0 (degrees
Height:	0.5	
Position		
Base:	Corner	\$ Style: Solid
res	0	Name: R1
	-0.25	

FIGURE 6.104 2D axisymmetric Inductive\_Heating\_1 model Rectangle edit window

Click OK, and then click the Zoom Extents button. See Figure 6.106.

**NOTE** The rectangle (R2) is the 2D representation of a cylinder in cross section. The circle is the cross-section profile of the current loop (coil), as was shown earlier in Figure 6.98.

#### **Physics Settings**

Select Physics > Scalar Variables. Enter 500 in the nu\_emqa edit window. See Figure 6.107. Click OK.

Select Options > Expressions > Scalar Expressions. Enter the scalar expression for sigma\_T as shown in Table 6.26; also see Figure 6.108. Click OK.

Size		Rotat	ion angle	
Width:	0.03	α:	0	(degrees)
Height:	0.1			
Position				
Base:	Corner	\$ Style:	Solid	¢
r:	0	Name:	R2	
z:	-0.05			

**FIGURE 6.105** 2D axisymmetric Inductive\_Heating\_1 model Rectangle edit window

**FIGURE 6.106** 2D axisymmetric Inductive\_Heating\_1 model

5.67e-8         W/(m <sup>2</sup> ·K <sup>4</sup> )         Stefan-Boltzmann constant           8.31451         J/(mol·K)         Universal gas constant           8.854187817e-12         F/m         Permittivity of vacuum           4 <sup>+</sup> pi*1e-7         H/m         Permeability of vacuum           500         Hz         Frequency	Name	Expression	Unit	Description
8.854187817e-12 F/m Permittivity of vacuum 4*pi*1e-7 H/m Permeability of vacuum	igma_htgh	5.67e-8	$W/(m^2 \cdot K^4)$	Stefan-Boltzmann constant
4*pi*1e-7 H/m Permeability of vacuum		Universal gas constant		
	epsilon0_emqa	8.854187817e-12	F/m	Permittivity of vacuum
500 Hz Frequency	nu0_emqa	4*pi*1e-7	H/m	Permeability of vacuum
	nu_emqa	500	Hz Frequency	
	nu_emqa	500	Hz	Frequency

**FIGURE 6.107** 2D axisymmetric Inductive\_Heating\_1 model Application Scalar Variables edit window

## Table 6.26 Scalar Expressions Edit Window

Name	Expression	Description
sigma_T	1/(r_Cu*(1 + alpha_Cu*(T-T_ref)))	Electrical conductivity copper

Scalar Expressions			
Name	Expression	Unit	Description
sigma_T	1/(r_Cu*(1+alpha_Cu*(T-T_ref)))	S/m	Electrical conductivity copper
c 🛛	Help		pply Cancel OK

FIGURE 6.108 2D axisymmetric Inductive\_Heating\_1 model Scalar Expressions edit window

NOTE The scalar expression for sigma\_T couples the resistivity of copper  $(r_Cu)$  and the temperature (T).

## Physics Subdomain Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Multiphysics > Azimuthal Induction Currents, Vector Potential (emqa). Using the menu bar, select Physics > Subdomain Settings > Electric Parameters. In the Subdomain edit windows, enter the information shown in Table 6.27; also see Figure 6.109. Click OK.

Subdomains Groups	Magnetic Paran	neters Electric Parameters	Infinite Elements Forces
subdomain selection	-Electric material Library materia Constitutive re $\textcircled{O} D = \varepsilon_0 \varepsilon_r E$ Quantity $V_{loop}$ $\int_{\phi}^{\varepsilon} \phi$	lation	ad $D = \varepsilon_0 \varepsilon_r E + D_r$ Unit Description V Loop potential $A/m^2$ External current density
	σ	sigma_1	S/m Electric conductivity
iroup: 🔹 🗘	٤٢	1	1 Relative permittivity

FIGURE 6.109 2D axisymmetric Inductive\_Heating\_1 model Subdomain Settings (2), Electric Parameters edit window

Subdomain	Name	Expression	Description
1	σ	0	Electric conductivity
2	σ	sigma_T	Electric conductivity
3	σ	0	Electric conductivity

#### Table 6.27 Subdomain Edit Window

#### Table 6.28 Boundary Settings Edit Window

Boundary	Boundary Condition	Figure Number
1, 3, 5	Axial symmetry	6.110
2, 7, 9	Magnetic insulation	6.111

## Physics Boundary Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.28. Check the Interior boundaries check box. See Figures 6.110 and 6.111.



**FIGURE 6.110** 2D axisymmetric Inductive\_Heating\_1 model Boundary Settings (1, 3, 5) edit window

Boundaries Groups	Conditions Material Properties Color/Style
Roundary selection	Boundary sources and constraints Boundary condition: Magnetic insulation

**FIGURE 6.111** 2D axisymmetric Inductive\_Heating\_1 model Boundary Settings (2, 7, 9) edit window

In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.29. Click OK. See Figure 6.112.

Table 6.29 Boundary Settings Edit Window

Boundary	Boundary Condition	Value	Figure Number
10–13	Surface current	Js_0	6.112

Subdomain	Name	Expression	Description	Figure Number
1	k (isotropic)	k_air	Thermal conductivity	6.113
	ρ	rho_air	Density	
	Cp	Cp_air	Heat capacity	
2, 3	k (isotropic)	k_Cu	Thermal conductivity	6.114
	ρ	rho_Cu	Density	
	$C_{\rm p}$	Cp_Cu	Heat capacity	

#### Table 6.30Subdomain Edit Window

## Physics Subdomain Settings: General Heat Transfer (htgh)

Using the menu bar, select Multiphysics > General Heat Transfer (htgh). Using the menu bar, select Physics > Subdomain Settings > Conduction. In the Subdomain edit windows, enter the information shown in Table 6.30. See Figures 6.113 and 6.114.

Subdomains Groups	Conducti	on Convection Id	eal Gas Init Element Color
Subdomain selection  (default)  (default)	Thermal properties a Library material: Quantity $\delta_{ts}$ o k (isotropic) ${\rho}$ k (anisotropic) ${\rho}$ Q Q Opacity:	Value/Expression  K_air  400 0 0 400  rho_air  Cp_air  Qav_emqa  Opaque	Load       Description         1       Time-scaling coefficient         W/(m·K)       Thermal conductivity         W/(m·K)       Thermal conductivity         W/(m·K)       Thermal conductivity         J/(kg·K)       Density         J/(kg·K)       Heat capacity at constant pressure         W/m³       Heat source
Group: 🔹 🗘			

FIGURE 6.113 2D axisymmetric Inductive\_Heating\_1 model Subdomain Settings (1) edit window

Subdomains Groups	Conductio	on Convection Id	eal Gas Init Element Color
Subdomain selection	<ul> <li>Thermal properties at Library material:</li> <li>Quantity         δts         <ul> <li></li></ul></li></ul>	Value/Expression           I         k_Cu         400 0 0 400         rho_Lu         Cp_Cu         Qav_emqa         Opaque	Load Unit Description Time-scaling coefficient W/(m·K) Thermal conductivity W/(m·K) Thermal conductivity kg/m <sup>3</sup> Density J/(kg·K) Heat capacity at constant pressure W/m <sup>3</sup> Heat source
Group: default 🛟			

FIGURE 6.114 2D axisymmetric Inductive\_Heating\_1 model Subdomain Settings (2, 3) edit window

Select the Init tab. Select all subdomains (1–3). Enter T\_ref in the  $T(t_0)$  edit window. See Figure 6.115. Click OK.

### Physics Boundary Settings: General Heat Transfer (htgh)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – General Heat Transfer (htgh) edit windows, enter the information shown in Table 6.31. Click OK. See Figures 6.116 and 6.117.

Boundary	Boundary Condition	Value	Figure Number
1, 3, 5	Axial symmetry	—	6.116
2, 7, 9	Temperature	T_ref	6.117

quation	Subdomain Settings – General Heat Transfer (htgh)		
$pC_{D}\partial T/\partial t + \nabla \cdot (-k\nabla T) = Q$			
= temperature			
Subdomains Groups	Conduction Convection Ideal Gas Init Element Color		
Subdomain selection	- Initial value		
2 (default) 3 (default)	T(t <sub>0</sub> ) [ <u>r_ref</u> K Temperature		
Group: 🚺 🛟			

FIGURE 6.115 2D axisymmetric Inductive\_Heating\_1 model Subdomain Settings (1–3), Init edit window

## **Mesh Generation**

On the toolbar, click the Initialize Mesh button once. This results in a mesh of approximately 1200 elements. See Figure 6.118.

## Solving the 2D Axisymmetric Inductive\_Heating\_1 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

**NOTE** The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter linspace(0,1200,21) in the Times edit window. See Figure 6.119. Click the Advanced tab. Check the Use complex functions with real input check box. See Figure 6.120. Click OK.

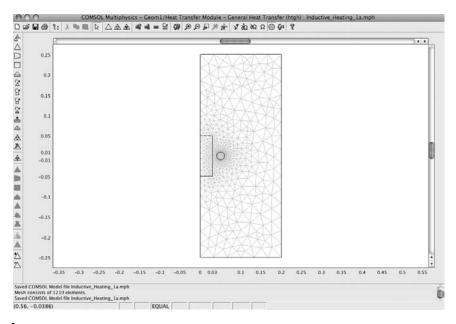


FIGURE 6.118 2D axisymmetric Inductive\_Heating\_1 model mesh window

Analysis:	Ge	neral Time St	epping Adv	anced
Transient	Time stepping			),1200,21)
Stationary Time dependent Eigenvalue Parametric	Relative tolerance: Absolute tolerance: Allow complex numbers		0.01	
Stationary segregated Parametric segregated Adaptive mesh refinement	Linear system solver Linear system solver Preconditioner	Direct (UMFPA	СК)	Settings
	Matrix symmetry:	Automatic		
		(Help)	(Apply)	Cancel OK

**FIGURE 6.119** 2D axisymmetric Inductive\_Heating\_1 model Solver Parameters edit window

000	Solver Parameters		
Analysis:		me Stepping Advanced	
Transient 🛟	Constraint handling method:	Elimination	\$
Solver:	Null-space function:	Automatic	\$
Stationary Time dependent	Assembly block size:	5000	
igenvalue	Use Hermitian transpose of co	nstraint matrix and in symmetry d	etection
Parametric Stationary segregated	Use complex functions with rea	al input	
Parametric segregated	Stop if error due to undefined	operation	
	Store solution on file		
	Solution form:	Automatic	\$
Adaptive mesh refinement	Scaling of variables		
	Type of scaling:	Automatic	\$
	Manual scaling:		
	Row equilibration:	On	=
	Manual control of reassembly	·	
	✓ Load constant	🗹 Jacobian constant	
	Constraint constant	🗹 Constraint Jacobian consta	int
	Damping (mass) constant	Mass constant	

FIGURE 6.120 2D axisymmetric Inductive\_Heating\_1 model Solver Parameters, Advanced edit window

### Time-Dependent Solving of the 2D Inductive\_Heating\_1 Model

Select Solve > Solve Problem. See Figure 6.121.

#### **Postprocessing and Visualization**

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pull-down list shows "Temperature." Select "degC" or " $^{\circ}$ C" from the Unit pull-down list. See Figure 6.122.

Click OK. See Figure 6.123.

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.124.

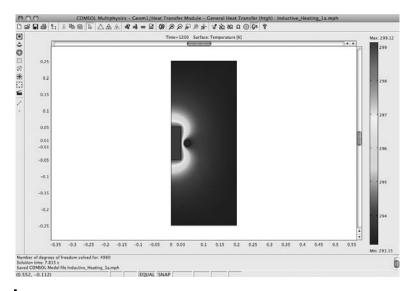


FIGURE 6.121 2D axisymmetric Inductive\_Heating\_1 model solution

	Surface	Contour	Boundary	A.	row	Principal	
General	Surface	Contour	Boundary	AI	row	Principai	
Surface	e plot						
		Surface Data	Height Data				
Predefine	d quantities:	Temperature	-	10	Range	2	
Expressi		Т			Smooth		
	on,				smooth		
Unit		°c					
Coloring a						_	
Coloring:	Interpo	lated 🗘	Fill style:	Filled		\$	
Surface co	lor						
Color	map:	jet 🗘	Colors: 1024		Color se	ale	
O Unifo	rm color:	Color		-			
O onito	in color.	Color	10 M				

**FIGURE 6.122** 2D axisymmetric Inductive\_Heating\_1 model Plot Parameters edit window

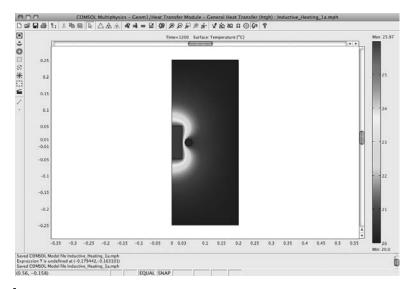
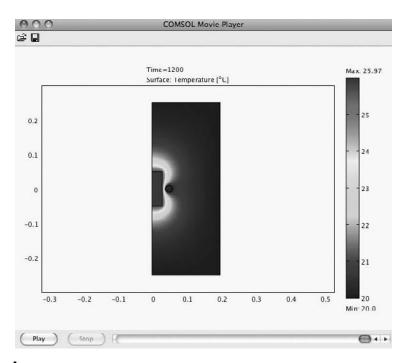


FIGURE 6.123 2D axisymmetric Inductive\_Heating\_1 model, degrees Centigrade

Principal St	reamline Particle Tr	acing Max/M	in Deform	Anima
Movie settings	14	Solutions to us	se	
File type:	AVI	Select via	Stored output ti	mes 🛟
Width (in pixels)	640	0		m
Height (in pixels):	480	60 120		
Frames per second:	10	180 240		Ψ
	( Advanced )	300 360		
Static / Ligenfunctio	n animation	420		4
	r 1000	540		*
Cycle type:	Full harmonic	1540		
Number of frames:		Times:		
Number of frames:	n l	and the second sec		

I FIGURE 6.124 2D axisymmetric Inductive\_Heating\_1 model Plot Parameters edit window



**FIGURE 6.125** 2D axisymmetric Inductive\_Heating\_1 model animation, final frame

Click the Start Animation button. See Figure 6.125.

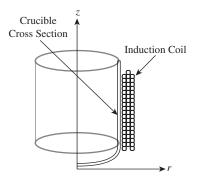
Alternatively, you can play the file Movie6\_IH\_1.avi that was supplied with this book.

## First Variation on the 2D Axisymmetric Inductive Heating Model

The following numerical solution model (Inductive\_Heating\_2) is derived from the Inductive\_Heating\_1 model. This variation continues the introduction of the coupling of two important basic physical materials properties—Joule heating and heat transfer—and expands on these concepts. The coupling of these two properties in this model demonstrates one of the applications normally found in typical engineering or process research.

Note This model is similar to the Inductive\_Heating\_1 model in the use of the induction heating method. However, in this case, the modeler will build an inductively heated crucible.

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat generation during a process and either adds heat or removes heat so as to achieve or



**FIGURE 6.126** 3D rendition of the 2D axisymmetric Inductive\_Heating\_2 model

maintain a desired temperature. Figure 6.126 shows a 3D rendition of the 2D axisymmetric Inductive\_Heating\_2 model geometry, as will be modeled here.

To start building the Inductive\_Heating\_2 Model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select AC/DC Module > Electro-Thermal Interaction > Azimuthal Induction Heating > Transient analysis. See Figure 6.127. Click OK.

0.0	Model Navigat	tor
New	Model Library User Mode	ls Open Settings
▼ 📄 Electro-Thern ▼ 📄 Azimuthal ♪ Steady ┣ Trans	Electric Magnetic Electromagnetic nal Interaction Induction Heating r-state analysis ent analysis I Induction Heating	AC/DC Module Frequencies of the second seco
Dependent variables: Application mode name:	T J Aphidr htgh emqa	Transient analysis.
Element:	Lagrange - Aphidr <sub>2</sub>	Multiphysics

**FIGURE 6.127** 2D axisymmetric Inductive\_Heating\_2 Model Navigator setup

NOTE There are two mode names in the Application mode name window (htgh emqa). The names in the window show which applications are coupled in this application: General Heat Transfer (htgh) and Azimuthal Induction Currents, Vector Potential (emqa).

#### **Options and Settings**

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.32; also see Figure 6.128. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.2 and a height of 0.5. Select "Base: Corner" and set r equal to 0 and z equal to -0.1 in the Rectangle edit window. Click OK.

Name	Expression	Description
I_0	2.5e2[A]	Coil current
d_0	1e-2[m]	Coil diameter
c_0	pi*d_0	Coil circumference
Js_0	I_0/c_0	Coil surface current density
T_ref	20[degC]	Reference temperature
r_Cu	1.67e-8[ohm*m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	Resistivity coefficient copper
rho_air	1.293[kg/m^3]	Density air STP
Cp_air	1.01e3[J/(kg*K)]	Heat capacity air
k_air	2.6e-2[W/(m*K)]	Thermal conductivity air
rho_Cu	8.96e3[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity copper

#### Table 6.32 Constants Edit Window

Name	Expression	Value	Description		
I_0	2.5e2[A]	250[A]	Coil current		
d_0	1e-2[m]	0.01[m]	Coil diameter		
c_0	pi*d_0	0.031416[m]	Coil circumference		
Js_0	I_0/c_0	7957.747155[A/m]	Coil surface current density		
T_ref	20[degC]	293.15[K]	Reference temperature		
r_Cu	1.67e-8[ohm*m]	(1.67e-8)[Ω·m]	Resistivity copper		
alpha_Cu	3.9e-3[1/K]	0.0039[1/K]	Resistivity coefficient copper		
rho_air	1.293[kg/m^3]	1.293[kg/m <sup>3</sup> ]	Density air STP		
Cp_air	1.01e3[J/(kg*K)]	1010[J/(kg·K)]	Heat capacity air		
k_air	2.6e-2[W/(m*K)]	0.026[W/(m·K)]	Thermal conductivity air		
rho_Cu	8.96e3[kg/m^3]	8960[kg/m <sup>3</sup> ]	Density copper		
Cp_Cu	3.8e2[J/(kg*K)]	380[J/(kg·K)]	Heat capacity copper		
k_Cu	3.94e2[W/(m*K)]	394[W/(m·K)]	Thermal conductivity copper		
$\leftarrow$			) • •		
¢					

**FIGURE 6.128** 2D axisymmetric Inductive\_Heating\_2 model Constants edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1 and a height of 0.25. Select "Base: Corner" and set r equal to 0 and z equal to 0.05 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes of 0.1 and B-semiaxes of 0.05. Select "Base: Center" and set r equal to 0 and z equal to 0.05 in the Ellipse edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1 and a height of 0.30. Select "Base: Corner" and set r equal to -0.1 and z equal to 0 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter E1–R3 in the Set formula edit window. See Figure 6.129. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter R2+CO1 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 6.130.

Click OK. See Figure 6.131.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter width of 0.1-0.005 and a height of 0.25. Select "Base: Corner" and set r equal to 0 and z equal to 0.05 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes of 0.1-0.005 and B-semiaxes of 0.05-0.005. Select "Base: Corner" and set r equal to 0 and z equal to 0.05 in the Ellipse edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1 and a height of 0.30. Select "Base: Corner" and set r equal to -0.1 and z equal to 0 in the Rectangle edit window. Click OK.

	14	٤:	关际的			33 =1	! = ≅	())	ø,	(C)	99	#*	\$ 80	20 06	0 (P	8	35						
		15							_			-	-			-							34.5
124	0.4									-									1.01				
4	0.4																						
/	0.35																						
7	0.3	- 3								1			1										
Ⅲ																							
民	0.25																						
E																							
0	0.2			13	120			(3)				5 C		*			35	28	100	100	100	<b>*</b>	18
8																							
開	0.15	- 11		3	2.60	100	1	(*)				4		1			199	30	1981	18. 1		1	1
0																							
_	0.1	1						(+)			17	-		*				1.4		÷	+	+	*
										co		RZ		Ĩ.									
E	0.05	[ ]										N.	1	+									+
	0	1	92	21	643	187	1127	12				_	110				91	2011	20	110	125	12	15
	-0.05	. 1	1		2	. 7	1.5	đ					1	10				10	01	10	ħ	ts	10
	-0.1			10		(2)	12	(3)	1=	-			R1		-		31 I.	28	35	100	100	大	18
		-0.3	5 -0.3	-0.25	-0.2	-0.15	-0.1	-0.0		0	0.	05	0.1	0.15	0.2	1	0.25	0.3	0.35	0.4	0.45	0.5	0.55
			file Induction label 'R3'.	/e_Heati	ing_2.mph																		

FIGURE 6.129 2D axisymmetric Inductive\_Heating\_2 model Create Composite Object edit window

Using the menu bar, select Draw > Create Composite Object. Enter E1–R3 in the Set formula edit window. See Figure 6.132. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter R2+CO2 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 6.133.

000	Create Composite Object
Object type Solids Curves Points	Select All Help
Object selection:	Set formula:
R1	R2+CO2
R2 CO2	Keep interior boundaries  Repair Repair tolerance: 1.0E-4

I FIGURE 6.130 2D axisymmetric Inductive\_Heating\_2 model Create Composite Object edit window

**FIGURE 6.131** 2D axisymmetric Inductive\_Heating\_2 model (CO2)

Click OK. See Figure 6.134.

Using the menu bar, select Draw > Create Composite Object. Enter: CO1-CO3 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 6.135.

000 c	reate Composite Object
Object type Solids Curves Points	Select All Help
Object selection:	Set formula:
R1	E1-R3
CO1 R2 E1 R3	Keep interior boundaries         Repair         Repair tolerance:         1.0E-4

FIGURE 6.132 2D axisymmetric Inductive\_Heating\_2 model Create Composite Object edit window

FIGURE 6.133 2D axisymmetric Inductive\_Heating\_2 model Create Composite Object edit window

**FIGURE 6.134** 2D axisymmetric Inductive\_Heating\_2 model (C01, C03)

000	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
R1	C01-C03
CO1 CO3	Keep interior boundaries  Repair  Repair tolerance: 1.0E-4

I FIGURE 6.135 2D axisymmetric Inductive\_Heating\_2 model Create Composite Object edit window

Click OK. See Figure 6.136.

Having built the crucible, the next step is to build the first layer of the heating coil. Using the menu bar, select Draw > Specify Objects > Circle. Create each of the circles shown in Table 6.33.

Name	Radius	Base	r	z
C1	0.005	Center	0.11	0.260
C2	0.005	Center	0.11	0.248
C3	0.005	Center	0.11	0.236
C4	0.005	Center	0.11	0.224
C5	0.005	Center	0.11	0.212
C6	0.005	Center	0.11	0.200
C7	0.005	Center	0.11	0.188
C8	0.005	Center	0.11	0.176
C9	0.005	Center	0.11	0.164
C10	0.005	Center	0.11	0.152
C11	0.005	Center	0.11	0.140
C12	0.005	Center	0.11	0.128
C13	0.005	Center	0.11	0.116
C14	0.005	Center	0.11	0.104
C15	0.005	Center	0.11	0.092
C16	0.005	Center	0.11	0.080

#### Table 6.33Circle Edit Window

Building the second layer of the heating coil is easier than building the first layer. Select circles C1–C16. Using the menu bar, select Edit > Copy. Using the menu bar, select Edit > Paste. Enter r = 0.01 and z = -0.006. Click OK.

Using the menu bar, select Edit > Paste. Enter r = 0.02 and z = 0. Click OK. See Figure 6.137.

#### **Physics Settings**

Select Physics > Scalar Variables. Enter 500 in the nu\_emqa edit window. See Figure 6.138. Click OK.

Select Options > Expressions > Scalar Expressions. Enter the scalar expression for sigma\_Cu\_T as shown in Table 6.34; also see Figure 6.139. Click OK.

NOTE The scalar expression for sigma\_Cu\_T couples the resistivity of copper  $(r_Cu)$  and the temperature (T).

The final geometric step is to create a composite object. Using the menu bar, select Draw > Create Composite Object. Select all of the objects. Check the Keep interior boundaries check box. See Figure 6.140.

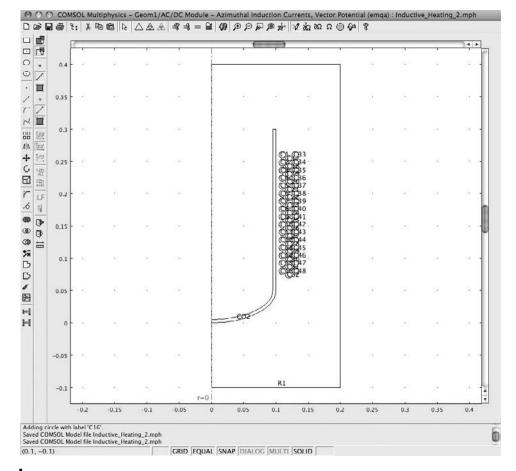


FIGURE 6.137 2D axisymmetric Inductive\_Heating\_2 model, crucible and coil

Name	Expression	Unit	Description				
sigma_htgh	5.67e-8	$W/(m^2 \cdot K^4)$	Stefan-Boltzmann constan				
Rg_htgh	8.31451	J/(mol·K)	Universal gas constant				
epsilon0_emqa	8.854187817e-12	F/m	Permittivity of vacuum				
mu0_emqa	4*pi*1e-7	H/m	Permeability of vacuum				
nu_emqa	500	Hz	Frequency				

**FIGURE 6.138** 2D axisymmetric Inductive\_Heating\_2 model Application Scalar Variables edit window

Name	Expression	Description
sigma_Cu_T	1/(r_Cu*+(1 + alpha_Cu*(T-T_ref)))	Electrical conductivity, Cu

Table 6.34 S	Scalar Expressions	Edit Window
--------------	--------------------	-------------

Click OK. See Figure 6.141.

Now that the 2D axisymmetric Inductive\_Heating\_2 model geometry is built, export it to a file for future use. Select File > Export > Geometry Objects to File. Enter IH\_2\_Geometry in the Save as window. Select "DXF File" from the File Format pull-down list. Click OK.

Name	Expression	Unit	Description
igma_Cu_T	1/(r_Cu*(1+alpha_Cu*(T-T_ref)))	S/m	Electrical conductivity, Cu

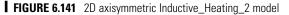
**FIGURE 6.139** 2D axisymmetric Inductive\_Heating\_2 model Scalar Expressions edit window

000	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula: R1+CO2+C1+C2+C3+C4+C5+C6+
C43 C44 C45 C46 C47 C48	Repair Repair 1.0E-4

**FIGURE 6.140** 2D axisymmetric Inductive\_Heating\_2 model Create Composite Object edit window

0 0 0 .....

	88	ዩ:	χ.			4	= 😫 🔇		₽ <b>#</b>	26 26 2	Ω ⊚ ∲	8				
Ľ		FC.	_					e	)						11	FI.
ß				12	6	1					32			1	1	r
٠	0.4			10	τ.			63							8	1
1																L
																1
v	0.35	1		10				•3							258	1
-																L
	0.3	ŀ		43					1		24		14		4	4
置																1
0.0																L
-	0.25	ŀ		10	<b>T</b>			57	i ĝo	Ö.	1				25	1
盟									189							L
<b>正</b> 前前	0.2										1					1
0	0.2															L
1	-								183							1
•	0.15	÷	•	60	¥.;	-			196		1			*	84	ł
eð 10																ſ
	0.1	F							188							1
																L
	0.05	ŀ		6				•	de la		2	7.			15	ł
																L
																L
	0	1	1	-	*	•		•//	*1				200		34	1
																L
	-0.05	-					CO				2					
																1
	-0.1	F	1	8		•			2	14				8	1	
							r=0									
		-(	0.2	-0.15	-0.1	-0.05	0	0.05	0.1	0.15	0.2	0.25	0.3	0.35	0.4	
ed Co	OMSOL N	Aodel	file In	ductive_Hea	ting_2.mph											
ed Co	OMSOL N	Model	file In	ductive_Hea ductive_Hea	ting_2.mph											
	).25)				pi		FOUAL SN	IAP DIALO	G MULTI	SOLID						



# Physics Subdomain Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Multiphysics > Azimuthal Induction Currents, Vector Potential (emqa). Using the menu bar, select Physics > Subdomain Settings > Electric Parameters. In the Subdomain edit windows, enter the information shown in Table 6.35. Click OK. See Figures 6.142, 6.143, and 6.144.

Subdomain	Name	Expression	Description	Figure Number
1	σ	0	Electric conductivity	6.142
2	σ	sigma_Cu_T	Electric conductivity	6.143
3–50	σ	0	Electric conductivity	6.144

# Table 6.35 Subdomain Edit Window

# Physics Boundary Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.36. Check the Interior boundaries check box. See Figures 6.145 and 6.146.

### Table 6.36 Boundary Settings Edit Window

Boundary	Boundary Condition	Figure Number
1, 3, 4	Axial symmetry	6.145
2, 5, 9	Magnetic insulation	6.146

Subdomains Groups	Magnetic Param	eters Electric Parameter	s Infinit	e Elements Forces
ubdomain selection	Electric material	properties and current source	es	
	1 throws material		Load	
U	Library material		Load	
	Constitutive re	lation		
	(•) $\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E}$	$\bigcirc$ D = $\epsilon_0 E + P$	() D = 1	$v_0 \varepsilon_r \mathbf{E} + \mathbf{D}_r$
	Quantity	Value/Expression		Description
	Vicop	0	v	Loop potential
0	J <sup>e</sup> φ	0	A/m*	External current density
1	σ	0	S/m	Electric conductivity
roup:	ε,	1	1	Relative permittivity
		1.		readine permittivity
Select by group				
Active in this domain				

**FIGURE 6.142** 2D axisymmetric Inductive\_Heating\_2 model Subdomain Settings (1), Electric Parameters edit window

Subdomains Groups	Magnetic Paran	neters Electric Parameter	s Infinite Elements Forces
Subdomain selection	Library materia		$D = \varepsilon_0 \varepsilon_r E + D_r$ Unit Description $V$ Loop potential $A/m^2$ External current density $S/m$ Electric conductivity $1$ Relative permittivity

**FIGURE 6.143** 2D axisymmetric Inductive\_Heating\_2 model Subdomain Settings (2), Electric Parameters edit window

Subdomains Groups	Magnetic Paran	neters Electric Paramete	ers Infinit	e Elements Forces
ubdomain selection	Electric material	properties and current source	ces	
0	Library materia		Load	
11	Library materia			
2	Constitutive re	lation		
3	$\mathbf{O} \mathbf{D} = \epsilon_0 \epsilon_r \mathbf{E}$	$\bigcirc$ <b>D</b> = $\epsilon_0 \mathbf{E} + \mathbf{P}$	🔘 D = a	$_{0}\varepsilon_{r}\mathbf{E}+\mathbf{D}_{r}$
5	Quantity	Value/Expression	Unit	Description
6	Vloop	0	v	Loop potential
7				
18 19	۶ <sup>е</sup> ф	0	A/m <sup>2</sup>	External current density
0	σ	0	S/m	Electric conductivity
roup:	٤	1	1	Relative permittivity
Select by group				
Active in this domain				

**FIGURE 6.144** 2D axisymmetric Inductive\_Heating\_2 model Subdomain Settings (3–50), Electric Parameters edit window

Boundaries Groups	Conditions Material Properties Color/Style
oundary selection	Boundary sources and constraints
0	Boundary condition: Axial symmetry
Ť	
roup:	
Select by group	
Select by group	

**FIGURE 6.145** 2D axisymmetric Inductive\_Heating\_2 model Boundary Settings (1, 3, 4) edit window

Boundaries Groups	Conditions Material Properties Color/Style
soundary selection	Boundary sources and constraints Boundary condition: Magnetic insulation
Select by group	

FIGURE 6.146 2D axisymmetric Inductive\_Heating\_2 model Boundary Settings (2, 5, 9) edit window

Table 6.37	Boundary	/ Settings	Edit	Window
------------	----------	------------	------	--------

Boundary	Boundary Condition	Value	Figure Number
12–203	Surface current	Js_0	6.147

FIGURE 6.147 2D axisymmetric Inductive\_Heating\_2 model Boundary Settings (12–203) edit window

In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.37. See Figure 6.147. Click OK. See Figure 6.148.

### Physics Subdomain Settings: General Heat Transfer (htgh)

Using the menu bar, select Multiphysics > General Heat Transfer (htgh). Using the menu bar, select Physics > Subdomain Settings > Conduction. In the Subdomain edit windows, enter the information shown in Table 6.38. See Figures 6.149 and 6.150.

Subdomain	Name	Expression	Description	Figure Number
1	k (isotropic)	k_air	Thermal conductivity	6.149
	ρ	rho_air	Density	
	Ср	Cp_air	Heat capacity	
2–50	k (isotropic)	k_Cu	Thermal conductivity	6.150
	ρ	rho_Cu	Density	
	Ср	Cp_Cu	Heat capacity	

### Table 6.38 Subdomain Edit Window

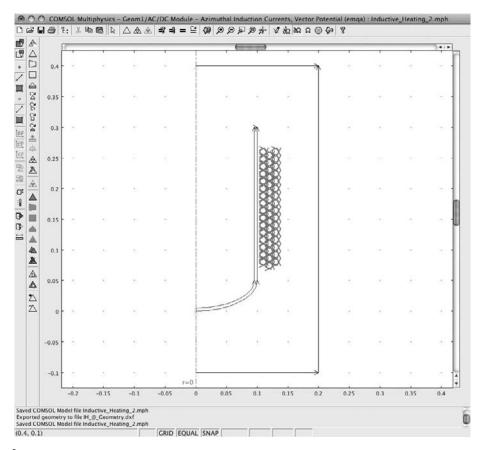


FIGURE 6.148 2D axisymmetric Inductive\_Heating\_2 model, coil

Select the Init tab. Select subdomains 1–50. Enter T\_ref in the  $T(t_0)$  edit window. See Figure 6.151. Click OK.

### **Physics Boundary Settings: General Heat Transfer (htgh)**

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – General Heat Transfer (htgh) edit windows, enter the information shown in Table 6.39. Click OK. See Figures 6.152 and 6.153.

Table 6.3	89 Bound	ary Settings	Edit Window
-----------	----------	--------------	-------------

Boundary	Boundary Condition	Value	Figure Number
1, 3, 4	Axial symmetry	—	6.152
2, 5, 9	Temperature	T_ref	6.153

= temperature Subdomains Groups	Conduction Convection Ideal Gas Init	Element Color
Subdomains election Subdomain selection 2 (default) 3 (default) 3 (default) 5 (default) 5 (default) 9 (default) 10 (default) 10 (default) 11 (default) 12 (default) 13 (default) 13 (default) 14 (default) 15 (default) 16 (default) 16 (default) 17 (default) 18 (default) 18 (default) 19 (default) 10 (defaul	$\label{eq:constraint} \begin{array}{c c c c c c c c c c c c c c c c c c c $	scription ne-scaling coefficient ermal conductivity ermal conductivity nsity at capacity at constant pressure at source

FIGURE 6.149 2D axisymmetric Inductive\_Heating\_2 model Subdomain Settings (1) edit window

Subdomains Groups	Conduction	on Convection Id	eal Gas Init Element Color
Subdomain selection  31 (default)  33 (default)  33 (default)  34 (default)  35 (default)  35 (default)  35 (default)  36 (default)  41 (default)  41 (default)  42 (default)  42 (default)  44 (default)  45 (default)  46 (default)  46 (default)  46 (default)  46 (default)  47 (default)  48 (default)  48 (default)  48 (default)  48 (default)  50 (defau	Thermal properties a Library material: Quantity δ <sub>rc</sub> (e) k (isotropic) (c) k (anisotropic) ρ C <sub>μ</sub> Q Opacity:	And heat sources/sinks	Load Unit Description I Time-scaling coefficient W/(m·K) Thermal conductivity W/(m·K) Thermal conductivity kg/m² Density J/(kg·K) Heat capacity at constant pressur W/m³ Heat source

FIGURE 6.150 2D axisymmetric Inductive\_Heating\_2 model Subdomain Settings (2–50) edit window

$pC_p \partial T / \partial t + \nabla \cdot (-k \nabla T) = Q$ = temperature	
Subdomains Groups	Conduction Convection Ideal Gas Init Element Color
Subdomain selection	- Initial value
11 (default) 32 (default) 33 (default) 34 (default) 35 (default) 36 (default) 36 (default) 38 (default) 40 (default) 41 (default) 41 (default) 43 (default) 43 (default) 45 (default) 45 (default) 45 (default) 46 (default) 47 (default) 47 (default) 48 (default) 49 (default) 50	T(t <sub>0</sub> ) T_ref

FIGURE 6.151 2D axisymmetric Inductive\_Heating\_2 model Subdomain Settings (1–50), Init edit window

Boundaries Groups	Boundary Cond	ition Highly Conducti	ve Layer	Element Color/Style
Soundary selection	Boundary sources and	d constraints		
2 0	Library coefficient:	•	Load	
3	Boundary condition:	Axial symmetry	6	
5	Quantity	Value/Expression	Unit	Description
5	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
9	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
10	T <sub>inf</sub>	273.15	к	External temperature
11	т	273.15	к	Temperature
13	Radiation type:	None	<b></b>	
15	٤	0	1	Surface emissivity
	Tamb	0	к	Ambient temperature
iroup:	Jo	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Select by group	Member of group(s):	epsilon_nign_sigma_n	1	surface radiosity expression

**FIGURE 6.152** 2D axisymmetric Inductive\_Heating\_2 model Boundary Settings (1, 3, 4) edit window

juation = T <sub>0</sub>				
Boundaries Groups	Boundary Cond	ition Highly Conducti	ve Layer	Element Color/Style
Boundary selection	Boundary sources and	l constraints		
1	Library coefficient:	•	Load	
2 3 4	Boundary condition:	Temperature	•	
5	Quantity	Value/Expression	Unit	Description
б 7	۹ <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
9	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
10	Tinf	273.15	к	External temperature
11 12	т	T_ref	к	Temperature
13	Radiation type:	None	\$	
15	ε	0	1	Surface emissivity
iroup:	T <sub>amb</sub>	0	к	Ambient temperature
Select by group	J <sub>0</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

**FIGURE 6.153** 2D axisymmetric Inductive\_Heating\_2 model Boundary Settings (2, 5, 9) edit window

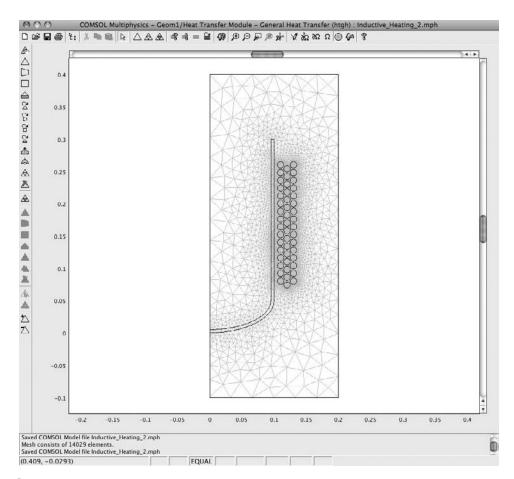
#### **Mesh Generation**

On the toolbar, click the Initialize Mesh button once. This mesh yields approximately 14,000 elements. See Figure 6.154.

### Solving the 2D Axisymmetric Inductive\_Heating\_2 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.



**FIGURE 6.154** 2D axisymmetric Inductive\_Heating\_2 model mesh window

Enter linspace(0,1200,21) in the Times edit window. See Figure 6.155.

Click the Advanced tab. Check the Use complex functions with real input check box. See Figure 6.156. Click OK.

#### Time-Dependent Solving of the 2D Resistive\_Heating\_2 Model

Select Solve > Solve Problem. See Figure 6.157.

#### Postprocessing and Visualization

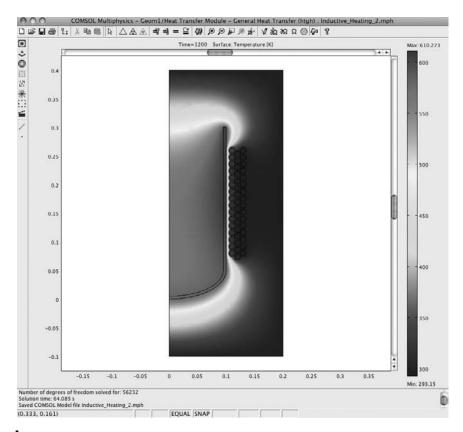
The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot Parameters > Surface. Verify that the Surface plot check box is checked and that

Analysis:	G	eneral Time Stepping	Advanced
Transient     Image: Constraint of the solution of t		linspa 0.01 0.001	ce(0,1200,21)
	Matrix symmetry:	Automatic	Settings
		(Help) (Apply	Cancel OK

**FIGURE 6.155** 2D axisymmetric Inductive\_Heating\_2 model Solver Parameters edit window

Analysis:	General T	ime Stepping Advanced	
Transient	Constraint handling method:	Elimination	•
Solver.	Null-space function:	Automatic	•
Stationary Lime dependent	Assembly block size:	5000	
Eigenvalue	Use Hermitian transpose of co	nstraint matrix and in symmetry	detection
Parametric Stationary segregated	Use complex functions with re	al input	
Parametric segregated	Stop if error due to undefined	operation	
	Store solution on file		
	Solution form:	Automatic	0
Adaptive mesh refinement	Scaling of variables		
	Type of scaling:	Automatic	\$
	Type of searing.		
	Manual scaling		
		On	(\$
	Manual scaling		•
	Manual scaling: Row equilibration:		•
	Manual scaling: Row equilibration:	/	

**FIGURE 6.156** 2D axisymmetric Inductive\_Heating\_2 model Solver Parameters, Advanced edit window



**FIGURE 6.157** 2D axisymmetric Inductive\_Heating\_2 model solution

the Predefined quantities pull-down list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.158.

Click OK. See Figure 6.159.

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.160.

Click the Start Animation button. See Figure 6.161.

Alternatively, you can play the file Movie6\_IH\_2.avi that was supplied with this book.

# Second Variation on the 2D Axisymmetric Inductive Heating Model

The following numerical solution model (Inductive\_Heating\_3) is derived from the Inductive\_Heating\_2 model. It continues the introduction of the coupling of two

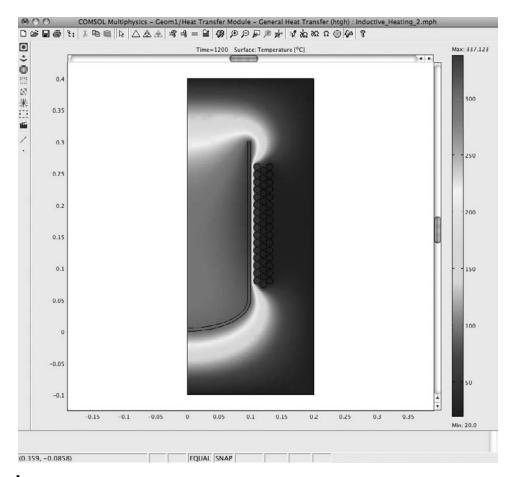
Predefined quantities:	Contour Surface Data Temperature T °C	Boundary Height Data	Arrow Range	
Predefined quantities: ( Expression: [ Unit: (	Temperature T			
Predefined quantities: ( Expression: Unit: (	Temperature T			
Expression:	т			
Expression:	т			
Unit: (	<u> </u>	•	Smoot	n
	°С	·		
Coloring and fill				
Coloring and fill				
Coloring: Interpola	ted 🗘	Fill style:	Filled	\$
Surface color				
O Colormap:	et 🛟	Colors: 1024	Color	scale
O Uniform color:	Color			

**FIGURE 6.158** 2D axisymmetric Inductive\_Heating\_2 model Plot Parameters edit window

important basic physical materials properties—Joule heating and heat transfer—and expands on these concepts. The coupling of these two properties in this model demonstrates one of the applications normally found in typical engineering or process research.

This model is similar to the Inductive\_Heating\_2 model in the use of the induction heating method. However, in this case, the modeler will build a filled (loaded) inductively heated crucible. Here, bismuth<sup>21,22</sup> is the material of choice. Additionally, the crucible will be surrounded by nitrogen<sup>23</sup> to prevent oxidation of the heated bismuth.

Modeling Joule heating is important in a wide variety of physical design and applied engineering problems. Typically, the modeler desires to understand Joule heat



**FIGURE 6.159** 2D axisymmetric Inductive\_Heating\_2 model, degrees Centigrade

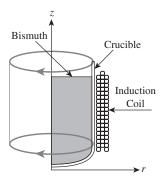
generation during a process and either adds heat or removes heat so as to achieve or maintain a desired temperature. Figure 6.162 shows a 3D rendition of the 2D axisymmetric Inductive\_Heating\_3 model geometry, as will be modeled here.

To start building the Inductive\_Heating\_3 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select AC/DC Module > Electro-Thermal Interaction > Azimuthal Induction Heating > Transient analysis. See Figure 6.163. Click OK.

There are two mode names in the Application mode name window (htgh emqa). The names in the window show which applications are coupled in this application: General Heat Transfer (htgh) and Azimuthal Induction Currents, Vector Potential (emqa).

Principal Str Movie settings File type:	eamline Particle Tra	Solutions to use	Deform Animat
		- Colutions to use	
File type:		solutions to use	
· · · · · · · · · · · · · · · · · · ·	AVI 🛟	Select via Store	ed output times 🔹
Width (in pixels)	640	0	
Height (in pixels):	480	60 120	
Frames per second:	10	180	U
rrames per second.		240 300	
	(Advanced)	360	
Static / Eigenfunctio	n animation	420 480	¥.
Cycle type:	Full harmonic	540	•
Number of frames:	11	Times:	
			Start Animation

**FIGURE 6.160** 2D axisymmetric Inductive\_Heating\_2 model Plot Parameters window



**FIGURE 6.162** 3D rendition of the 2D axisymmetric Inductive\_Heating\_3 model

# **Options and Settings**

# Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 6.40; see also Figure 6.164. Click OK.

Select File > Import > CAD Data From File > IH\_2\_Geometry.dxf. Click the Import button.

0 0	Model Naviga	itor
New	Model Library User Mode	els Open Settings
Space dimension:	Axial symmetry (2D)	
<ul> <li>Electro-Therr</li> <li>Azimuthal</li> <li>Steady</li> <li>Trans</li> </ul>	Electric Magnetic Electromagnetic nal Interaction Induction Heating r-state analysis ent analysis I Induction Heating	AC/DC Module Description: A predefined combination of application modes for induction heating. Combines Azimuthal Induction Currents with heat transfer by conduction for modeling of electro-thermal applications such as induction heating.
Dependent variables:	T J Aphidr	Transient analysis.
Application mode name: Element:	htgh emqa Lagrange - Aphidr <sub>2</sub>	Multiphysics

FIGURE 6.163 2D axisymmetric Inductive\_Heating\_3 Model Navigator setup

Name	Expression	Description
I_0	4e2[A]	Coil current
d_0	1e-2[m]	Coil diameter
c_0	pi*d_0	Coil circumference
Js_0	I_0/c_0	Coil surface current density
T_refCu	20[degC]	Reference temperature Cu
T_refBi	0[degC]	Reference temperature Bi
r_Cu	1.67e-8[ohm*m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	Resistivity coefficient copper
rho_N2	1.25[kg/m^3]	Density nitrogen STP
Cp_N2	1.03e3[J/(kg*K)]	Heat capacity N2
k_N2	2.512e-2[W/(m*K)]	Thermal conductivity N2
rho_Cu	8.96e3[kg/m^3]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	Thermal conductivity copper
r_Bi	1.068e-6[ohm*m]	Resistivity Bi at T_refBi
alpha_Bi	1.7e-3[1/K]	Temperature coefficient Bi
k_Bi	8.374[W/(m*K)]	Thermal conductivity Bi
rho_Bi	9.8e3[kg/m^3]	Density Bi
Cp_Bi	1.23e2[J/(kg*K)]	Heat capacity Bi

Table 6.40	<b>Constants Edit Window</b>	
------------	------------------------------	--

Name	Expression	Value	Description
I_0	4e2[A]	400[A]	Coil current
d_0	1e-2[m]	0.01[m]	Coil diameter
c_0	pi*d_0	0.031416[m]	Coil circumference
Js_0	I_0/c_0	12732.395447[A/m]	Coil surface current density
T_refCu	20[degC]	293.15[K]	Reference temperature Cu
T_refBi	0[degC]	273.15[K]	Reference temperature Bi
r_Cu	1.67e-8[ohm*m]	(1.67e-8)[Ω·m]	Resistivity copper
alpha_Cu	3.9e-3[1/K]	0.0039[1/K]	Resistivity coefficient copper
rho_N2	1.25[kg/m^3]	$1.25[kg/m^3]$	Density nitrogen STP
Cp_N2	1.03e3[J/(kg*K)]	1030[J/(kg·K)]	Heat capacity N2
k_N2	2.512e-2[W/(m*K)]	0.02512[W/(m·K)]	Thermal conductivity N2
rho_Cu	8.96e3[kg/m^3]	8960[kg/m <sup>3</sup> ]	Density copper
Cp_Cu	3.8e2[J/(kg*K)]	380[J/(kg·K)]	Heat capacity copper
k_Cu	3.94e2[W/(m*K)]	394[W/(m·K)]	Thermal conductivity copper
r_Bi	1.068e-6[ohm*m]	(1.068e-6)[Ω·m]	Resistivity Bi at T_refBi
alpha_Bi	1.7e-3[1/K]	0.0017[1/K]	Temperature coefficient Bi
k_Bi	8.374[W/(m*K)]	8.374[W/(m·K)]	Thermal conductivity Bi
rho_Bi	9.8e3[kg/m^3]	9800[kg/m <sup>3</sup> ]	Density Bi
Cp_Bi	1.23e2[J/(kg*K)]	123[J/(kg·K)]	Heat capacity Bi

I FIGURE 6.164 2D axisymmetric Inductive\_Heating\_3 model Constants edit window

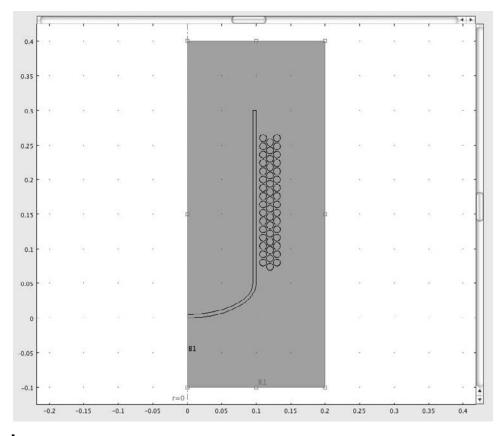


FIGURE 6.165 2D axisymmetric Inductive\_Heating\_3 model, imported crucible and coil with rectangle

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.2 and a height of 0.5. Select "Base: Corner" and set r equal to 0 and z equal to -0.1 in the Rectangle edit window. Click OK. See Figure 6.165.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1-0.005 and a height of 0.25-0.05. Select "Base: Corner" and set r equal to 0 and z equal to 0.05 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes of 0.1-0.005 and B-semiaxes of 0.05-0.005. Select "Base: Corner" and set r equal to 0 and z equal to 0.05 in the Ellipse edit window. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.1 and a height of 0.30. Select "Base: Corner" and set r equal to -0.1 and z equal to 0 in the Rectangle edit window. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter E1–R3 in the Set formula edit window. Click OK.

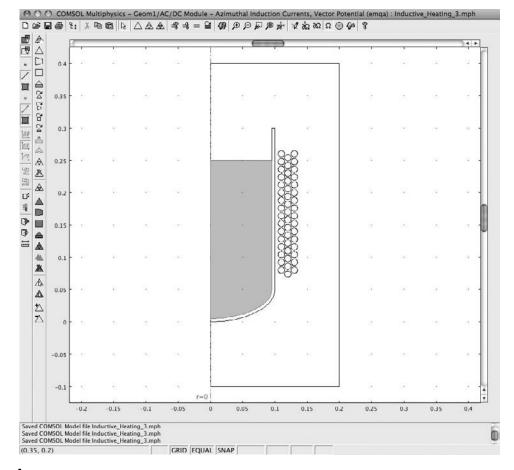


FIGURE 6.166 2D axisymmetric Inductive\_Heating\_3 model, bismuth subdomain 3

Using the menu bar, select Draw > Create Composite Object. Enter R2+CO1 in the Set formula edit window. Verify that the Keep interior boundaries check box is not checked. Click OK.

Select Physics > Subdomain Settings. Select subdomain 3 to verify that the bismuth subdomain has been properly added. Click OK. See Figure 6.166.

#### **Physics Settings**

Select Physics > Scalar Variables. Enter 500 in the nu\_emqa edit window. See Figure 6.167. Click OK.

Select Options > Expressions > Scalar Expressions. Enter the scalar expressions for sigma\_Cu\_T and sigma\_Bi\_T as shown in Table 6.41; also see Figure 6.168. Click OK.

Name	Expression	Description
sigma_Cu_T	1/(r_Cu*(1+alpha_Cu*(T-T_refCu)))	Electrical conductivity, Cu
sigma_Bi_T	1/(r_Bi*(1+alpha_Bi*(T-T_refBi)))	Electrical conductivity, Bi

### Table 6.41 Scalar Expressions Edit Window

	Expression	Unit	Description	
sigma_htgh Rg_htgh epsilon0_emqa mu0_emqa	5.67e-8	$W/(m^2 \cdot K^4)$	Stefan-Boltzmann constan	
	8.31451	J/(mol·K)	Universal gas constant	
	8.854187817e-12	F/m	Permittivity of vacuum Permeability of vacuum	
	4*pi*1e-7	H/m		
nu_emqa	500	Hz	Frequency	
nu_emqa	500	Hz	Frequency	

FIGURE 6.167 2D axisymmetric Inductive\_Heating\_3 model Application Scalar Variables edit window

**NOTE** The scalar expression for sigma\_Cu\_T couples the resistivity of copper (r\_Cu) and the temperature (T). The scalar expression for sigma\_Bi\_T couples the resistivity of bismuth (r\_Bi) and the temperature (T).

# Physics Subdomain Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Multiphysics > Azimuthal Induction Currents, Vector Potential (emqa). Using the menu bar, select Physics > Subdomain Settings > Electric

Subdomain	Name	Expression	Description	Figure Number
1	σ	0	Electric conductivity	6.169
2	σ	sigma_Cu_T	Electric conductivity	6.170
3	σ	sigma_Bi_T	Electric conductivity	6.171
4–51	σ	0	Electric conductivity	6.172

### Table 6.42 Subdomain Edit Window

Parameters. In the Subdomain edit windows, enter the information shown in Table 6.42. Click OK. See Figures 6.169–6.172.

### Physics Boundary Settings: Azimuthal Induction Currents, Vector Potential (emqa)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.43. Check the Interior boundaries check box. See Figures 6.173 and 6.174.

Subdomains Groups	Magnetic Paran	eters Electric Paramete	rs Infinite Elements	Forces
ubdomain selection	Electric material	properties and current sourc	es	
0	Library materia		Load	
U	LIDIALY MALEMA		LUau	
	Constitutive re	lation		
	$\bigcirc$ D = $\epsilon_0 \epsilon_r E$	$\bigcirc$ D = $\epsilon_0 E + P$	$\bigcirc$ <b>D</b> = $\epsilon_0 \epsilon_r \mathbf{E} + \mathbf{D}_r$	
	Quantity	Value/Expression	Unit Descriptio	'n
	V <sub>loop</sub>	0	V Loop poter	
		U	Loop poter	itiai
	۶ <sup>е</sup> φ	0	A/m <sup>2</sup> External cu	urrent density
0 A	σ	0	S/m Electric con	nductivity
		U		lauctivity
roup:	٤ŗ	1	1 Relative pe	rmittivity
Select by group				
Active in this domain				

FIGURE 6.169 2D axisymmetric Inductive\_Heating\_3 model Subdomain Settings (1), Electric Parameters edit window

Subdomains Groups	Magnetic Paran	neters Electric Parameter	s Infinite	e Elements Forces
ubdomain selection	Library materia		D = a Unit	0 <sup>ε</sup> r <mark>E + D</mark> r <b>Description</b> Loop potential External current density Electric conductivity
iroup:	٤r	1	1	Relative permittivity

**FIGURE 6.170** 2D axisymmetric Inductive\_Heating\_3 model Subdomain Settings (2), Electric Parameters edit window

Subdomains Groups	Magnetic Paran	neters Electric Parameters	Infinite Elements Forces
ubdomain selection	-Electric material	properties and current source	S
	Library material		.oad )
	Constitutive re	lation	
	$\square$ D = $\varepsilon_{\alpha}\varepsilon$ E	$\bigcirc$ D = $\epsilon_0 E + P$	$\bigcirc$ <b>D</b> = $\varepsilon_0 \varepsilon_r \mathbf{E} + \mathbf{D}_r$
	0.0	0	Unit Description
	Quantity	Value/Expression	v
	Vloop	0	V Loop potential
	J <sup>e</sup> پ	0	A/m <sup>2</sup> External current density
0			
.1 🔹	σ	sigma_Bi_T	S/m Electric conductivity
roup:	٤	1	1 Relative permittivity
Select by group			
Active in this domain			

**FIGURE 6.171** 2D axisymmetric Inductive\_Heating\_3 model Subdomain Settings (3), Electric Parameters edit window

# Table 6.43 Boundary Settings Edit Window

Boundary	Boundary Condition	Figure Number
1, 3–7	Axial symmetry	6.173
2, 9, 14	Magnetic insulation	6.174

Subdomains Groups	Magnetic Param	eters Electric Parameter	rs Infinite Elements Forces
ubdomain selection	Electric material	properties and current sourc	es
1 2 3 4	Library material		Load
4 5	(i) $D = \epsilon_0 \epsilon_r E$	$\bigcirc$ D = $\epsilon_0 E + P$	$\bigcirc D = \epsilon_0 \epsilon_r E + D_r$
6	Quantity	Value/Expression	Unit Description
7	Vloop	0	V Loop potential
* n			
9	٦ <sup>e</sup> qu	0	A/m <sup>2</sup> External current density
1	σ	0	S/m Electric conductivity
roup:	ε,		1 Palatius normisticitu
		1	Relative permittivity
Select by group			
Active in this domain			

**FIGURE 6.172** 2D axisymmetric Inductive\_Heating\_3 model Subdomain Settings (4–51), Electric Parameters edit window

Boundaries Groups	Conditions Material Properties Color/Style
oundary selection	Boundary sources and constraints
0	Boundary condition: Axial symmetry
roup:	
Select by group	
Interior boundaries	

**FIGURE 6.173** 2D axisymmetric Inductive\_Heating\_3 model Boundary Settings (1, 3–7) edit window

Boundary	Boundary Condition	Value	Figure Number
17–208	Surface current	Js_0	6.175

p = 0	
Boundary selection Boundary selection Boundary selection B D D D D D D D D D D D D D D D D D D	Conditions         Material Properties         Color/Style           Boundary sources and constraints         Boundary condition:         Magnetic insulation

**FIGURE 6.174** 2D axisymmetric Inductive\_Heating\_3 model Boundary Settings (2, 9, 14) edit window

In the Boundary Settings – Azimuthal Induction Currents, Vector Potential (emqa) edit windows, enter the information shown in Table 6.44. See Figure 6.175.

Click OK. See Figure 6.176.

Boundaries Groups	Condition	ons Material Properti	es Color/Style
Boundary selection	Boundary sources and con	straints	
201	Boundary condition:	Surface current	•
202 203 204	Quantity	Value/Expression	Unit Description
205	$J_{s\phi}$	Js_0	A/m Surface current density
207			
Group.			
Select by group			
Interior boundaries			

FIGURE 6.175 2D axisymmetric Inductive\_Heating\_3 model Boundary Settings (17–208) edit window

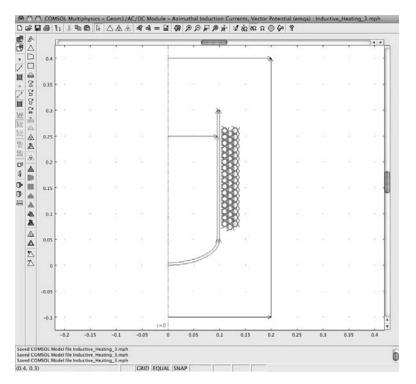


FIGURE 6.176 2D axisymmetric Inductive\_Heating\_3 model, coil

# Physics Subdomain Settings: General Heat Transfer (htgh)

Using the menu bar, select Multiphysics > General Heat Transfer (htgh). Using the menu bar, select Physics > Subdomain Settings > Conduction. In the Subdomain edit windows, enter the information shown in Table 6.45. See Figures 6.177, 6.178, and 6.179.

	Table 6.45	Subdomain Edit Window
--	------------	-----------------------

Subdomain	Name	Expression	Description	Figure Number
1	k (isotropic)	k_N2	Thermal conductivity	6.177
	ρ	rho_N2	Density	
	Ср	Cp_N2	Heat capacity	
2, 4–51	k (isotropic)	k_Cu	Thermal conductivity	6.178
	ρ	rho_Cu	Density	
	<i>С</i> р	Cp_Cu	Heat capacity	
3	k (isotropic)	k_Bi	Thermal conductivity	6.179
	ρ	rho_Bi	Density	
	Ср	Cp_Bi	Heat capacity	

quation $_{ts}\rho C_{\rho} \partial T / \partial t + \nabla \cdot (-k \nabla T) = Q$ = temperature		
Subdomains Groups Subdomain selection 2 (default) 3 (default) 4 (default) 5 (default) 5 (default) 9 (default) 10 (default) 11 (default) 11 (default) 13 (default) 13 (default) 14 (default) 14 (default) 15 (default) 16 (default) 17 (default) 16 (defaul	Conduction     Conv       Thermal properties and heat sou     Library material:       Quantity     Value/Exp $\delta_{rc}$ 1 $\Theta$ k (isotropic)       k (anisotropic)     400 0 0 4 $\rho$ rho_N2 $C_p$ $C_pN2$ Q     Qav_enq       Opacity:     Opaque	Unit       Description         1       Time-scaling coefficient         W/(m·K)       Thermal conductivity         W/(m·K)       Thermal conductivity         W/(m-K)       Thermal conductivity         W/(m-K)       Thermal conductivity         J/(kg·K)       Density         J/(kg·K)       Heat capacity at constant pressure         ya       W/m <sup>3</sup>

FIGURE 6.177 2D axisymmetric Inductive\_Heating\_3 model Subdomain Settings (1) edit window

Subdomains Groups	Conduction	on Convection Id	eal Gas Init Element Color
Subdomain selection	Thermal properties a Library material: Quantity δ <sub>rr</sub> (c) k (inisotropic) ρ C <sub>p</sub> Q Opacity:	Value/Expression  K Cu  K Cu  K Cu  Cp.Cu  Qav_emqa  Opaque  C	Load Unit Description I Time-scaling coefficient W/(m-K) Hhermal conductivity W/(m-K) Kg/m <sup>2</sup> Density J/(kg-K) Heat capacity at constant pressure W/m <sup>3</sup> Heat source

FIGURE 6.178 2D axisymmetric Inductive\_Heating\_3 model Subdomain Settings (2, 4–51) edit window

Subdomains Groups	Conduction	on Convection Id	eal Gas Init Element Color
Subdomain selection	Thermal properties a Library material: Quantity $\delta_{ts}$ o k (isotropic) $\rho$ $C_p$ Q Opacity:	Value/Expression          I         k_Bi         400 0 0 400         rho_Bi         Cp_Bi         Qav_emqa         Opaque	Load Unit Description I Time-scaling coefficient W/(m·K) Thermal conductivity W/(m·K) Thermal conductivity kg/m <sup>3</sup> Uensity J/(kg·K) Heat capacity at constant pressure W/m <sup>3</sup> Heat source

**FIGURE 6.179** 2D axisymmetric Inductive\_Heating\_3 model Subdomain Settings (3) edit window

Select the Init tab. Select subdomains 1–51. Enter T\_refCu in the  $T(t_0)$  edit window. See Figure 6.180. Click OK.

# Physics Boundary Settings: General Heat Transfer (htgh)

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings – General Heat Transfer (htgh) edit windows, enter the information shown in Table 6.46. Click OK. See Figures 6.181 and 6.182.

Table 6.46	Boundary	/ Settings	Edit	Window
------------	----------	------------	------	--------

Boundary	Boundary Condition	Value	Figure Number
1, 3–7	Axial symmetry	—	6.181
2, 9, 14	Temperature	T_refCu	6.182

uation		
$\partial C_p \partial T / \partial t + \nabla \cdot (-k \nabla T) = Q$		
temperature		
Subdomains Groups	Conduction Convection Ideal Gas Init Element Color	
ibdomain selection	r Initial value	
2 3 4 5 6 6 7 8 9 0 1 2 3 4 4 5 6 6 7 8 9 0 1 2 4 5 5 6 6 7 8 9 0 1 1 2 4 5 5 6 6 7 8 9 0 0 1 1 4 5 5 6 6 7 7 8 9 0 0 1 1 4 5 5 6 6 7 7 8 9 9 0 0 1 1 5 5 6 6 7 7 8 9 9 0 0 1 1 5 5 6 6 7 7 8 9 9 0 0 1 1 5 5 6 6 7 7 8 9 9 0 0 1 1 5 7 7 8 9 9 0 0 1 1 5 7 8 9 9 0 0 1 1 1 5 7 7 8 9 9 0 0 1 1 1 5 7 7 8 9 9 0 0 1 1 5 7 8 9 9 0 0 1 1 1 1 5 7 8 9 9 0 0 1 1 1 1 3 1 1 1 1 1 1 1 1 1 1 1 1 1	T(fp) [J_refCu K Temperature	
roup: 🚺 🗘		

**FIGURE 6.180** 2D axisymmetric Inductive\_Heating\_3 model Subdomain Settings (1–51), Init edit window

uation = T <sub>0</sub>				
Boundaries Groups	Boundary Cond	ition Highly Conduct	ve Layer	Element Color/Style
Boundary selection	Boundary sources and Library coefficient:	l constraints	Load	
3 4	Boundary condition:		•	
5 6	Quantity q <sub>0</sub>	Value/Expression	Unit W/m <sup>2</sup>	Description
8	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
9 10	T <sub>inf</sub>	273.15	к	External temperature
11 12	τ <sub>0</sub>	T_refCu	к	Temperature
13 14	Radiation type:	None	\$	
15 16	ε	0	1	Surface emissivity
iroup:	T <sub>amb</sub>	0	к	Ambient temperature
Select by group	J <sub>o</sub>	epsilon_htgh*sigma_h	W/m <sup>2</sup>	Surface radiosity expression
Interior boundaries	Member of group(s):		1	

FIGURE 6.182 2D axisymmetric Inductive\_Heating\_3 model Boundary Settings (2, 9, 14) edit window

#### **Mesh Generation**

On the toolbar, click the Initialize Mesh button once. This mesh yields approximately 13,500 elements. See Figure 6.183.

#### Solving the 2D Axisymmetric Inductive\_Heating\_3 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Time dependent solver.

**NOTE** The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Enter linspace(0,1200,21) in the Times edit window. See Figure 6.184.

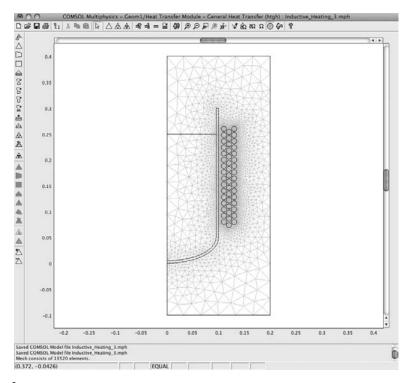


FIGURE 6.183 2D axisymmetric Inductive\_Heating\_3 model mesh window

Iransient 🗘	Gene	ral Time Stepp	ning Advanced	1
Auto select solver Solver: Stationary Time dependent Ligenvalue Parametric Stationary segregated Parametric segregated	Time stepping Times: Relative tolerance: Absolute tolerance: Allow complex numb Linear system solver Linear system solver:	ers Direct (UMFPACK	linspace(0,1200 0.01 0.0010	,21)
Adaptive mesh refinement	Preconditioner:		\$	(Settings
	Matrix symmetry:	Automatic		

FIGURE 6.184 2D axisymmetric Inductive\_Heating\_3 model Solver Parameters edit window

Analysis:	General Tim	ne Stepping Advanced	
Transient 🛟		re stepping ridraneed	
Auto select solver	Constraint handling method:	Elimination	
olver:	Null-space function:	Automatic	
itationary	Assembly block size:	5000	
'ime dependent igenvalue	Use Hermitian transpose of cons	straint matrix and in symmetry	/ detectio
arametric tationary segregated	Use complex functions with real	input	
arametric segregated	Stop if error due to undefined of		
	Store solution on file		
	Solution form:	Automatic	1
Adaptive mesh refinemen	Scaling of variables	L	
	Type of scaling:	Automatic	\$
	Manual scaling:		(esse
		0	
	Row equilibration:	On	\$
	Manual control of reassembly		
	Manual control of reassembly		
	Load constant	🗹 Jacobian constant	
		☑ Jacobian constant ☑ Constraint Jacobian con	stant

FIGURE 6.185 2D axisymmetric Inductive\_Heating\_3 model Solver Parameters, Advanced edit window

Click the Advanced button. Check the Use complex functions with real input check box. See Figure 6.185. Click OK.

#### Time-Dependent Solving of the 2D Resistive\_Heating\_3 Model

Select Solve > Solve Problem. See Figure 6.186.

#### Postprocessing and Visualization

The default plot shows the temperature distribution in kelvins. The temperature distribution can also be shown in degrees Centigrade. To do so, select Postprocessing > Plot Parameters > Surface. Verify that the Surface plot check box is checked and that the Predefined quantities pull-down list shows "Temperature." Select "degC" or "°C" from the Unit pull-down list. See Figure 6.187.

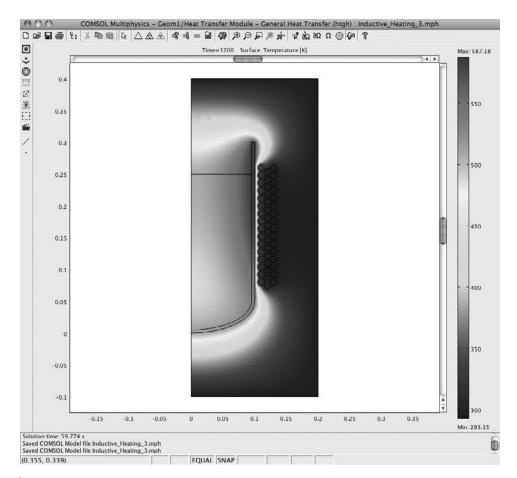


FIGURE 6.186 2D axisymmetric Inductive\_Heating\_3 model solution

Click OK. See Figure 6.188.

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 6.189.

Click the Start Animation button. See Figure 6.190.

Alternatively, you can play the file Movie6\_IH\_3.avi that was supplied with this book.

# 2D Axisymmetric Inductive Heating Models: Summary and Conclusions

The models presented in this section of Chapter 6 have introduced the following new concepts: AC induction and mixed-materials modeling. Previously introduced

0		Plot F	Parameters		
General	Surface	Contour	Boundary	Arrow	Principal
🗹 Surface	plot				
	(	Surface Data	Height Data		
Predefine	d quantities:	Temperature	\$	Range	
Expressio	n:	Т		Smoo	th
Unit:		°C	\$	)	
Coloring an	d fill				
Coloring:	Interpol	ated	Fill style:	Filled	\$
<ul> <li>Colorr</li> <li>Unifor</li> </ul>		Color	Colors: 1024	Color	scale

FIGURE 6.187 2D axisymmetric Inductive\_Heating\_3 model Plot Parameters edit window

concepts included Ohm's law, Joule heating, mixed-mode modeling, triangular mesh, transient analysis, the good first approximation, and 2D axisymmetric coordinates.

The three 2D axisymmetric inductive heating models demonstrate the difference in level of complexity between single-coil and multi-coil models. In the Inductive\_Heating\_1 model, the concept of inductively produced heating was introduced. In the Inductive\_Heating\_2 model, the concept of inductively

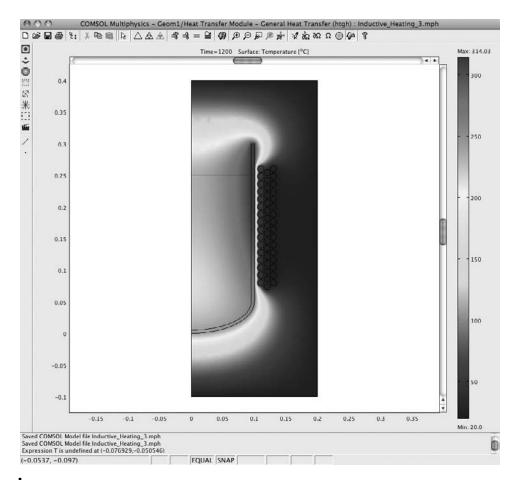


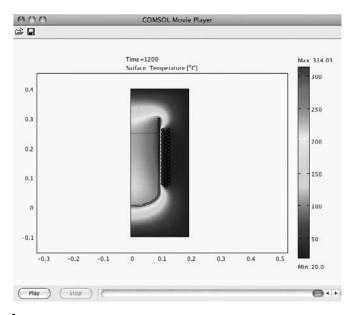
FIGURE 6.188 2D axisymmetric Inductive\_Heating\_3 model degrees Centigrade

produced heating as applied to a practical application (a heated crucible) was used to present one example of the diverse applied scientific and engineering model designs that can be explored using electro-thermal coupling and transient analysis. In the Inductive\_Heating\_3 model, the crucible was filled with a commonly used metal for melting.

These models are examples of the good first approximation type of model. They demonstrate the significant power of relatively simple physical principles, such as Ohm's law and Joule's law, when applied in the COMSOL Multiphysics modeling environment. They could, of course, be modified by the addition of calculations insulating materials and heat loss through convection, among other changes.

Princip	pal 9	Streamline	Particle Tracing	Max/Min	Deform	Anima
ovie set				olutions to use		
	ungs	C 410				
le type:		AVI	•	elect via Sto	red output tim	ies 😜
idth (in	pixels)	640		)		m.
tight (in	pixels):	480		0 20		
ames p	er second	d: 10				Ψ
				40 100		
		Adv	anced			
atic / Ei	genfunct	ion animation		20 180		¥
cle type	e;	Full harr		40	_	•
umber o	f frames:	11	Т	imes:		
					Start A	nimation
						(Start A

**FIGURE 6.189** 2D axisymmetric Inductive\_Heating\_3 model Plot Parameters window





#### References

- 1. http://en.wikipedia.org/wiki/Ohm%27s\_Law
- 2. http://en.wikipedia.org/wiki/Joule\_Heating
- 3. http://en.wikipedia.org/wiki/Direct\_current
- 4. http://en.wikipedia.org/wiki/AC\_current
- 5. http://en.wikipedia.org/wiki/Three\_body\_problem
- 6. http://en.wikipedia.org/wiki/Georg\_Ohm
- 7. http://en.wikipedia.org/wiki/James\_Joule
- 8. http://en.wikipedia.org/wiki/Ohm%27s\_Law
- 9. http://en.wikipedia.org/wiki/Joule%27s\_Law
- 10. http://en.wikipedia.org/wiki/Issac\_newton
- 11. http://en.wikipedia.org/wiki/Newton%27s\_Law\_of\_Cooling #Newton.27s\_ law\_of\_cooling
- 12. http://en.wikipedia.org/wiki/Joseph\_Fourier
- 13. http://en.wikipedia.org/wiki/Fourier%27s\_Law
- 14. http://en.wikipedia.org/wiki/Ohm%27s\_Law
- 15. http://en.wikipedia.org/wiki/Joule%27s\_Law
- 16. http://en.wikipedia.org/wiki/James\_Joule
- 17. http://en.wikipedia.org/wiki/Léon\_Foucault
- 18. http://en.wikipedia.org/wiki/Eddy\_currents
- 19. file:///Applications/COMSOL34/doc/acdc/wwhelp/wwhimpl/common/html/ wwhelp.htm?context=acdc&file=modeling\_acdc.3.4.html#142268
- 20. http://en.wikipedia.org/wiki/Induction\_heating
- 21. http://www.boulder.nist.gov/div853/lead\_free/part2.html#%202.2.4.
- 22. http://en.wikipedia.org/wiki/Bismuth
- 23. http://en.wikipedia.org/wiki/Nitrogen

# Exercises

- 1. Build, mesh, and solve the COMSOL 2D resistive heating model problem presented in this chapter.
- 2. Build, mesh, and solve the first variation of the 2D resistive heating model problem presented in this chapter.
- 3. Build, mesh, and solve the second variation of the 2D resistive heating model problem presented in this chapter.

- 4. Build, mesh, and solve the 2D axisymmetric inductive heating model presented in this chapter.
- 5. Build, mesh, and solve the first variation of the 2D axisymmetric inductive heating model presented in this chapter.
- 6. Build, mesh, and solve the second variation of the 2D axisymmetric inductive heating model presented in this chapter.
- 7. Explore other materials as applied in the 2D resistive heating models.
- 8. Explore other heater geometries similar to those seen in the 2D resistive heating models.
- Explore how a change of the gas (e.g., N<sub>2</sub> → He) modifies the behavior of the 2D axisymmetric inductive heating model.
- 10. Explore how changes in the crucible geometry affect the heating rate of the 2D axisymmetric inductive heating model.

# 7

# 2D Complex Mixed-Mode Modeling

# In This Chapter

2D Complex Mixed-Mode Guidelines for New COMSOL® Multiphysics® Modelers 2D Complex Mixed-Mode Modeling Considerations 2D Coordinate System Electrical Impedance Theory 2D Electric Impedance Sensor Model: Basic Basic 2D Electric Impedance Sensor Model: Summary and Conclusions 2D Electric Impedance Sensor Model: Advanced 2D Electric Impedance Sensor Models: Summary and Conclusions Generator and Power Distribution Basics 2D AC Generators: Static and Transient 2D AC Generator Model (2D\_ACG\_1): Static 2D AC Generator Model (2D\_ACG\_2): Transient 2D AC Generators, Static and Transient Models: Summary and Conclusions 2D AC Generator: Sector-Static and Transient 2D AC Generator Sector Model (2D\_ACGS\_1): Static 2D AC Generators, Static Sector Model: Summary and Conclusions 2D AC Generator Sector Model (2D\_ACGS\_2): Transient 2D AC Generators, Static and Transient Models: Summary and Conclusions

# 2D Complex Mixed-Mode Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers

# 2D Complex Mixed-Mode Modeling Considerations

In this chapter, the basic material on 2D modeling presented in Chapters 4, 5, and 6 will be utilized and expanded. In the earlier chapters, models were built and solved using static, quasi-static, and transient methods. In this chapter, all of those methods of solution will be employed. The physics of transient models is intrinsically more difficult

than that for either the static or quasi-static models. Transient models require a firmer understanding of the underlying physical principles being modeled and a more complete (better) characterization of the materials employed in the model.

NOTE In transient or time-dependent (e.g., dynamic, unsteady) models, at least one of the dependent variables changes as a function of time.

The 2D models in this chapter implicitly assume, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables of interest are homogeneous, isotropic, or constant, unless otherwise specified (e.g., time dependent), throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model.

The three models presented in this chapter—the 2D electric impedance sensor model, the 2D generator model, and the 2D generator sector model—are developed using the AC/DC Module. Each of these three models introduces the modeler to different modeling aspects in the employment of the AC/DC Module to explore a range of similar design, test, and engineering problems.

Electrical resistance tomography is a sensing technology that applies currents, measures the resulting voltages on the surface of a body (e.g., inanimate, animate) and infers impedances from those data. This technology was developed independently in several diverse areas of study (geophysics,<sup>1</sup> industrial process imaging,<sup>2</sup> and medical imaging,<sup>3</sup> to name a few). As a result, substantially the same technology has come to be known by different names in the literature (e.g., electrical resistivity tomography, electrical resistance tomography, electrical impedance tomography). This technology is widely used in the previously mentioned areas and is one of the most promising noninvasive measurement techniques available.

The 2D electric impedance sensor models, both basic and advanced, employ the high-frequency currents (1 MHz alternating currents  $AC^4$ ). These currents are applied to the material of the modeled body to locate volumes that differ in impedance from the impedance of the bulk material by monitoring the local impedance. The basic 2D electric impedance sensor model detects the location of a fixed volume difference area. The advanced 2D electric impedance sensor model detects the location of a fluctuating difference volume, such as might be seen in a medical application measuring lung function. 2D electric impedance tomography research is exploring the application of this measurement technology to the detection of breast cancer,<sup>5</sup> lung function,<sup>6</sup> brain function,<sup>7</sup> and numerous other areas.

The 2D generator model introduces the modeler to rotary motion and the conversion of mechanical energy to electrical energy.<sup>8</sup> The 2D generator sector model employs symmetry to reduce the geometric difficulty of building the generator model and adds an ordinary differential equation<sup>9</sup> to handle the mechanical dynamics and calculate the torque caused by magnetic forces.

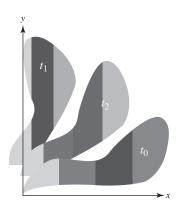


FIGURE 7.1 2D coordinate system, plus time

# **2D Coordinate System**

In a steady-state solution to a 2D model, parameters can vary only as a function of position in space (x) and space (y) coordinates. Such a 2D model represents the parametric condition of the model in a time-independent mode (quasi-static). In a transient solution model, parameters can vary both by position in space (x), and space (y) and in time (t); see Figure 7.1.

The transient solution model is essentially a sequential collection of (quasi-static) solutions, except that one or more of the dependent variables [f(x, y, t)] has changed with time. The space coordinates (x) and (y) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values  $(x_{\min} \le x \le x_{\max})$  and  $(y_{\min} \le y \le y_{\max})$ . The time coordinate (t) represents the range of values  $(t_{\min} \le t \le t_{\max})$  from the beginning of the observation period  $(t_{\min})$  to the end of the observation period  $(t_{\max})$ .

#### Electrical Impedance Theory

The concept of electrical impedance,<sup>10</sup> as used in alternating current (AC) theory, is an expansion of the basic concept of resistance as exemplified by Ohm's law,<sup>11</sup> in direct current theory.

NOTE Ohm's law was discovered by Georg Ohm and published in 1827:

$$I = \frac{V}{R} \tag{7.1}$$

where I = current in amperes (A) V = voltage (electromotive force) in volts (V)R = resistance in ohms In AC theory, both voltage (V) and current (I) alternate periodically as a function of time. Typically, the alternating behavior—frequency (f)—of the voltage and current are separately represented either as a single sinusoidal wave or as a sum of several sinusoidal waves.

**NOTE** The analysis of complex waveforms is handled by Fourier analysis.<sup>12</sup> That topic will not be presented here. However, modelers are encouraged to expand their technological horizons by exploring the subject of waveform analysis further.

In this case, for clarity, the exploration of the concept of impedance will be confined to single frequency analysis. The concept of impedance was developed and named by Oliver Heaviside<sup>12</sup> in 1886. Impedance was reformulated in the currently used complex number formulation by Arthur E. Kennelley<sup>13</sup> in 1893.

The first factor that needs to be considered, when expanding modeling calculations from the DC realm [frequency equals zero (f = 0)] to the AC realm [frequency greater than zero (f > 0)], is that the resistance (R) maps into the impedance (Z) as follows:<sup>14</sup>

$$Z = R + j\left(\omega L - \frac{1}{\omega C}\right) = R + jX = (R^2 + X^2)^{1/2} e^{j\tan^{-1}(X/R)}$$
(7.2)

where

Ε

Z = complex impedance (A) R = resistance (ohm)  $j = (-1)^{1/2}$  $\omega = 2\pi f$  = angular frequency<sup>15</sup>

 $X = \text{reactance (ohm)}^{16}$ 

L = inductance (henry)

C =capacitance (farad)

The relative vector-phase relationship of an AC voltage applied to a simple series circuit containing resistance, inductance, and capacitance is shown in Figure 7.2.

$$I*\omega*L = I*X_{L}$$

$$I*R$$

$$\downarrow \frac{-1*I}{\omega*C} = -I*X_{C}$$

$$= I*R + j*I[\omega*L - 1/(\omega*C)]$$

**FIGURE 7.2** AC voltage resistive/reactive vector phase diagram

A second factor that needs to be considered by the modeler, when modeling in the AC realm, is the skin depth  $(\delta)$ .<sup>17</sup> In any material, as a function of the complex permittivity,<sup>18</sup> electromagnetic waves (AC) will be attenuated (i.e., dissipated, turned into heat) and shifted in phase as a function of the distance (depth) traveled in that material. As an example, for a transverse electromagnetic wave propagating in the *z* direction, the voltage relationship would be expressed as follows:

$$E_x = E_0^* e^{-kz} = E_0^* e^{-\alpha z_*} e^{-j^* \beta z}$$
(7.3)

where  $E_x$  = transverse electromagnetic propagating in the *z* direction

 $E_0 =$  scalar voltage amplitude

k =complex propagation constant

$$j = (-1)^{1/2}$$

e = base of natural logarithms

 $\alpha$  = attenuation constant

 $\beta$  = wave solution constant

where  $\alpha$  is

$$\alpha = \omega * \left( \frac{\mu \varepsilon}{2} \left( 1 + \left( 1 + \left( \frac{\sigma}{\omega \varepsilon} \right)^2 \right)^{\frac{1}{2}} \right) \right)^{\frac{1}{2}}$$

and where

 $\varepsilon$  = permittivity  $\mu$  = permeability  $\omega$  = angular frequency  $\sigma$  = conductivity

For a good conductor, where  $1 << \sigma/\omega\varepsilon$ , the 1's in the preceding equation can be ignored and  $\alpha$  becomes

$$\alpha = \sqrt{\frac{\omega\mu\sigma}{2}} \tag{7.4}$$

The skin depth ( $\delta$ ) is the point at which the amplitude decreases to  $E_0 * e^{-1}$  and, therefore, is

$$\delta = \frac{1}{\alpha} \tag{7.5}$$

The first example presented in this chapter, the basic 2D electric impedance sensor model (2D\_EIS\_1 model), explores the sensing of multiple small-volume differential conductivity regions. The model is solved for a material that has a bulk conductivity of 1  $e^{-3}$  S/m. The model is implemented using the AC/DC Module Small In-Plane Currents Application Mode and solved using a Stationary solver.

In the advanced 2D electric impedance sensor model (2D\_EIS\_2 model), a new quasi-static model is built to explore a configurational change using the AC/DC Module Small In-Plane Currents Application Mode and solved using a Parametric solver.

#### 2D Electric Impedance Sensor Model: Basic

The following numerical solution model (2D\_EIS\_1) is derived from a model that was originally developed by COMSOL<sup>®</sup> as a Multiphysics<sup>®</sup> General Industrial Applications demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL Multiphysics model in the AC/DC Module Model Library.

NOTE As mentioned earlier in this chapter, knowing the skin depth ( $\delta$ ) model limitations is important. For this model, the parameters are as follows:

$$\omega = 2\pi f = 2*3.14159*1E6 = 9.9892E7 \tag{7.6}$$

$$\mu = 4\pi E - 7 [H/m] = 1.2566E - 6 [H/m]$$
(7.7)

$$\sigma = 1E - 3[S/m] \tag{7.8}$$

$$\delta = \left(\frac{2}{\omega\mu\sigma}\right)^{\frac{1}{2}} = \left(\frac{2}{9.9892E7*1.2566E - 6*1E - 3}\right)$$
$$= 15.933 \,[\text{m}]$$
(7.9)

Because the largest dimensions in the model are approximately 1 meter (m), the skin depth ( $\delta$ ) consideration will pose no problem and will not have to be factored into the calculation. (This is a first principles observation.)

To start building the 2D\_EIS\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select AC/DC Module > Quasi-Statics, Electric > In-Plane Electric Currents > Time-harmonic analysis. See Figure 7.3. Click OK.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 7.1; see also Figure 7.4. Click OK.

When building a model, it is usually best to consolidate the calculational parameters (e.g., constants, scalar expressions) in a small number of appropriate, convenient locations (e.g., a Constants file, a Scalar Expressions file) so that they are easy to find and modify as needed. Because the settings in the Constants and Scalar

00	Model Navigato	or
New	Model Library User Models	s Open Settings
Space dimension:	2D 🗘	
<ul> <li>☐ Time-</li> <li>☐ Transi</li> <li>► ☐ Small-</li> <li>► ☐ Quasi-Statics</li> <li>► ☐ Quasi-Statics</li> <li>► ☐ Quasi-Statics</li> <li>► ☐ Rotating Mach</li> </ul>	, Electric lectric Currents harmonic analysis ent analysis signal analysis , Magnetic , Electromagnetic	Description: Quasi-statics of conducting and dielectric materials with small current in the plane and a negligible coupling between the electric and magnetic fields. Time-harmonic analysis.
Application mode name:	emqvw	
Element:	Lagrange - Quadra 🗘	Multiphysics

FIGURE 7.3 2D\_EIS\_1 Model Navigator setup

Expressions edit windows can be imported and exported as text files, the appropriate text file can be modified in a text editor and then reimported into the correct Constants or Scalar Expressions edit window.

	Table 7.1	Constants	<b>Edit Window</b>
--	-----------	-----------	--------------------

Name	Expression	Description
sig_bulk	1[mS/m]	Bulk conductivity
eps_r_bulk	5	Relative permittivity in bulk
x_0	-0.35[m]	x position of cavity center
y_0	-0.15[m]	y position of cavity center
r_0	0.09[m]	Cavity radius
x_1	0.0[m]	x position of cavity center
y_1	-0.3[m]	y position of cavity center
r_1	0.12[m]	Cavity radius
x_2	0.35[m]	x position of cavity center
y_2	-0.15[m]	y position of cavity center
r_2	0.06[m]	Cavity radius

Name	Expression	Value	Description
sig_bulk	1[mS/m]	0.001[S/m]	Bulk conductivity
eps_r_bulk	5	5	Relative permittivity in bulk
x_0	-0.35[m]	-0.35[m]	x position of cavity center
y_0	-0.15[m]	-0.15[m]	y position of cavity center
r_0	0.09[m]	0.09[m]	Cavity radius
x_1	0.0[m]	0[m]	x position of cavity center
y_1	-0.3[m]	-0.3[m]	y position of cavity center
r_1	0.12[m]	0.12[m]	Cavity radius
x_2	0.35[m]	0.35[m]	x position of cavity center
y_2	-0.15[m]	-0.15[m]	y position of cavity center
r_2	0.06[m]	0.06[m]	Cavity radius

FIGURE 7.4 2D\_EIS\_1 model Constants edit window

Select File > Save As. Enter 2D\_EIS\_1 in the Save As edit window. See Figure 7.5. Click the Save button.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 1.0 and a height of 0.5. Select "Base: Corner" and set x equal to -0.5 and y equal to -0.5 in the Rectangle edit window. See Figure 7.6.

Click OK, and then click the Zoom Extents button. See Figure 7.7.

Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter x: -0.01 space 0.01, y: 0 space 0. See Figure 7.8.

Click OK. See Figure 7.9.

9_Models	\$
Name	Date Modified
EIS_2b.mph	Sunday, March 1, 2009 11:07 AM
EIS_2b_Constants.txt	Thursday, March 5, 2009 2:18 PM
EIS_2b_SV.txt	Thursday, March 5, 2009 2:14 PM
EIT_2a.mph	Monday, March 2, 2009 3:39 PM
EIT_2a_Constants.txt	Thursday, March 5, 2009 2:16 PM
EIT_2a_SV.txt	Thursday, March 5, 2009 2:16 PM
Generator_2D_1.mph	Thursday, February 19, 2009 7:04 AM
Generator_2D_1.mphbin	Thursday, February 19, 2009 5:04 PM
Generator_2D_1_Geometry.dxf	Wednesday, February 18, 2009 7:36 PM
Generator 2D 2.mph	Thursday, February 19, 2009 7:32 AM

FIGURE 7.5 2D\_EIS\_1 model Save As edit window

Size		Rotat	ion angle	
Width:	1.0	α:	0	(degrees
Height:	0.5			
Position				
Base:	Corner	\$ Style:	Solid	¢
x:	-0.5	Name:	R1	
	-0.5			

**FIGURE 7.6** 2D\_EIS\_1 model Rectangle edit window

**NOTE** The rectangle is the 2D representation of a 3D rectangular body in cross section. The points are added to the 2D rectangle to define the location of the electrode (between the points) on the boundary of the rectangle.

Coord	inates	
x: -0	.01 0.01	ОК
y: 0	0	Cancel
Name:	PT1	Apply
		Help

FIGURE 7.8 2D\_EIS\_1 model Point edit window

## **Physics Settings: Scalar Expressions**

Using the menu bar, select Options > Expressions > Scalar Expressions. In the Scalar Expressions edit window, enter the information shown in Table 7.2; see Figure 7.10. Click OK.

## **Physics Settings: Scalar Variables**

Select Physics > Scalar Variables. Enter 1e6 in the nu\_emqvw Application Scalar Variables edit window. See Figure 7.11. Click OK.

Name	Expression	Description
sigma_0	sig_bulk*(((x-x_0)^2+(y-y_0)^2)>r_0^2)	Conductivity bulk
epsilon_r_0	$1 + (eps_r_bulk-1)*(((x-x_0)^2+(y-y_0)^2)>r_0^2)$	Permittivity relative
sigma_1	sig_bulk*(((x-x_1)^2+(y-y_1)^2)>r_1^2)	Conductivity bulk
epsilon_r_1	$1 + (eps_r_bulk-1)^*(((x-x_1)^2+(y-y_1)^2)>r_2^2)$	Permittivity relative
sigma_2	sig_bulk*(((x-x_2)^2+(y-y_2)^2)>r_1^2)	Conductivity bulk
epsilon_r_2	$1+(eps_r_bulk-1)*(((x-x_2)^2+(y-y_2)^2)>r_2^2)$	Permittivity relative
sigma_tot	(sigma_0+sigma_1+sigma_2)/3	Conductivity total
epsilon_r_tot	(epsilon_r_0+epsilon_r_1+epsilon_r_2)/3	Permittivity total

# Table 7.2 Scalar Expressions Edit Window

Name	Expression	Unit	Description
sigma_0	sig_bulk*(((x-x_0)^2+(y-y_0)^2)>r_0^2)	S/m	Conductivity bulk
epsilon_r_0	$1+(eps_r_bulk-1)*(((x-x_0)^2+(y-y_0)^2)>r_0^2)$		Permittivity relative
sigma_1	sig_bulk*(((x-x_1)^2+(y-y_1)^2)>r_1^2)	S/m	Conductivity bulk
epsilon_r_1	$1+(eps_r_bulk-1)^*(((x-x_1)^2+(y-y_1)^2)>r_2^2)$		Permittivity relative
sigma_2	sig_bulk*(((x-x_2)^2+(y-y_2)^2)>r_1^2)	S/m	Conductivity bulk
epsilon_r_2	$1+(eps_r_bulk-1)*(((x-x_2)^2+(y-y_2)^2)>r_2^2)$		Permittivity relative
sigma_tot	(sigma_0+sigma_1+sigma_2)/3	S/m	Conductivity total
epsilon_r_tot	(epsilon_r_0+epsilon_r_1+epsilon_r_2)/3		Permittivity total
c	(Help) (	Apply	Cancel OK

**FIGURE 7.10** 2D\_EIS\_1 model Scalar Expressions edit window

Name	Expression	Unit	Description
epsilon0_emqvw	8.854187817e-12	F/m	Permittivity of vacuum
mu0_emqvw	4*pi*1e-7	H/m	Permeability of vacuum
nu_emqvw	1e6	Hz	Frequency

**FIGURE 7.11** 2D\_EIS\_1 model Application Scalar Variables edit window

Table 7.3	Subdomain Edit Windows

Name	Expression	Description
$\sigma$ (isotropic)	sigma_tot	Electrical conductivity
εr (isotropic)	epsilon_r_tot	Relative permittivity

#### Physics Subdomain Settings: In-Plane Electric Currents (emqvw)

Having established the geometry for the 2D\_EIS\_1 model of a block with an electrode, the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). In the Subdomain edit windows, enter the information shown in Table 7.3.

Click the D =  $\varepsilon_0 \varepsilon_r E$  radio button. See Figure 7.12. Click OK.

#### **Physics Boundary Settings: In-Plane Electric Currents (emqvw)**

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 7.4. See Figures 7.13, 7.14, and 7.15.

Subdomains Groups	Phys	ics Infinite Elements Force	s Init Element Color
ubdomain selection	Material proper	ties and sources	
	Library materi Constitutive r $\textcircled{D} = \varepsilon_0 \varepsilon_F E$ Quantity J <sup>e</sup>	elation $O D = \epsilon_0 E + P$ Value/Expression	ad $D = \varepsilon_0 \varepsilon_r E + D_r$ Unit Description
	Q <sub>j</sub>	0 0	A/m <sup>2</sup> External current density A/m <sup>3</sup> Current source m Thickness
Foup:	σ ε <sub>r</sub>	sigma_tot epsilon_r_tot	S/m Electric conductivity 1 Relative permittivity

**FIGURE 7.12** 2D\_EIS\_1 model Subdomain Settings edit window

# Table 7.4 Boundary Settings – In-Plane Electric Currents (emqvw) Edit Window

Boundary	Boundary Condition	Figure Number
1, 2, 6	Ground	7.13
3, 5	Electrical insulation	7.14
4	Port	7.15

Boundaries Groups	Conditions Port Color/Style
Roundary selection	Boundary sources and constraints Boundary condition: Ground
Group: 🔹 🗘	

FIGURE 7.13 2D\_EIS\_1 model Boundary Settings (1, 2, 6) edit window

Boundaries Groups	Conditions Port Color/Style
loundary selection	-Boundary sources and constraints
1	Boundary condition: Electric insulation
3	
4	
5	
iroup:	
Select by group	

FIGURE 7.14 2D\_EIS\_1 model Boundary Settings (3, 5) edit window

quation = Y + V	
Boundaries Groups Boundary selection 1 2 3 6 6	Conditions Port Color/Style Boundary sources and constraints Boundary condition: Port
Group: 2	

**FIGURE 7.15** 2D\_EIS\_1 model Boundary Settings (4) edit window

Select the Port tab. Check Use port as input. Select "Fixed current density" from the input property pull-down list. See Figure 7.16. Click OK.

#### **Mesh Generation**

From the toolbar, select Mesh > Free Mesh Parameters > Global. Select Predefined mesh sizes > Normal (from the pull-down list). Select "Custom mesh size." Enter 0.01 in the Maximum element size edit window. See Figure 7.17.

Click the Remesh button, and then click OK. See Figure 7.18.

7 Equation JV / area = Z · I	Boundary Settings – In-Plane Electric Currents (emqvw)
Boundaries     Grou       Roundary selection     1       12     3       4     5       6     6       Group:	Port definition Port number: 1 Use port as input Input property: Fixed current density \$
	(Help) (Cancel) (Cancel)

FIGURE 7.16 2D\_EIS\_1 model Boundary Settings (Port) edit window

		_
O Predefined mesh sizes:	Normal	Can
Custom mesh size		Ap
Maximum element size:	0.01	
Maximum element size scaling facto	pr: 1	(He
Element growth rate:	1.3	
Mesh curvature factor:	0.3	
Mesh curvature cutoff:	0.001	
Resolution of narrow regions:	1	
Optimize quality		
Refinement method: Regular	3	
vennement metriod. Kegulai 🗣	3	

**FIGURE 7.17** 2D\_EIS\_1 model Free Mesh Parameters edit window

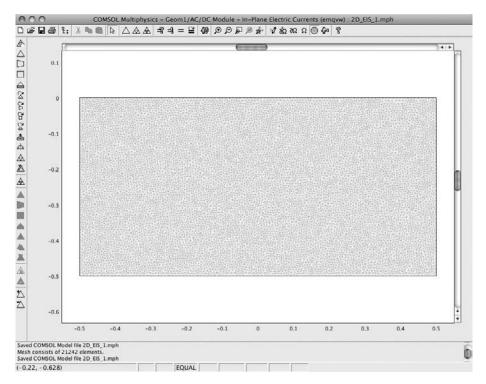


FIGURE 7.18 2D\_EIS\_1 model mesh

**FIGURE 7.19** 2D\_EIS\_1 model Solver Parameters edit window

#### Solving the 2D\_EIS\_1 Model

Using the menu bar, select Solve > Solver Parameters.

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Select "Stationary Solver." See Figure 7.19. Click OK. Using the menu bar, select Solve > Solve Problem. See Figure 7.20.

#### Postprocessing and Visualization

The default plot shows a surface plot of the electric potential (V) distribution in volts. To visualize the detected regions of differential conductivity, the plot parameters will need to be modified.

FIGURE 7.20 2D\_EIS\_1 model solution

Select Postprocessing > Plot Parameters > Surface. Select "Total current density, norm" from the Predefined quantities pull-down list. Change the expression in the edit window from normJ\_emqvw to  $20*log10(normJ_emqvw)$ . Click the Range button. Unselect the Auto check box. Enter -35 in the Min edit window and 35 in the Max edit window; see Figure 7.21. Click OK.

Select "wave" from the Colormap pull-down list. See Figure 7.22. Click OK. See Figure 7.23.

# **Basic 2D Electric Impedance Sensor Model: Summary and Conclusions**

The basic 2D electric impedance sensor model has been built and operated. This model employs a high-frequency current (1 MHz alternating current AC) to explore the differential impedance within the body of a material in a noninvasive manner. Such 1 MHz currents are applied to the material of the modeled body to locate volumes that differ in impedance from the impedance of the bulk material by monitoring the local impedance, as shown in Figure 7.23. The basic model shows the location of three areas of fixed-volume impedance difference.

	uto
Min:	-35
Max:	35

**FIGURE 7.21** 2D\_EIS\_1 model solution, Color Range edit window

0.0	Plot I	Parameters			
General Surface	Contour	Boundary	Arrow	Principal	
🗹 Surface plot					
	Surface Data	Height Data			
			-	_	
Predefined quantities:			Ran	ge	
Expression:	20*log10(norn	nJ_emqvw)	Sm Sm	ooth	
Unit:			•		
Coloring and fill	lated		Filled	\$	
	lated	Fill style:	Filled	•	
Surface color					
• Colormap:	wave 🛟	Colors: 1024	Col	or scale	
O Uniform color: (	Color				
				-	
	DOV				
	POV	VER <mark>EN</mark> .IR			
	POV	VER <mark>EN</mark> .IR			
	POV	VER <mark>EN</mark> .IR	/		

**FIGURE 7.22** 2D\_EIS\_1 model solution Plot Parameters edit window

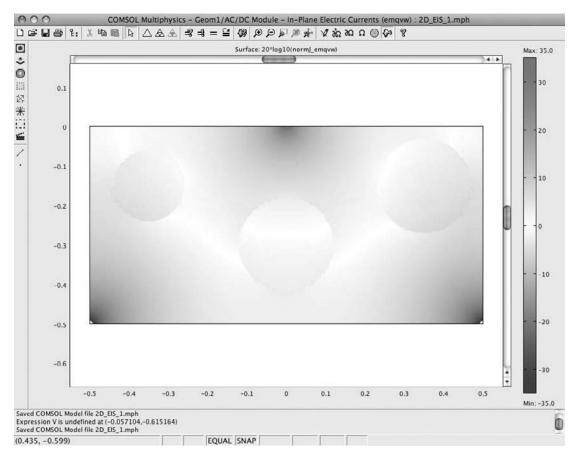


FIGURE 7.23 2D\_EIS\_1 model solution with detected areas

# 2D Electric Impedance Sensor Model: Advanced

To start building the 2D\_EIS\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select AC/DC Module > Quasi-Statics, Electric > In-Plane Electric Currents > Time-harmonic analysis. See Figure 7.24. Click OK.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 7.5; see also Figure 7.25. Click OK.

Select File > Save As. Enter 2D\_EIS\_2 in the Save As edit window. See Figure 7.26. Click the Save button.

New       Model Library       User Models       Open       Settings         Space dimension:       2D       Image: Composition Modes       Image: Composition Modes <th>y and the second /th> <th>Model Naviga</th> <th>tor</th>	y and the second	Model Naviga	tor
	Space dimension: Application Modes COMSOL Multiph COMSOL Multiph AC/DC Module Statics Quasi-Statics COMSOL Multiph Trans Statics COMSOL Multiph COMSOL Multiph Statics COMSOL Statics COMSOL Statics CO	2D ysics s, Electric Electric Currents harmonic analysis sient analysis -signal analysis s, Magnetic s, Electromagnetic hinery mal Interaction	Image: Constraint of the second se
Element: Lagrange – Quadra	Application mode name:	emqvw	

**FIGURE 7.24** 2D\_EIS\_2 Model Navigator setup

Table 7.5	<b>Constants Edit</b>	Window
-----------	-----------------------	--------

Name	Expression	Description
sig_bulk	1[mS/m]	Bulk conductivity
eps_r_bulk	5	Relative permittivity in bulk
x_0	-0.12[m]	x position of cavity center
y_0	0[m]	y position of cavity center
r_0	0.07[m]	Cavity radius
x_1	0.12[m]	x position of cavity center
y_1	0[m]	y position of cavity center
r_1	0.07[m]	Cavity radius
t_0	0	Time init

000		Constants	
Name	Expression	Value	Description
sig_bulk	1[mS/m]	0.001[S/m]	Bulk conductivity
eps_r_bulk	5	5	Relative permittivity in bulk
x_0	-0.12[m]	-0.12[m]	x position of cavity center
y_0	0[m]	0[m]	y position of cavity center
r_0	0.07[m]	0.07[m]	Cavity radius
x_1	0.12[m]	20000000 Vol 00 0 0	x position of cavity center
y_1	0[m]	0[m]	y position of cavity center
r_1	0.07[m]	0.07[m]	Cavity radius
t_0	0	0	Time init
_			4 1 6
c c c c c c c c c c c c c c c c c c c		Help	Apply Cancel OK

FIGURE 7.25 2D\_EIS\_2 model Constants edit window

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes of 0.3 and B-semiaxes of 0.15. Select "Base: Center" and set x equal to 0 and y equal to 0 in the Ellipse edit window. See Figure 7.27.

Click OK, and then click the Zoom Extents button. See Figure 7.28.

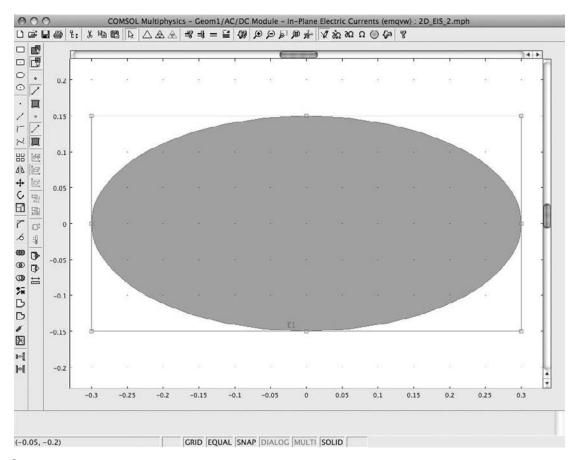
9_Models	\$
Name	Date Modified
2D_EIS_1.mph	Friday, March 6, 2009 7:54 AM
2D_EIS_2.mph	Friday, March 6, 2009 2:05 PM
EIS_2b.mph	Sunday, March 1, 2009 11:07 AM
EIS_2b_Constants.txt EIS_2b_SV.txt	Thursday, March 5, 2009 2:18 PM
EIS_2b_SV.txt	Thursday, March 5, 2009 2:14 PM
EIT_2a.mph	Monday, March 2, 2009 3:39 PM
EIT 2a Constants.txt	Thursday, March 5, 2009 2:16 PM
EIT_2a_SV.txt	Thursday, March 5, 2009 2:16 PM
Generator_2D_1.mph	Thursday, February 19, 2009 7:04 AM
Generator_2D_1.mphbin	Thursday, February 19, 2009 5:04 PM
Generator_2D_1_Geometry.dxf	Wednesday, February 18, 2009 7:36 PN
Generator_2D_2.mph	Thursday, February 19, 2009 7:32 AM
Generator_Sector_1a.mph	Friday, February 20, 2009 10:47 AM
Generator_Sector_1b.mph	Tuesday, February 24, 2009 8:08 AM
Generator_Sector_2b.mph	Tuesday, February 24, 2009 8:18 AM

FIGURE 7.26 2D\_EIS\_2 model Save As edit window

#### 480 CHAPTER 7 2D COMPLEX MIXED-MODE MODELING

Size		Rotat	ion angle	
A-semiaxes:	0.3	α:	0	(degrees)
B-semiaxes:	0.15			
Position				
Base:	Center	\$ Style:	Solid	\$
			F1	
<b>x</b> :	0	Name:	E1	

FIGURE 7.27 2D\_EIS\_2 model Ellipse edit window



**FIGURE 7.28** 2D\_EIS\_2 model rectangle

481

#### **FIGURE 7.29** 2D\_EIS\_2 model Rectangle edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter a width of 0.04 and a height of 0.001. Select "Base: Corner" and set x equal to -0.02 and y set equal to 0.15-0.001 in the Rectangle edit window. See Figure 7.29. Click OK.

**NOTE** The purpose of adding the rectangle to the ellipse is to provide a known placement location for the points that will define the edges of the electrode.

Using the menu bar, select Draw > Create Composite Object. Enter E1+R1 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 7.30.

Click OK. See Figure 7.31.

000	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: E1 R1	Set formula: E1+R1 Keep interior boundaries Repair Repair tolerance: 1.0E-4

FIGURE 7.30 2D\_EIS\_2 model Create Composite Object edit window

**FIGURE 7.31** 2D\_EIS\_2 model Create Composite Object result

Using the menu bar, select Draw > Specify Objects > Point. In the Point edit window, enter  $-0.02\ 0.02$  for x and 0.15 0.15 for y. See Figure 7.32. Click OK. See Figure 7.33.

Note: The ellipse is the 2D representation of a 3D elliptical body in cross section (e.g., similar to the cross section that might be seen in the examination of a reclining human body). The points are added to the 2D ellipse to define the location of the electrode (between the points) on the boundary of the ellipse.

00	0	Point	
Coord	inates		
x: -0	.02 0.02		ОК
y: 0.	15 0.15		Cancel
Name:	PT1		Apply
			Help

FIGURE 7.32 2D\_EIS\_2 model Point edit window

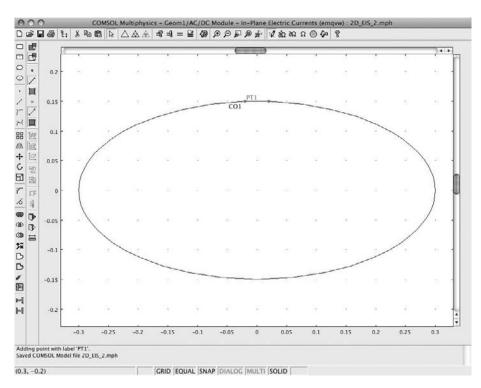


FIGURE 7.33 2D\_EIS\_2 model ellipse and points

# **Physics Settings: Scalar Expressions**

Using the menu bar, select Options > Expressions > Scalar Expressions. In the Scalar Expressions edit window, enter the information shown in Table 7.6; also see Figure 7.34. Click OK.

#### Table 7.6 Scalar Expressions Edit Window

Name	Expression	Description
sigma_0	sig_bulk*(((x-x_0)^2+(y-y_0)^2)>r_00^2)	Conductivity bulk
epsilon_r_0	$1 + (eps_r_bulk-1)*(((x-x_0)^2+(y-y_0)^2)>r_00^2)$	Permittivity relative
sigma_1	sig_bulk*(((x-x_1)^2+(y-y_1)^2)>r_01^2)	Conductivity bulk
epsilon_r_1	$1+(eps_r_bulk-1)*(((x-x_1)^2+(y-y_1)^2)>r_01^2)$	Permittivity relative
sigma_tot	(sigma_0+sigma_1)/2	Conductivity total
epsilon_r_tot	(epsilon_r_0+epsilon_r_1)/2	Permittivity total
r_00	r_0*(1.6-cos((t_0*pi)/8))/2	Radius ratio
r_01	r_1*(1.6-cos((t_0*pi)/8))/2	Radius ratio

Name	Expression	Unit	Description
sigma_0	sig_bulk*(((x-x_0)^2+(y-y_0)^2)>r_00^2)	S/m	Conductivity bulk
psilon_r_0	$1+(eps_r_bulk-1)*(((x-x_0)^2+(y-y_0)^2)>r_00^2)$		Permittivity relative
igma_1	sig_bulk*(((x-x_1)^2+(y-y_1)^2)>r_01^2)	S/m	Conductivity bulk
epsilon_r_1	$1+(eps_r_bulk-1)^*(((x-x_1)^2+(y-y_1)^2)>r_01^2)$		Permittivity relative
sigma_tot	(sigma_0+sigma_1)/2	S/m	Conductivity total
epsilon_r_tot	(epsilon_r_0+epsilon_r_1)/2		Permittivity total
r_00	r_0*(1.6-cos((t_0*pi)/8))/2	m	Radius ratio
r_01	r_1*(1.6-cos((t_0*pi)/8))/2	m	Radius ratio
c	Help	) (A	pply Cancel OK

**FIGURE 7.34** 2D\_EIS\_2 model Scalar Expressions edit window

#### **Physics Settings: Scalar Variables**

Select Physics > Scalar Variables. Enter 1e6 in the nu\_emqvw Application Scalar Variables edit window. See Figure 7.35. Click OK.

#### Physics Subdomain Settings: In-Plane Electric Currents (emqvw)

Having established the geometry for the 2D\_EIS\_2 model of an elliptical block with an electrode, the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window (the only available subdomain). In the Subdomain edit windows, enter the information shown in Table 7.7.

Click the D =  $\varepsilon_0 \varepsilon_1 E$  radio button. See Figure 7.36. Click OK.

#### Physics Boundary Settings: In-Plane Electric Currents (emqvw)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 7.8. See Figures 7.37, 7.38, and 7.39.

Name	Expression	Unit	Description		
epsilon0_emqvw	8.854187817e-12	F/m	Permittivity of vacuum		
mu0_emqvw	4*pi*1e-7	H/m	Permeability of vacuum		
nu_emqvw	1e6	Hz	Frequency		

FIGURE 7.35 2D\_EIS\_2 model Application Scalar Variables edit window

# Table 7.7 Subdomain Edit Window

Name	Expression	Description	
$\sigma$ (isotropic)	sigma_tot	Electrical conductivity	
$\boldsymbol{\epsilon}_{r}$ (isotropic)	epsilon_r_tot	Relative permittivity	

# Table 7.8 Boundary Settings – In-Plane Electric Currents (emqvw) Edit Window

Boundary	<b>Boundary Condition</b>	Figure Number	
1, 4, 5, 8	Electric Insulation	7.37	
2, 3	Port	7.38	
6, 7	Ground	7.39	

Subdomains Groups	Physi	cs Infinite Elements For	ces Init	Element Color			
Subdomain selection	Material properties and sources						
1	Library materia	Library material:					
	Constitutive re						
		$\bigcirc \mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P}$	Ο D = ε	<sub>0</sub> ε <sub>r</sub> E + D <sub>r</sub>			
	Quantity	Value/Expression	Unit	Description			
	Je	0 0	A/m <sup>2</sup>	External current density			
	Qj	0	A/m <sup>3</sup>	Current source			
	d	1	m	Thickness			
	σ	sigma_tot	S/m	Electric conductivity			
roup:	٤		1				
Select by group	r	epsilon_r_tot	-	Relative permittivity			
Active in this domain							



FIGURE 7.37 2D\_EIS\_2 model Boundary Settings (1, 4, 5, 8) edit window

quation = Y · V	
Boundaries Groups Boundary selection  1  2 3 4 5 6 6 7 8 Group:  Select by group  Interior boundaries	Conditions Port Color/Style Boundary sources and constraints Boundary condition: Port

**FIGURE 7.38** 2D\_EIS\_2 model Boundary Settings (2, 3) edit window

**FIGURE 7.39** 2D\_EIS\_2 model Boundary Settings (6, 7) edit window

Select boundaries 2 and 3 in the Boundary selection window. Select the Port tab. Check "Use port as input." Select "Fixed current density" from the input property pull-down list; see Figure 7.40. Click OK. 488

0.0	Free Mesh Parameters	
Global Subdomain	Boundary Point Advanced	ОК
O Predefined mesh sizes:	lormal 🔹	Cance
Custom mesh size		Apply
Maximum element size:	0.01	Help
Maximum element size scaling factor:	1	Снер
Element growth rate:	1.3	
Mesh curvature factor:	0.3	
Mesh curvature cutoff:	0.001	
Resolution of narrow regions:	1	
Soptimize quality		
Refinement method: Regular 🗘		
Reset to Defaults Remesh	Mesh Selected	

**FIGURE 7.41** 2D\_EIS\_2 model Free Mesh Parameters edit window

#### **Mesh Generation**

From the toolbar, select Mesh > Free Mesh Parameters > Global. Select Predefined mesh sizes > Normal (from the pull-down list). Select "Custom mesh size." Enter 0.01 in the Maximum element size edit window. See Figure 7.41.

Click the Remesh button, and then click OK. See Figure 7.42.

#### Solving the 2D\_EIS\_2 Model

Using the menu bar, select Solve > Solver Parameters. The COMSOL Multiphysics software automatically selects the Stationary solver.

NOTE The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. In this case, the modeler will need to change the chosen solver and the parametric settings.

Select "Parametric." Enter  $t_0$  in the Parameter name edit window. Enter linspace(0.0,32,32) in the Parameter values edit window. See Figure 7.43. Click OK.

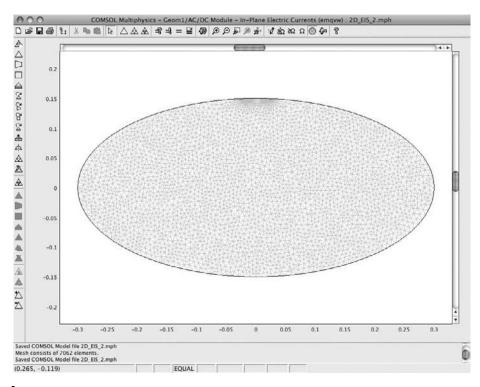


FIGURE 7.42 2D\_EIS\_2 model mesh

**NOTE** The linspace (0.0,32,32) command causes the solver to step the value of t\_0 32 times between 0.0 and 32. For later versions of COMSOL Multiphysics software use the command range (0,32/32,32) in place of the linspace command.

In COMSOL Multiphysics software version 3.5a, the linspace  $(x_1,x_2,x_3)$  function (where  $x_1 =$  start value,  $x_2 =$  end value, and  $x_3 =$  number of intervals) has been changed to the range  $(y_1,y_2,y_3)$  function (where  $y_1 =$  start value,  $y_2 =$  interval width, and  $y_3 =$  end value).

Using the menu bar, Select Solve > Solve Problem. See Figure 7.44.

#### **Postprocessing and Visualization**

The default plot shows a surface plot of the electric potential (V) distribution in volts. To visualize the detected regions of differential conductivity, the plot parameters will need to be modified.

Select Postprocessing > Plot Parameters > Surface. Select "Total current density, norm" from the Predefined quantities pull-down list. Change the expression in the edit window from normJ\_emqvw to 20\*log10(normJ\_emqvw). Click the Range button.

Analysis:	General Pa	rametric	Stationary	Adaptive Advar	L hear
Time-harmonic, electric currents	Parameter name:	rametric	t_0	Auguve Augu	iceu
Solver. Stationary Time dependent	Parameter values:		linspace(0.	0,32,32)	
Eigenvalue Parametric Stationary segregated Parametric segregated	Linear system solver: Preconditioner:	Direct (UM	IFPACK)	¢) \$	
Adaptive mesh refinement	Matrix symmetry:	Automatic		(5	etting
		Help	) Apply	Cancel	(

FIGURE 7.43 2D\_EIS\_2 model Solver Parameters edit window

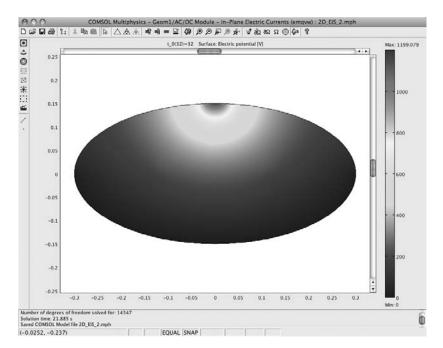


FIGURE 7.44 2D\_EIS\_2 model solution

A	uto
Min:	-35
Max:	35

FIGURE 7.45 2D\_EIS\_2 model solution Color Range edit window

Unselect the Auto check box. Enter -35 in the Min edit window and 35 in the Max edit window. See Figure 7.45. Click OK.

Select "wave" from the Colormap pull-down list. See Figure 7.46.

Click OK. See Figure 7.47.

		-	Parameters		
General	Surface	Contour	Boundary	Arrow	Principal
Surface p	lot				
		Surface Data	Height Data		
Predefined	quantities:	[		Ran	ge
Expression		20°log10(norn		S Sm	
Unit:	10	20 log10(iloin	i)_ciiiqiii)	- 	John
oloring and	fill				
Coloring:	Interpo	lated 🛟	Fill style:	Filled	\$
urface color	-				
		wave 0	Colors: 1024		orscale
• Colorm	ap: (	wave 🗘	Colors: 1024	Col	or scale
Colorm	ap: (	wave	Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale
	ap: (		Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale
- T.	ap: (		Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale
• Colorm	ap: (		Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale
Colorm	ap: (		Colors: 1024	Col	or scale

FIGURE 7.46 2D\_EIS\_2 model solution Plot Parameters edit window

**FIGURE 7.47** 2D\_EIS\_2 model solution with detected areas

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 7.48.

Click the Start Animation button. See Figure 7.49.

Alternatively, you can play the file Movie7\_EIS\_2.avi that was supplied with this book.

#### 2D Electric Impedance Sensor Models: Summary and Conclusions

In this part of the chapter two 2D electric impedance sensor models, basic and advanced, were built and operated. These models employ a high-frequency current, (1 MHz alternating current AC) to explore the differential impedance within a body of material in a noninvasive manner. Such currents may be applied to the material of the modeled body to locate volumes that differ in impedance from the impedance of the bulk material by monitoring the local impedance. The basic 2D electric impedance sensor model shows the location of a fixed-volume impedance difference. The advanced 2D electric impedance sensor model shows the location of a fluctuating difference volume,

Streamline	Particle Tracing	Max/Min	Deform	Animat
Movie settings		Solutions t	o use	
File type.	AVI \$	Select via:		\$
Width (in pixels):	640	0	_	
		1.032258		
Height (in pixels):	480	2.064516 3.096774		Ű,
Frames per second:	10	4.129032		
	Advanced	5.16129 6.193548		
-Static / Eigenfunctio		7.225806		
	Full harmonic	8.258065 9.290323		+
Cycle type:		Times:		
Number of frames:	11			

FIGURE 7.48 2D\_EIS\_2 model Plot Parameters window

as might be seen in a medical application measuring lung function. 2D electric impedance tomography research is currently exploring the application of this impedancesensing measurement technology to the detection of breast cancer, lung function, brain function, and numerous other areas.

The new concepts introduced in this section of Chapter 7 are complex AC theory, complex impedance, and skin depth.

# Generator and Power Distribution Basics

Shortly after Georg Ohm discovered and published Ohm's law in 1827, Michael Faraday<sup>19</sup> discovered and published the basic operating principle of both DC and AC generators, known as electromagnetic induction.<sup>20</sup>

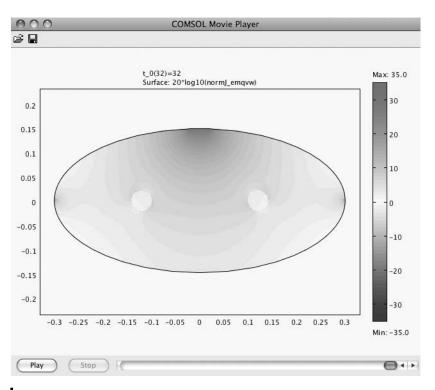


FIGURE 7.49 2D\_EIS\_2 model animation, final frame

NOTE Ohm's law is:

$$I = \frac{E}{R} \tag{7.10}$$

where I = current in amperes (A)E = electromotive force in volts (V)R = resistance in ohms

Thomas Alva Edison<sup>21</sup> took the initial lead in the development and commercialization of DC electrical power generation and distribution systems. During approximately the same time period, George Westinghouse<sup>22</sup> and Nikola Tesla<sup>23</sup> were developing and commercializing AC electrical power generation and distribution systems. The resulting intense industrial competition led to what has been called the "War of Currents."<sup>24</sup>

Independent of all the rhetoric exchanged during the "War of Currents," two fundamental physical factors would mandate that the ultimate winner of this intense contest was to be AC power, even before the first battle was fought. Those basic physical factors were (1) the intrinsic nature of DC (steady) and (2) Joule's first law.<sup>25</sup> It is the intrinsic nature of DC that it is, by definition, a steady, fixed voltage. Thus it is by definition not transformable to a different voltage. For DC to be transformable, it must be converted to AC, transformed, and then converted back to DC. Therefore, when different voltages were needed, different dynamos (DC generators) had to be built to generate the different voltage.

Joule's first law, published in 1841, states that the power dissipated in a resistor can be expressed as follows:

$$P = I^{2*}R \tag{7.11}$$

By Ohm's law

$$R = \frac{E}{I} \tag{7.12}$$

Thus

$$P = I^{2*}\left(\frac{E}{I}\right) = I^*E$$

It is the intrinsic nature of AC that both the current and the voltage normally fluctuate. Hence, AC can be converted (transformed) from one voltage to a different voltage.<sup>26</sup> Because AC can be transformed to the first order (assuming no systemic losses) and assuming conservation of energy (no sources or sinks in the transformation process), then

**NOTE** The term "to the first order" means formulating the most basic mathematical statement of the problem, without secondary corrections.

Primary input power:  $P_{\rm P} = I_{\rm P} * E_{\rm P}$  (7.13)

Secondary output power:  $P_{\rm S} = I_{\rm S} * E_{\rm S}$  (7.14)

Conservation of energy: 
$$P_{\rm P} = P_{\rm S}$$
 (7.15)

where  $P_{\rm P}$  = power input to the transformer in watts (W)  $E_{\rm P}$  = electromotive force input to the transformer in volts (V)  $I_{\rm P}$  = current input to the transformer in amperes (A)  $P_{\rm S}$  = power output from the transformer in watts (W)  $E_{\rm S}$  = electromotive force output from the transformer in volts (V)  $I_{\rm S}$  = current output from the transformer in amperes (A) Assuming a lossless transformer,

$$E_{\rm S} = \frac{N_{\rm S}}{N_{\rm P}} * E_{\rm P} \tag{7.16}$$

where  $E_{\rm p}$  = electromotive force at the primary input of the transformer in volts (V)

- $E_{\rm S}$  = electromotive force at the secondary output of the transformer in volts (V)
- $N_{\rm P}$  = number of turns in the primary winding of the transformer
- $N_{\rm S}$  = number of turns in the secondary winding of the transformer

Because the input power equals the output power, the current (I) and the electromotive force (E) have an inverse relationship. As E goes up, I goes down, in a direct proportionality.

Thus

$$I_{\rm S} = \frac{N_{\rm P}}{N_{\rm S}} * I_{\rm P} \tag{7.17}$$

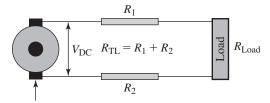
where  $I_{\rm P}$  = current in the primary winding of the transformer in amperes (A)  $I_{\rm S}$  = current in the secondary winding of the transformer in amperes (A)  $N_{\rm P}$  = number of turns in the primary winding of the transformer

 $N_{\rm S}$  = number of turns in the secondary winding of the transformer

In the case of a DC distribution system, as shown in Figure 7.50, the transmission line has losses.

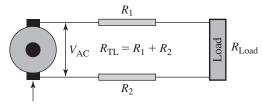
In the case of DC power, all of the load current flows through the transmission line resistance and generates thermal losses:

$$P_{\rm DCLoss} = I_{\rm DCLoad}^2 R_{\rm TL} \tag{7.18}$$



DC Generator

**FIGURE 7.50** DC power transmission system



AC Generator

FIGURE 7.51 Untransformed AC power transmission system

If the AC power transmission system were configured in the same manner as a DC power transmission system, then the systems would be equivalent. See Figure 7.51.

However, when transformers are employed, the physics changes significantly. See Figure 7.52.

In the case of transformed AC power, for example, raising the AC electromotive force (EMF) of the transmission line by transforming the EMF by a factor of 100 causes the current in the transmission line to be lowered by a factor of 100:

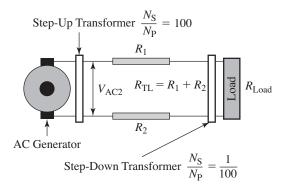
$$\frac{E_{\rm S}}{E_{\rm P}} = \frac{N_{\rm S}}{N_{\rm P}} = 100 \tag{7.19}$$

Then

$$I_{\text{ACTL}} = \frac{1}{100} * I_{\text{ACLoad}} \tag{7.20}$$

and

$$P_{\rm ACLoss} = \left(\frac{I_{\rm ACLoad}}{100}\right)^2 * R_{\rm TL}$$
(7.21)



**FIGURE 7.52** Transformed AC power transmission system

498

where

 $P_{\rm DS}$  = dissipated power in watts (W)

- $E_{\rm p}$  = electromotive force at the primary input of the transformer in volts (V)
- $E_{\rm S}$  = electromotive force at the secondary output of the transformer in volts (V)
- $N_{\rm P}$  = number of turns in the primary winding of the transformer
- $N_{\rm S}$  = number of turns in the secondary winding of the transformer
- $I_{\rm AC \, TL}$  = transmission line current

 $I_{AC Load} = AC load current$ 

 $P_{\rm AC Loss}$  = transmission line current

 $R_{\rm TL}$  = transmission line resistance

Assuming that the load currents are equivalent

$$I_{\rm DCLoad} = I_{\rm ACLoad} \tag{7.22}$$

then the relative transmission line power loss for AC compared to DC is

$$\frac{P_{\rm ACLoss}}{P_{\rm DCLoss}} = \left(\frac{1}{100}\right)^2 = 1*10^{-4}$$
(7.23)

Thus transformed AC became the obvious choice for power line transmission, based both on versatility and on reduced power losses.

## 2D AC Generators: Static and Transient

In the following subsections of Chapter 7, models are developed that provide an analysis of the rotating machines (AC generators) that convert mechanical energy into electrical energy (AC power). The generation of AC power is accomplished through the application of Faraday's law of induction. In the following models, a magnetic vector potential ( $\mathbf{A}$ ) is employed that has only a *z* component.

Rotation is modeled using the deformed Mesh Application Mode (ALE). The rotor and the stator are drawn separately and then combined as an assembly.<sup>27</sup>

The materials employed in this model are high-energy samarium-cobalt magnets with nonlinear soft iron pole pieces.

NOTE A pole piece is the magnetically soft (easily altered) material that is inserted in the magnet circuit to guide the path of the magnetic flux to a desired location.

## 2D AC Generator Model (2D\_ACG\_1): Static

The numerical solution model developed in this section (2D\_ACG\_1) is derived from a model that was originally developed by COMSOL as an AC/DC Module Motors and Drives Library Model. Here, the 2D generator model (2D\_ACG\_1) will be built as a

New	Model Library User Mode	ls Open Settings
Space dimension:	2D 🛟	AC/DC Module
▼ 📄 Rotating Macl	, Electric , Magnetic , Electromagnetic ninery terpendicular Currents nal Interaction	Description: A predefined combination of application modes for rotating electro-mechanical devices. Combines Perpendicular Induction Currents with Moving Mesh for modeling of rotating electro-mechanical applications such as electric motors.
	Lagrange - Quadra	Multiphysics

**FIGURE 7.53** 2D\_ACG\_1 Model Navigator setup

static (stationary) model. In the next subsection, the static model will be used as the starting point for the transient (rotating) model (2D\_ACG\_2).

To start building the 2D\_ACG\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select AC/DC Module > Rotating Machinery > Rotating Perpendicular Currents. See Figure 7.53. Click OK.

NOTE The Model Navigator shows two names in the Application Mode name edit window: emqa and ale.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 7.9; see also Figure 7.54. Click OK.

When the modeler enters the constant t, the text will become red in color to indicate that the modeler has entered a reserved variable name (t =time, in COMSOL Multiphysics software). However, because the first model is stationary, t needs to be assigned a value (in this case, 0). Once the transient model is built, the transient solver will override the assigned value during the solving process.

Name	Expression	Description
t	0[s]	Time equals zero (stationary solution)
rpm	60[1/min]	Revolutions per minute
А	pi*(0.02[m])^2	Area, stator wire
L	0.4[m]	Length, generator
NN	1	Stator winding turns

Table 7.9	Constants	Edit Window
14510 710	oonotanto	

t 0[s] 0[s] Time equals zero (stationary solution) rpm 60[1/min] 1[1/s] Revolutions per minute A pi*(0.02[m])^2 0.001257[m <sup>2</sup> Area, stator wire L 0.4[m] 0.4[m] Length, generator NN 1 1 1 Stator winding turns	Name	Expression	Value	Description	
rpm         60[1/min]         1[1/s]         Revolutions per minute           A         pi*(0.02[m])^2         0.001257[m²         Area, stator wire           L         0.4[m]         0.4[m]         Length, generator					
A pi*(0.02[m])^2 0.001257[m <sup>2</sup> Area, stator wire L 0.4[m] 0.4[m] Length, generator	rpm				1
L 0.4[m] 0.4[m] Length, generator		pi*(0.02[m])^2	0.001257(m <sup>2</sup>	Area, stator wire	1
NN 1 1 Stator winding turns	L	0.4[m]			
	IN 1	1 1	1	Stator winding turns	
( ) ( ) ( ) ( ) ( ) ( ) ( ) ( ) ( ) ( )					1
(					1
	=			) + )	•

FIGURE 7.54 2D\_ACG\_1 model Constants edit window

Select File > Save As. Enter  $2D_ACG_1$  in the Save As edit window. See Figure 7.55. Click the Save button.

## **Generator Geometry**

**NOTE** The 2D\_ACG\_1 geometry is very complex. Be sure to follow the steps carefully and in sequence. After completion of all of the geometry and assembly steps, there should be a total of 152 boundaries.

Using the menu bar, select Draw > Specify Objects > Circle and create the circles indicated in Table 7.10.

Select File > Save. See Figure 7.56.

Name	Radius	Base	X	Y	Rotation Angle
C1	0.3	Center	0	0	22.5
C2	0.235	Center	0	0	0
C3	0.225	Center	0	0	0
C4	0.4	Center	0	0	0

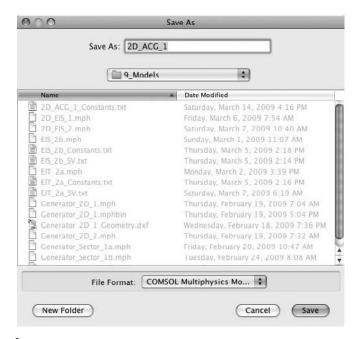


FIGURE 7.55 2D\_ACG\_1 model Save As edit window

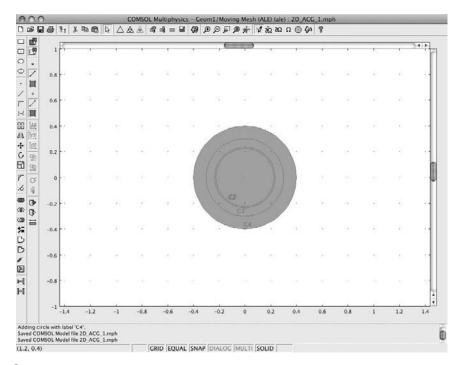


FIGURE 7.56 2D\_ACG\_1 model created circles

Name	Width	Height	Base	X	Y	<b>Rotation Angle</b>
R1	0.1	1.0	Center	0	0	0
R2	0.1	1.0	Center	0	0	45
R3	0.1	1.0	Center	0	0	90
R4	0.1	1.0	Center	0	0	135

Table 7.11	Geometry	Rectangles	Creation
------------	----------	------------	----------

Using the menu bar, select Draw > Specify Objects > Rectangle and create the rectangles indicated in Table 7.11.

Select File > Save. See Figure 7.57.

Select Draw > Create Composite Object. Uncheck the Keep interior boundaries check box. Enter C2+C1\*(R1+R2+R3+R4) in the Set formula edit window. See Figure 7.58.

Note The commands +, \*, and - equal union, intersection, and difference, respectively. Enter the formulas *exactly* as indicated, or the resulting geometry will be incorrect.

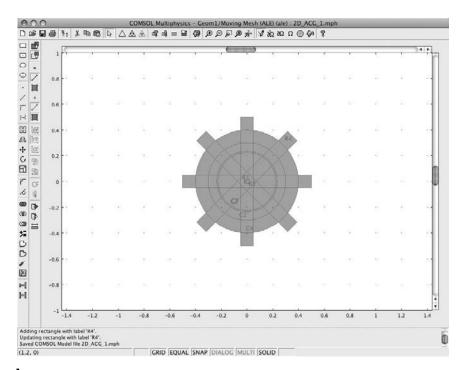


FIGURE 7.57 2D\_ACG\_1 model created circles and rectangles

000	Create Composite Object
Object type Solids	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
C1	C2+C1*(R1+R2+R3+R4)
C2 C3 C4 R1 R2 R3	Keep interior boundaries     Repair     Repair tolerance: 1.0E-4

FIGURE 7.58 2D\_ACG\_1 model Create Composite Object edit window

Click OK. See Figure 7.59.

Select Draw > Create Composite Object. Check the Keep interior boundaries check box. Enter C4+CO1–C3 in the Set formula edit window. See Figure 7.60.

Click OK. See Figure 7.61.

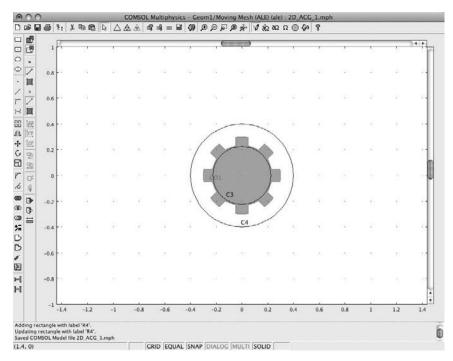


FIGURE 7.59 2D\_ACG\_1 model CO1

O O Cr	eate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: C3 C4 C01	Set formula: C4+CO1-C3 Keep interior boundaries Repair Repair tolerance: 1.0E-4

FIGURE 7.60 2D\_ACG\_1 model Create Composite Object edit window

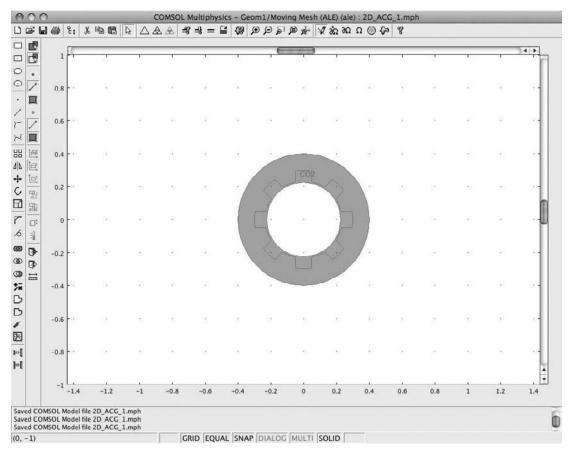


FIGURE 7.61 2D\_ACG\_1 model CO2

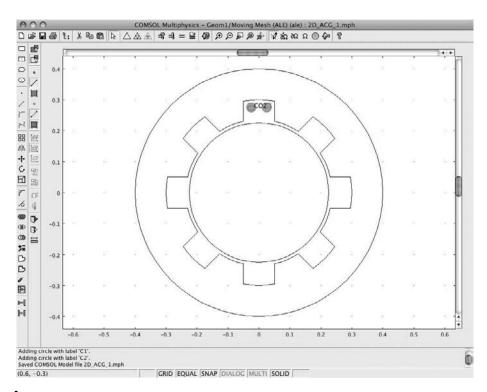


FIGURE 7.62 2D\_ACG\_1 model two new created circles

Click the Zoom Extents button. Using the menu bar, select Draw > Specify Objects > Circle and create the two new circles indicated in Table 7.12.

Click the Save button. See Figure 7.62.

Select C1 and C2. Using the menu bar, select Edit > Copy. Using the menu bar, select Edit > Paste. Verify that the displacements are X:0 and Y:0. See Figure 7.63. Click OK.

Using the menu bar, select Draw > Modify > Rotate. Enter 45 in the Rotation angle edit window. See Figure 7.64. Click OK.

Select C1 and C2. Using the menu bar, select Edit > Copy. Using the menu bar, select Edit > Paste. Verify that the displacements are X:0 and Y:0. Click OK.

Name	Radius	Base	X	Y	<b>Rotation Angle</b>
C1	0.015	Center	0.025	0.275	0
C2	0.015	Center	-0.025	0.275	0

Table 7.12	Geometry (	<b>Circles Creation</b>
------------	------------	-------------------------

0	0.0	Paste
Dis	placements	
<b>X</b> :	0	
Y:	0	Cancel

FIGURE 7.63 2D\_ACG\_1 model Paste C1 and C2

000	Rotate	
Rotation angle α: 45	(degrees)	ОК
Center point		Cancel
<b>X</b> : 0		Help
Y: 0		

FIGURE 7.64 2D\_ACG\_1 model rotated Paste C3 and C4

Using the menu bar, select Draw > Modify > Rotate, for the indicated angles for each circle pair shown in Table 7.13. See Figure 7.65.

Select Edit > Select All. Select Draw > Create Composite Object. Click the Union button and then click OK. See Figure 7.66.

NOTE The stationary portion (stator) of the generator has now been created. The rotating portion (rotor) will be created next.

Using the menu bar, select Draw > Specify Objects > Circle and create the circles indicated in Table 7.14.

Click the Save button. See Figure 7.67.

Table 7.13	Geometry	Circles: Cop	y, Rotate,	and Paste
------------	----------	--------------	------------	-----------

Name	<b>Rotation Angle</b>
C5, C6	90
C7, C8	135
C9, 10	180
C11, C12	-45
C9, 10	-90
C11, C12	-135

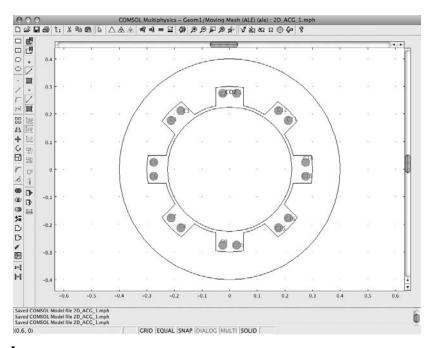


FIGURE 7.65 2D\_ACG\_1 model rotated, pasted circles

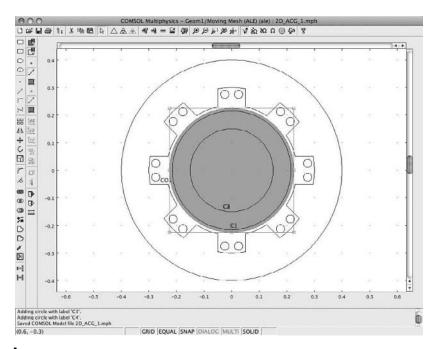


FIGURE 7.66 2D\_ACG\_1 model union of all objects, CO1

Name	Radius	Base	X	Y	<b>Rotation Angle</b>
C1	0.215	Center	0	0	0
C2	0.15	Center	0	0	22.5
C3	0.15	Center	0	0	22.5
C4	0.225	Center	0	0	0

 Table 7.14
 Rotor Geometry Circles Creation

Using the menu bar, select Draw > Specify Objects > Rectangle and create the rectangles indicated in Table 7.15.

Click the Save button. See Figure 7.68.

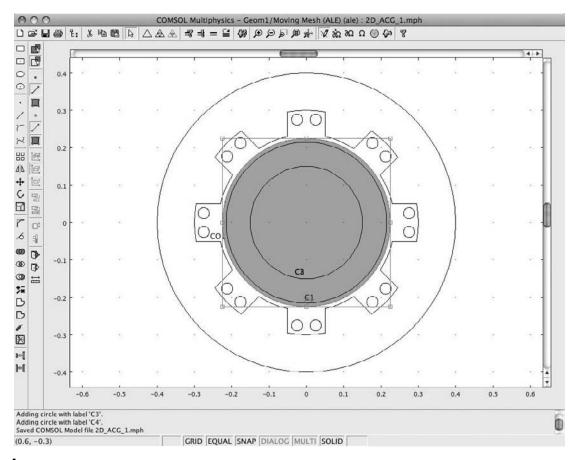


FIGURE 7.67 2D\_ACG\_1 model rotor-created circles

Name	Width	Height	Base	X	Y	Rotation Angle
R1	0.1	1.0	Center	0	0	22.5
R2	0.1	1.0	Center	0	0	-22.5
R3	0.1	1.0	Center	0	0	67.5
R4	0.1	1.0	Center	0	0	-67.5

Table 7.15	Rotor (	Geometry	Rectangles	Creation
------------	---------	----------	------------	----------

Select Draw > Create Composite Object. Uncheck the Keep interior boundaries check box. Enter C2+C1\*(R1+R2+R3+R4) in the Set formula edit window. See Figure 7.69. Click OK.

NOTE The commands +, \*, and - equal union, intersection, and difference, respectively. Enter the formulas *exactly* as indicated, or the resulting geometry will be incorrect.

000	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula: C2+C1*(R1+R2+R3+R4)
C2 C3 C4 R1 R2 R3	Keep interior boundaries

FIGURE 7.69 2D\_ACG\_1 model Create Composite Object edit window

Select Draw > Create Composite Object. Check the Keep interior boundaries check box. Enter CO2+C3+C4 in the Set formula edit window. See Figure 7.70. Click OK. See Figure 7.71.

Note The names Stator and Rotor can be assigned to the appropriate created composite objects by clicking on the composite object (selecting) and selecting Draw > Object Properties. You can then enter the chosen name (Stator, Rotor) in the Name edit window.

Select File > Export > Geometry Objects to File. Enter 2D\_ACG\_1\_Geometry in the Save As edit window. Select "COMSOL Multiphysics binary file (\*.mphbin)" from the File Format pull-down list. Click the Save button. See Figure 7.72.

OOO Cr	eate Composite Object
Object type     Solids     Curves     Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
C01	C02+C3+C4
C3 C4 CO2	Keep interior boundaries Repair Repair tolerance: 1.0E-4

FIGURE 7.70 2D\_ACG\_1 model Create Composite Object edit window

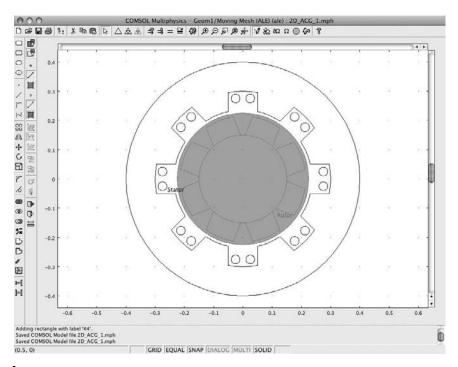


FIGURE 7.71 2D\_ACG\_1 model composite objects



FIGURE 7.72 2D\_ACG\_1 model stator and rotor composite objects



bject selection:	_ Сок
lotor tator	Cancel
	Apply
	Help
air type: Identity pair 🛊	
Create imprints	
Repair	-
керап	

FIGURE 7.73 2D\_ACG\_1 model stator and rotor Create Pairs edit window

## Assemble the Generator Geometry (Stator and Rotor)

Using the menu bar, select Draw > Create Pairs. Select both objects (Rotor and Stator). Select "Identity pair" from the pair type pull-down list. See Figure 7.73.

Click OK. See Figure 7.74.

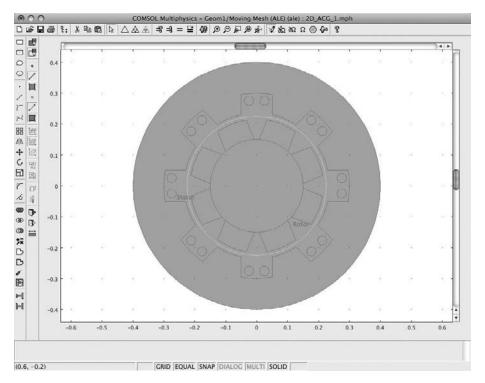


FIGURE 7.74 2D\_ACG\_1 model paired stator and rotor

Subdomains	Name	Expression	Integration Order	Global Destination
3, 4, 9–12, 17, 18	Vi	L*Ez_emqa/A	4	Yes
5–8, 13–16	Vi	-L*Ez_emqa/A	4	Yes

Note The pairing of the rotor and stator couples the boundaries of the two separately created geometric elements. This pairing facilitates the use of the sliding mesh at the boundary between the rotor and the stator, which would otherwise be discontinuous.

## **Physics Settings: Subdomain Integration Variables**

Using the menu bar, select Options > Integration Coupling Variables > Subdomain Variables. In the Subdomain Integration Variables edit window, enter the information shown in Table 7.16; also see Figure 7.75. Click OK.

**NOTE** The integration variable expressions for Vi are summed to yield the voltage induced into the windings of the generator.

Select Options > Materials/Coefficients Library. Verify that the Electric (AC/DC) Materials Properties Library is loaded. If not, you will need to load that library to complete this model. See Figure 7.76. Click OK.

# Physics Subdomain Settings: Perpendicular Induction Currents, Vector Potential (emqa)

Having established the geometry for the 2D\_ACG\_1 model, the next step is to define the fundamental Physics conditions. Using the menu bar, select Multiphysics >

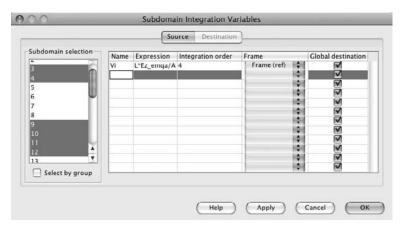


FIGURE 7.75 2D\_ACG\_1 model Subdomain Integration Variables edit window

Materials	-Material properties	
<ul> <li>Model (0)</li> <li>Basic Material Properties (28)</li> <li>Liquids and Gases (18)</li> <li>MEMS Material Properties (33)</li> <li>Heat Transfer Coefficients (8)</li> <li>Electric (AC/DC) Material Prop Copper Soft Iron (without losses)</li> <li>Samarium Cobalt (Radial, c Samarium Cobalt (Radial, c Samarium Cobalt (Radial, i Quartz Graphite Graphite Graphite felt Silicon Carbide</li> <li>Piezoelectric Material Properties User Defined Materials (1)</li> <li>FeNi Material Properties (5)</li> <li>New</li> </ul>		
Copy Paste Add Library	Hide undefined properties	Functions Plot

**FIGURE 7.76** Materials/Coefficients Library window

Perpendicular Induction Currents, Vector Potential (emqa). Using the menu bar, select Physics > Properties. In the Application Mode Properties dialog box, choose "On" from the Weak constraints pull-down list. See Figure 7.77. Click OK.

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window. Click the Load button.

laterials	Material properties	5	
Model (0) Soft Iron (without losses) (	Name: Samariun	n Cobalt (Radial, outward)	
Samarium Cobalt (Radial, ii Samarium Cobalt (Radial, c	Physics	Elastic Electric	Fluid Piezoelectric 🕨
<ul> <li>Basic Material Properties (28)</li> </ul>	Quantity	Value/Expression	Description
Liquids and Gases (18)	Br	0.84[T]*x/sqrt(x^2+y^2	
MEMS Material Properties (33)	epsilonr fH	1	Relative permittivity Nonlinear magneti
Heat Transfer Coefficients (8)	murtensor2D		Relative permeabil
Electric (AC/DC) Material Prop	normfH		Nonlinear magneti
Copper	sigma	0	Electric conductivity
Soft Iron (without losses)			
Samarium Cobalt (Radial, c			
Samarium Cobalt (Radial, ii			
Quartz			
Graphite			
Graphite felt			
Silicon Carbide			
Piezoelectric Material Propertie			
User Defined Materials (1)			
FeNi Material Properties (5)			
Ductile Iron Material Propertie:	Hide undefin	ed properties	Functions
▶ Library 10 (8)			
			Plot
	1		

FIGURE 7.78 2D\_ACG\_1 model Materials/Coefficients Library load window

Select Electric (AC/DC) Materials Properties Library > Soft Iron (without losses). Click the Apply button.

Select Electric (AC/DC) Materials Properties Library > Samarium Cobalt (Radial, inward). Click the Apply button.

Select Electric (AC/DC) Materials Properties Library > Samarium Cobalt (Radial, outward). Click the Apply button. See Figure 7.78. Click the Cancel button.

NOTE The last three commands added the three selected materials to the Model (0) Library for use in this model.

In the Subdomain edit windows, enter the information shown in Table 7.17. Click OK. See Figures 7.79–7.82.

#### Physics Subdomain Settings: Moving Mesh (ALE) (ale)

Using the menu bar, select Physics > Multiphysics > Moving Mesh (ALE) (ale). Using the menu bar, select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 7.18. Click OK. See Figure 7.83.

Subdomain	H ⇔ B Setting	Material	Figure Number
Subuomam		Wateria	ligure Number
1, 3–19	$B = \mu_0 \mu_r H$	—	7.79
2, 28	$H = f( B )e_B$	Soft iron (without losses)	7.80
20, 23, 24, 27	$B = \mu_0 \mu_r H + B_r$	Samarium cobalt (radial, inward)	7.81
21, 22, 25, 26	$B = \mu_0 \mu_r H + B_r$	Samarium cobalt (radial, outward)	7.82

## Table 7.17 Subdomain Edit Window

# Table 7.18 Subdomain Settings – Moving Mesh Edit Window

Subdomain	Group	Figure Number
19–28	rotate_CCW	7.83

## **Physics Boundary Settings**

Leave the Boundary Settings at the default conditions. The identity pair couples the stator and the rotor.

Subdomains Groups	Physics Infin	ite Elements Forces	Init Eler	ment Color
ubdomain selection	Material properties and sou	urces		
0	Library material:		\$	Load
11	Quantity Value/Expres	ssion	Unit De	scription
12	v 0	0	m/s Vel	ocity
.4	Δν 0	11.	V Bot	ential difference
5	1.5			ential difference
.6	L 1		m Len	gth
8	J <sup>e</sup> z 0		A/m <sup>2</sup> Ext	ernal current density
9 🖌	σ		S/m Elec	tric conductivity
1				
roup:	$\mathbf{H} \leftrightarrow \mathbf{B}  \left[ \mathbf{B} = \boldsymbol{\mu}_{0} \boldsymbol{\mu}_{r} \mathbf{H} \right]$	\$	Cor	nstitutive relation
Select by group	μ <sub>r</sub> 1		1 Rela	ative permeability
Active in this domain				

**FIGURE 7.79** 2D\_ACG\_1 model Subdomain Settings (1, 3–19) edit window

Material properties and sources	es)
Library material: Soft Iron (without losse	
	es) 🗘 (Load)
Quantity Value/Expression	Unit Description
A1/	v
0	v Potential difference
L 1	m Length
I <sup>e</sup>	A/m <sup>2</sup> External current density
1886 (	in the second
σ 0[S/m]	S/m Electric conductivity
$H \leftrightarrow B$ $H = f( B )e_{B}$	Constitutive relation
HI HB(normB_emga[1/T])[A/m]	A/m Magnetic field, norm
mb(normb_emqa[1/1]/[A/m]	magnetic neta, norm
	$L = \begin{bmatrix} 0 \\ 1 \\ J_{z}^{e} \end{bmatrix} = \begin{bmatrix} 0 \\ \sigma \end{bmatrix}$ $H \leftrightarrow B = f( B )e_{B} $

**FIGURE 7.80** 2D\_ACG\_1 model Subdomain Settings (2, 28) edit window

Subdomains Groups	Phys	sics Infinite Elements Forces	Init	Element Color		
ubdomain selection	Material properties and sources					
8						
9	Library materi	al: Samarium Cobalt (Radial, inw	ard)	Load		
0	Quantity V	alue/Expression	Unit	Description		
1	ΔV	0	v	Potential difference		
3	L		m			
4	-	1	1	Length		
6	J <sup>e</sup> z	0	A/m <sup>2</sup>	External current density		
7	σ	0	S/m	Electric conductivity		
8 🔻	H↔B					
roup:	п⇔в	$\mathbf{B} = \boldsymbol{\mu}_{0}\boldsymbol{\mu}_{r}\mathbf{H} + \mathbf{B}_{r}  \clubsuit$		Constitutive relation		
Colort hu annua	μ <sub>r</sub>	1	1	Relative permeability		
Select by group	Br		т			

FIGURE 7.81 2D\_ACG\_1 model Subdomain Settings (20, 23, 24, 27) edit window

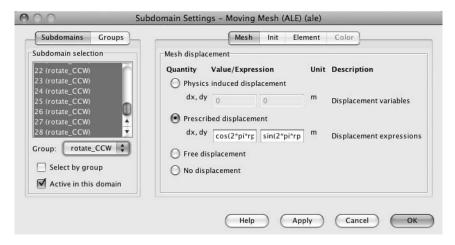
Subdomains Groups	Ph	ysics Infinite Elements Forces	Init	Element Color			
ubdomain selection	Material prop	Material properties and sources					
8	Library mate	Library material: Samarium Cobalt (Radial, outward)					
9							
1	Quantity ∆V	Value/Expression 0	Unit V	Description Potential difference			
3	L	1	m	Length			
5	J <sup>e</sup> z	0	A/m <sup>2</sup>	External current density			
7	σ	0	S/m	Electric conductivity			
oup:	H ↔ B	$  B = \mu_0 \mu_r H + B_r $		Constitutive relation			
Select by group	μ	1	1	Relative permeability			
Active in this domain	B,	0.84[T]*x/sqrt(x/ 0.84[T]*y/sqrt(x/	т	Remanent flux density			

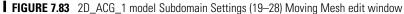
FIGURE 7.82 2D\_ACG\_1 model Subdomain Settings (21, 22, 25, 26) edit window

## **Mesh Generation**

From the toolbar, select Mesh > Free Mesh Parameters > Global. Select Predefined mesh sizes > Finer (from the pull-down list). Select "Custom mesh size." Enter 2 in the Resolution of narrow regions edit window. See Figure 7.84.

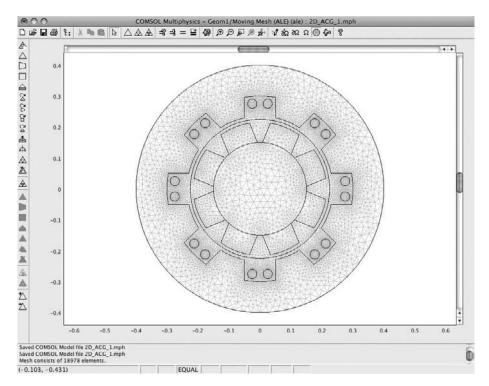
Click the Remesh button, and then click OK. See Figure 7.85.





	Global	Subdomain	Boundary	Point	Advanced	J	
O Predet	ined mesh s	izes:	Finer		۵.		Can
🕑 Custo	m mesh siz	e					Ap
Maximum	element size	:					
Maximum	element size	scaling factor	0.55				He
Element gr	owth rate:		1.25				
Mesh curv	ature factor		0.25				
Mesh curv	ature cutoff		0.0005				
Resolution	of narrow r	egions:	2				
🗹 Optimia	ze quality						
efinement	mathad:	Regular 🛟					
ennement		Regular					

**FIGURE 7.84** 2D\_ACG\_1 model Free Mesh Parameters edit window



**FIGURE 7.85** 2D\_ACG\_1 model mesh

000	Solver Pa	ameters		_
Analysis:	Genera	I Stationary Adapti	ve Advanced	
Static  Auto select solver Solver: Stationary Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated	Linear system solver Linear system solver: Preconditioner:	Direct (UMFPACK)	: Settin	gs)
Adaptive mesh refinement	Matrix symmetry:	Automatic		

**FIGURE 7.86** 2D\_ACG\_1 model Solver Parameters edit window

#### Solving the Static 2D\_ACG\_1 Model

Using the menu bar, select Solve > Solver Parameters.

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Select "Static" from the Analysis pull-down list. See Figure 7.86. Click OK. Using the menu bar, select Solve > Solve Problem. See Figure 7.87.

Click the Save button. Select File > Save As. Enter 2D\_ACG\_2.mph in the Save As edit window. See Figure 7.88. Click OK.

NOTE The 2D\_ACG\_1 model (static) solution was built to gain experience in the creation of a complex geometrical model. It was saved as 2D\_ACG\_2.mph and will act as the initial estimate for the 2D\_ACG\_2 model (transient) solution.

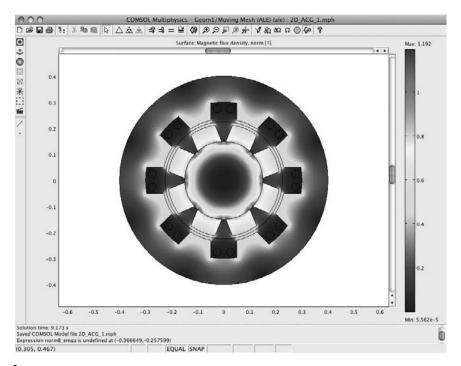


FIGURE 7.87 2D\_ACG\_1 model solution

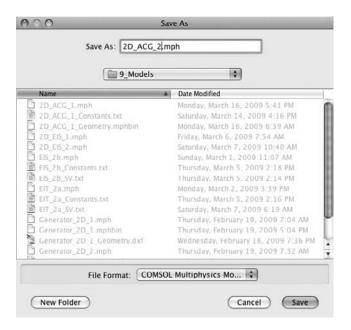


FIGURE 7.88 2D\_ACG\_2 model initial solution

		Model Navigator	
	New Model Libr	ary User Models Open	Settings
	9_M	odels 🗘	)
		Name 🔺	Date Modified
		2D_ACG_1.mph	Monday, March 16, 2
10		D 2D_ACG_1_Const	Saturday, March 14,
	RE	2D_ACG_1_Geo	Monday, March 16, 2
and the second second	ETS.	2D_ACG_2.mph	Tuesday, March 17,
E.L.		2D_EIS_1.mph	Friday, March 6, 200
Ei	14	2D_EIS_2.mph	Saturday, March 7, 2
		EIS_2b.mph	Sunday, March 1, 20
		EIS_2b_Constants	Thursday, March 5, 2
_		置 EIS_2b_SV.txt	Thursday, March 5, 2
Description:		EIT_2a.mph	Monday, March 2, 20
		置 EIT_2a_Constants	
		圖 EIT_2a_SV.txt	Saturday, March 7, 2
		Generator_2D_1	Thursday, February 1
L		Generator_2D_1	Thursday, February 1
Preview		Generator_2D_1	Wednesday, February
		COMSOL Multiphysics Mo	

FIGURE 7.89 2D\_ACG\_2 Model Navigator initial solution selection

## 2D AC Generator Model (2D\_ACG\_2): Transient

The following numerical solution model (2D\_ACG\_2) follows directly from the earlier model 2D\_ACG\_1 model, which was built in the preceding subsection. In this next subsection, the transient 2D generator model (2D\_ACG\_2) uses the static 2D\_ACG\_1 model as the initial solution to the transient problem. The transient version avoids all the complex geometrical building by starting with the earlier saved solution.

To start building the 2D\_ACG\_2 model (transient) solution, activate the COMSOL Multiphysics software. In the Model Navigator, select "Open." Select "2D\_ACG\_2.mph." See Figure 7.89.

Click OK. See Figure 7.90.

NOTE Because the initial solution to the transient 2D\_ACG\_2 model has already been built and verified, the modeler can proceed directly to implementing the necessary transient solver setup parameters.

#### Solving the Transient 2D\_ACG\_2 Model

Using the menu bar, select Solve > Solver Parameters. Select "Transient" from the Analysis pull-down list.

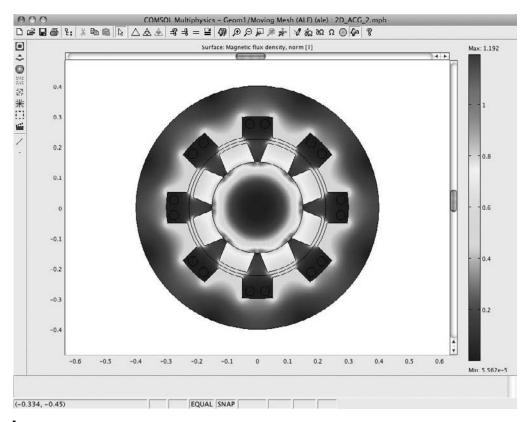


FIGURE 7.90 2D\_ACG\_2 Model Navigator initial solution

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

In the Times edit window, enter linspace(0,0.25,26). For later versions of COMSOL Multiphysics software enter range(0,0.25/25,0.25). In the Absolute tolerance edit window, enter Az 1e-3 lm1 5e3. Click the Apply button. See Figure 7.91. Click OK.

NOTE It is important to enter the Solver Parameters *exactly* as specified; otherwise, the modeler will see error messages.

Using the menu bar, select Solve > Restart. See Figure 7.92.

000	Solver Paramete	rs		
Analysis:	General	Time Stepping Advanced		
Transient  Auto select solver Solver: Stationary Lime dependent Eigenvalue	Time stepping Times: Relative tolerance: Absolute tolerance:	linspace(0,0.25,26) 0.01 Az 1e-3 lm1 5e3		
Parametric	Allow complex numbers			
Stationary segregated Parametric segregated	Preconditioner:	t (UMFPACK)		
	Matrix symmetry: Auto	matic 🗘		
	(He	lp Apply Cancel OK		

FIGURE 7.91 2D\_ACG\_2 model Solver Parameters edit window

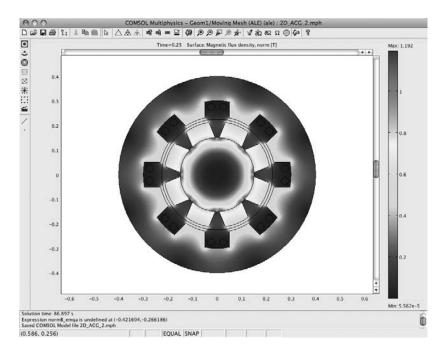


FIGURE 7.92 2D\_ACG\_2 model solution (transient)

General Surface	Contour Boundary Arrow Principal
International Surface  Surface  Surface  Contour  Boundary  Arrow  Principal  Streamline  Particle tracing  Max/min marker  Deformed shape  Geometry edges	Contour       Boundary       Arrow       Principal         Solution to use       Solution at time:       0.2       Image: Contour at angle (phase):       Image: Contour at angle (phase):

**FIGURE 7.93** 2D\_ACG\_2 model Plot Parameters, General tab

## Postprocessing and Visualization

The default plot shows a surface plot of the normal magnetic flux density in teslas for the solution at 0.25 second. To visualize the field distribution, the plot parameters will need to be modified.

Select Postprocessing > Plot Parameters > General. Check the Surface and Contour check boxes under Plot type. Uncheck the Geometry edges check box. Select "Solution at time 0.2 seconds" from the Solutions to use pull-down list. See Figure 7.93.

General Surfa ✓ Surface plot	ce Contour			
Surface plot		Boundary	Arrow	Principal
- Surface plot				
	Surface Data	Height Data		
5 1 C 1	Magnatic flux	density, norm 🛟		
Predefined quantiti		density, norm		
Expression:	normB_emqa	1000	Smoot	h
Unit:		\$	J	
Coloring and fill				
Coloring: Inte	rpolated 🗘	Fill style:	Filled	\$
Colormap:	jet 🗘	Colors: 1024	Color s	cale

**FIGURE 7.94** 2D\_ACG\_2 model Plot Parameters, Surface tab

Click the Surface tab. Select "Magnetic flux density, norm." See Figure 7.94. Click the Contour tab. Enter Az in the Expression edit window. Enter 15 in the Levels edit window. Click the Uniform color radio button. Click the Color button, and select black. See Figure 7.95.

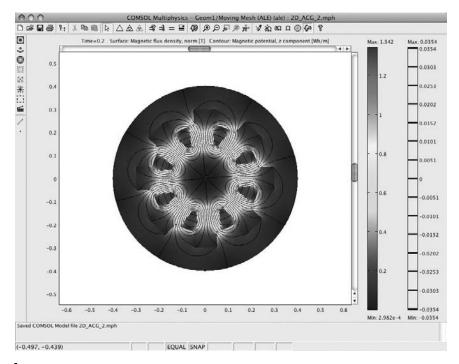
Predefined quantitie Expression: Unit: Contour levels		al, z com 🛟	ooth
Predefined quantitie Expression: Unit: Contour levels	S: Magnetic potentia	al, z com 🗘	ooth
Expression: Unit: Contour levels	Az	∑ Sm	ooth
Unit: Contour levels			ooth
Contour levels	Wb/m	\$	
Levels: 💽 🗍	mber of levels	Vector with isolevels	
Contour color			
O Colormap:	cool 🛟 (	Colors: 1024	Color scale
Uniform color:	Color		
Filled			

**FIGURE 7.95** 2D\_ACG\_2 model Plot Parameters, Contour tab

Click OK. See Figure 7.96.

To view the induced voltage as a function of time, select Postprocessing > Domain Plot Parameters > Point. Select point 10. Enter Vi\*NN in the expression edit window. See Figure 7.97.

Click OK. See Figure 7.98.



**FIGURE 7.96** 2D\_ACG\_2 model magnetic flux density and magnetic potential

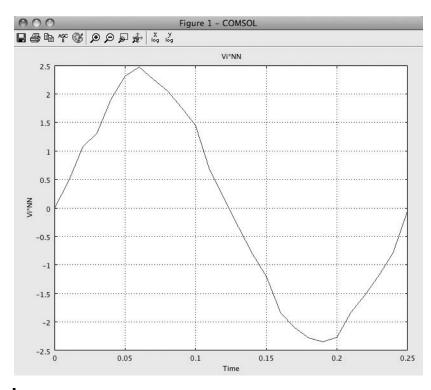


FIGURE 7.98 2D\_ACG\_2 model induced voltage vs. time

#### **Postprocessing Animation**

Select Postprocessing > Plot Parameters > Animate. Select or verify that all of the solutions in the Solutions to use window are selected. See Figure 7.99.

Click the Start Animation button. See Figure 7.100.

Alternatively, you can play the file Movie7\_ACG\_2.avi that was supplied with this book.

## 2D AC Generators, Static and Transient Models: Summary and Conclusions

In this section, we built two 2D AC generator models: static and transient. These models generate low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. They demonstrate the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of a rotating machine for the conversion of mechanical energy to electrical energy.

The modeling and physics concepts employed in this section of Chapter 7 are mechanical to electrical energy conversion, hard and soft nonlinear magnetic materials, moving mesh (ALE), and geometric assembly (pair creation across a boundary).

### 530 CHAPTER 7 2D COMPLEX MIXED-MODE MODELING

Principal	Streamline Particle T	racing Max/Min Deform Anima
Movie settings		Solutions to use
File type:	AVI	Select via: Stored output times
Width (in pixels):	640	0
Height (in pixels)	480	0.01
		0.02
Frames per secor	nd: 10	0.04
	Advanced	0.05 0.06
Static / Eigenfund	tion animation	0.07 🛁
		0.08
Cycle type:	Full harmonic	10.09
Cycle type:	Full harmonic	Times:
Number of frame	s: 11	10.09
Number of frame	s: 11	10.09

FIGURE 7.99 2D\_ACG\_2 model Plot Parameters, Animate tab

## **2D AC Generator: Sector—Static and Transient**

In this section, static and transient sector-based models are developed that are only one-eighth the size of the full-size model. The sector models utilize the inherent basic symmetry employed in AC generators. Additionally, an ordinary differential equation<sup>28</sup> is incorporated into the model to allow the exploration of the torques resulting from the magnet forces inherent in this AC generator design. In these machines, the generation of AC power is accomplished through the application of Faraday's law of induction. In the following models, a magnetic vector potential (**A**) is employed that has only the *z* component.

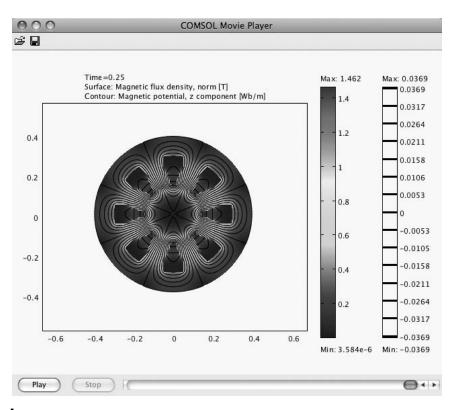


FIGURE 7.100 2D\_ACG\_2 model animation, final frame

Rotation is modeled using deformed Mesh Application Mode (ALE). The rotor and the stator are drawn separately and then combined as an assembly.

The materials employed in this model are high-energy samarium-cobalt magnets with nonlinear soft iron pole pieces.

## 2D AC Generator Sector Model (2D\_ACGS\_1): Static

The following numerical solution model (2D\_ACGS\_1) is derived from a model that was originally developed by COMSOL as an AC/DC Module Motors and Drives Library Model. First, this 2D generator model will be built as a static (stationary) model. In the next subsection, the static model will be used as the starting point for the transient (rotating) model (2D\_ACGS\_2).

To start building the 2D\_ACGS\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" (the default setting) from the Space dimension pull-down list. Select AC/DC Module > Rotating Machinery > Rotating Perpendicular Currents. Enter Az X Y Z in the Dependent variables edit window. See Figure 7.101. Click OK.

Ĩ

0	Model Navigat	tor
New	Model Library User Model	els Open Settings
Space dimension:	2D 🗘	
Space dimension: 20 Application Modes COMSOL Multiphysics AC/DC Module Statics Quasi-Statics, Electric Quasi-Statics, Electromagnetic Rotating Perpendicular Currents Electro-Thermal Interaction Heat Transfer Module RF Module RF Module		AC/DC Module Description: A predefined combination of application modes for rotating electro-mechanical devices. Combin Perpendicular Induction Currents with Moving Mesh for modeling of rotatin electro-mechanical applications such as electric motors.
Dependent variables:	AzXYZ	
Application mode name: Element:	emqa ale	Multiphysics

FIGURE 7.101 2D\_ACGS\_1 Model Navigator setup

Be sure to change the dependent variables from lowercase x, y, z to uppercase X, Y, Z. This demonstrates that the modeler can alter the Reference Frame in COMSOL Multiphysics Software, if needed. Also note that the Model Navigator shows two names in the Application mode name edit window: emqa and ale. Those names—emqa (Rotating Perpendicular Currents) and ale (Moving Mesh)—indicate the Application Modes employed in this modeling analysis.

Select File > Save As. Enter 2D\_ACGS\_1.mph in the Save As edit window. Click the Save button.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 7.19; also see Figure 7.102. Click OK.

#### **Generator Sector Geometry**

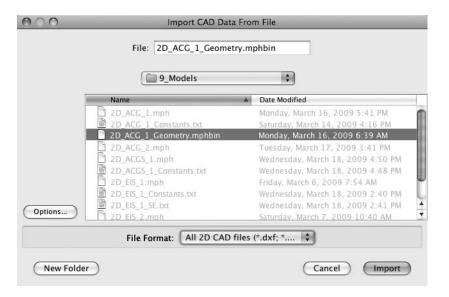
Select File > Import > CAD Data From File > 2D\_ACG\_1\_Geometry.mphbin. See Figure 7.103.

Name	Expression	Description
A	pi*(0.02[m])^2	Area of wires in stator
L	0.4[m]	Length of generator
NN	1	Number of turns in stator winding
Μ	1400[N*m]	Externally applied torque
Rc	1e-4[ohm]	Resistance of winding
10	100[kg*m^2]	External moment of inertia

#### Table 7.19 Constants Edit Window

Name	Expression	Value	Description	
A	pi*(0.02[m])^2	0.001257[m <sup>2</sup> ]	Area of wires in stator	ñ
L	0.4[m]	0.4[m]	Length of generator	1
NN	1	1	Number of turns in stator winding	1
м	1400[N*m]	1400[J]	Externally applied torque	1
Rc	1e-4[ohm]	(le-4)[Ω]	Resistance of winding	1
10	100[kg*m^2]	100[m <sup>2</sup> ·kg]	External moment of inertia	۲
				2

FIGURE 7.102 2D\_ACGS\_1 model Constants edit window



**FIGURE 7.103** Import CAD Data From File select window

**FIGURE 7.104** Imported CAD file

Click the Import button. See Figure 7.104.

Note Now that the geometry for the entire 2D generator has been imported, the modeler needs to create the 2D generator sector. Carefully enter *exactly* the following construction, to create the 2D generator sector.

Using the menu bar, select Draw > Specify Objects > Line. Select "Closed polyline (solid)" from the Style pull-down list. Enter the following formula in the x edit window:

0 0.4\*cos(67.5\*pi/180) 0.4 0.4\*cos(22.5\*pi/180) 0

Enter the following formula in the y edit window:

0 0.4\*sin(67.5\*pi/180) 0.4 0.4\*sin(22.5\*pi/180) 0

See Figure 7.105.

Coor	dinates	
<b>x</b> :	0 0.4*cos(67.5*pi/180) 0.4 0.4	ОК
y:	0 0.4*sin(67.5*pi/180) 0.4 0.4	Cancel
Style:	Closed polyline (solid)	Apply
Name:	C03	Help

**FIGURE 7.105** Line, Closed polyline (solid) edit window

Click OK. See Figure 7.106.

Select the newly created solid (CO3) only. Using the menu bar, select Edit > Copy. Using the menu bar, select Draw > Create Composite Object. Enter CO1\*CO3 in the Set formula window. See Figure 7.107. Click OK.

Using the menu bar, select Edit > Paste. Click OK on the x = 0, y = 0 Displacements dialog box.

000	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
CO2 CO3	Repair 1.0E-4

**FIGURE 7.107** Create Composite Object, intersection (CO1, CO3)

Using the menu bar, select Draw > Create Composite Object. Enter CO2\*CO1 in the Set formula window. See Figure 7.108.

Click OK, and then click the Zoom Extents button. See Figure 7.109.

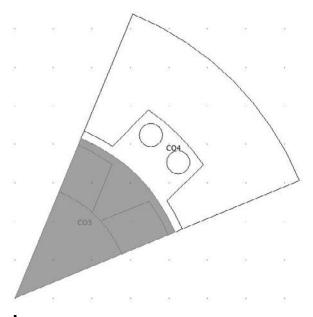
#### Assemble the Geometry (Stator and Rotor)

Select both objects (CO3, CO4). Locate the Create Pairs button on the Draw toolbar. See Figure 7.110.

Click the Create Pairs button on the Draw toolbar. See Figure 7.111.

OOO Crea	te Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: CO2 CO4 CO1	Set formula: CO2*CO1 Keep interior boundaries Repair Repair tolerance: 1.0E-4

FIGURE 7.108 Create Composite Object, intersection (CO2, CO1)



**FIGURE 7.109** Created one-eighth generator sector (CO3, CO4)



**FIGURE 7.110** Create Pairs button on Draw toolbar

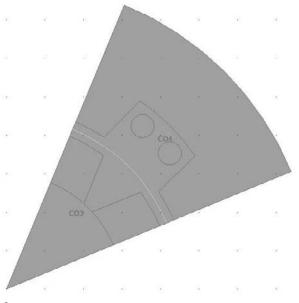


FIGURE 7.111 2D\_ACGS\_1 model paired stator (CO3) and rotor (CO4)

lentity pairs	Boundarie	es Advanced
Name: Pair 1	Boundaries Source boundaries 6 7 8 9 9 10 11 Check Selected Clear Selected Select Source	Destination boundaries

**FIGURE 7.112** Identity Boundary Pairs window

NOTE The pairing of the rotor and the stator couples the boundaries of the two separately created geometric elements. This pairing facilitates the use of the sliding mesh at the boundary between the rotor and the stator, which would otherwise be discontinuous.

Using the menu bar, select Physics > Identity Pairs > Identity Boundary Pairs. Select Pair 1. See Figure 7.112.

Click the Interchange source and destination button, which is located between the Source boundaries and Destination boundaries fields. See Figure 7.113. Click OK.

#### **Options: Global Expressions**

Using the menu bar, select Options > Expressions > Global Expressions. In the Global Expressions edit window, enter the information shown in Table 7.20; also see Figure 7.114. Click OK.



**FIGURE 7.113** Interchange source and destination button

#### Table 7.20 Global Expressions Edit Window

Name	Expression	Description
Tz	8*L[1/m]*F_torquez_emqa	Total torque for the entire device

FIGURE 7.114 2D\_ACGS\_1 model Global Expressions edit window

NOTE The insertion of [1/m] in the expression for Tz converts L into a unitless number. Because of the use of coupling variables, COMSOL Multiphysics software cannot determine the units of other variables that depend either directly or indirectly on the coupling variables. This may cause inconsistent unit warnings to appear. Such warnings can be ignored.

#### **Options: Subdomain Expressions**

Using the menu bar, select Options > Expressions > Subdomain Expressions. In the Subdomain Expressions edit window, enter the information shown in Table 7.21. Click OK. See Figures 7.115 and 7.116.

Using the menu bar, select Options > Expressions > Subdomain Expressions. In the Subdomain Expressions edit window, enter the information shown in Table 7.22. Click OK. See Figures 7.117 and 7.118.

#### **Options: Subdomain Integration Variables**

Using the menu bar, select Options > Integration Coupling Variables > Subdomain Variables. In the Subdomain Integration Variables edit window, enter the information shown in Table 7.23. Click OK. See Figures 7.119 and 7.120.

Subdomain	Name	Expression	Figure Number
1–4	u	-y*alphat	7.115
1–4	V	x*alphat	7.115
5–8	u	0	7.116
5–8	V	0	7.116

odomain selection	Name	Expression	Unit
	u	-y*alphat	0
	v	x*alphat	
Select by group			

FIGURE 7.115 2D\_ACGS\_1 model Subdomain Expressions (1-4) edit window

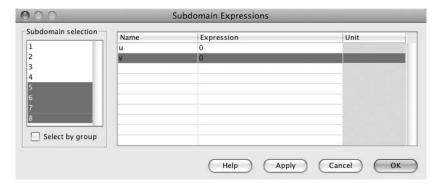


FIGURE 7.116 2D\_ACGS\_1 model Subdomain Expressions (5–8) edit window

## Table 7.22 Subdomain Expressions Edit Window

Subdomain	Name	Expression	Figure Number
1	rho	7870[kg/m^3]	7.117
2, 4	rho	8400[kg/m^3]	7.118

## Table 7.23 Subdomain Integration Variables Edit Window

Subdomain	Name	Expression	Integration Order	Global	Figure Number
1, 2, 4	I	8*L*rho*(x^2+y^2)	4	Yes	7.119
7, 8	Vi	8*L*NN*Ez_emqa/A	4	Yes	7.120

bdomain selection	Name	Expression	Unit
	u	-y*alphat	D
	v	x*alphat	D
	rho	7870[kg/m^3]	kg/m <sup>3</sup>
Select by group			

FIGURE 7.117 2D\_ACGS\_1 model Subdomain Expressions (1) edit window

	Name	Expression	Unit
	u	-y*alphat	0
	v	x*alphat	0
	rho	8400[kg/m^3]	kg/m <sup>3</sup>
Select by group			

**FIGURE 7.118** 2D\_ACGS\_1 model Subdomain Expressions (2, 4) edit window

ubdomain selection	Name	Expression	Integration order	Frame		Global destin
	I	8*L*rho*(x^2+y^2)	4	Frame (ref)	-	$\checkmark$
		1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 - 1994 -			-	
					-	
					-	
					4	V
					-	
					-	V
					-	
					4	
					-	222
					4	
					-	
Select by group	6					

FIGURE 7.119 2D\_ACGS\_1 model Subdomain Integration Variables (1, 2, 4) edit window

ubdomain selection	Name	Expression	Integration order	Frame	Global destir
L 2 3 4 5 5 7 3		8°L*NN^Ez_emqa/A	4		
Select by group	_				

FIGURE 7.120 2D\_ACGS\_1 model Subdomain Integration Variables (7, 8) edit window

Note Because of the use of coupling variables, COMSOL Multiphysics software cannot determine the units of other variables that depend either directly or indirectly on the coupling variables. This may cause inconsistent unit warnings to appear. Such warnings can be ignored.

#### Physics Subdomain Settings: Moving Mesh (ALE) (ale)

Using the menu bar, select Multiphysics > Moving Mesh (ALE) (ale). Select Physics > Subdomain Settings. Select subdomains 1–4. Select "angle\_CCW" from the Group pull-down list. See Figure 7.121. Click OK.

NOTE The selection of "angle\_CCW" makes the variable omega available for use in the ODE, once defined later.

## **Application Mode Properties: Perpendicular Induction Currents, Vector Potential (emqa)**

Using the menu bar, select Multiphysics > Perpendicular Induction Currents, Vector Potential (emqa). Select Physics > Properties. Select "Frame (ref)" from the Frame pull-down list. Select "On" from the Weak constraints pull-down list. See Figure 7.122. Click OK.

### Physics Subdomain Settings: Perpendicular Induction Currents, Vector Potential (emqa)

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 in the Subdomain selection window. Click the Load button.

Subdomain selection	⊢ Mesh displacement
1 (angle_CCW) 2 (angle_CCW) 3 (angle_CCW)	Quantity Value/Expression Unit Description O Physics induced displacement
4 (angle_CCW) 5 (default) 6 (default)	dX, dY 0 0 m Displacement variables
7 (default)	dX, dY cos(omega sin(omega) <sup>m</sup> Displacement expression: Free displacement No displacement

FIGURE 7.121 2D\_ACGS\_1 model Subdomain Settings (1–4) Moving Mesh (ALE) edit window

Select Electric (AC/DC) Materials Properties Library > Soft Iron (without losses). Click the Apply button.

Select Electric (AC/DC) Materials Properties Library > Samarium Cobalt (Radial, inward). Click the Apply button.

Select Electric (AC/DC) Materials Properties Library > Samarium Cobalt (Radial, outward). Click the Apply button. See Figure 7.123. Click the Cancel button.

Default element type:	Lagrange – Quadratic	+
Analysis type:	Transient	+
Bias application mode:	None	A V
Frame:	Frame (ref)	\$
Weak constraints:	On	\$
Constraint type:	Ideal	4

FIGURE 7.122 2D\_ACGS\_1 model Application Mode Properties, Perpendicular Induction Currents, Vector Potential (emqa)

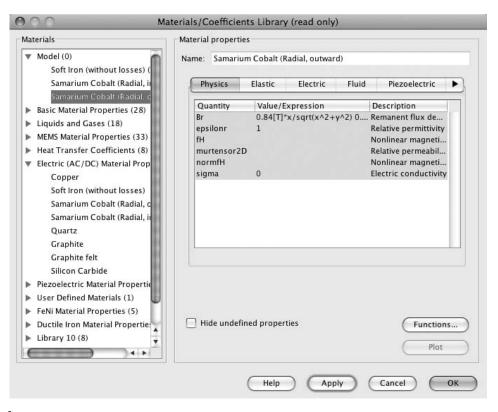


FIGURE 7.123 2D\_ACGS\_1 model Materials/Coefficients Library load window

NOTE The last three commands added the three selected materials to the Model (0) Library for use in this model.

In the Subdomain edit windows, enter the information shown in Table 7.24. Click the Apply button. See Figures 7.124–7.127.

Table 7.24	Subdomain	Edit Windows
------------	-----------	--------------

Subdomain	$H \Leftrightarrow B \text{ Setting}$	Material	Figure Number
1, 6	$H = f( B )e_B$	Soft iron (without losses)	7.124
2	$B = \mu_0 \mu_r H + B_r$	Samarium cobalt (radial, outward)	7.125
4	$B = \mu_0 \mu_r H + B_r$	Samarium cobalt (radial, inward)	7.126
3, 5, 7, 8	$B = \boldsymbol{\mu}_{0}\boldsymbol{\mu}_{r}H$	_	7.127

Subdomains Groups	Ph	ysics Infinite Elements Forces	Init	Element Color
ubdomain selection	Material prop	erties and sources		
	Library mate	rial: Soft Iron (without losses)		Load
	Quantity	Value/Expression	Unit	Description
	ΔV	0	v	Potential difference
	L	1	m	Length
	J <sup>e</sup> z	0	A/m <sup>2</sup>	External current density
	σ	0[S/m]	S/m	Electric conductivity
	H ↔ B	$H = f( B )e_{B}$		Constitutive relation
roup:	(H)	HB(normB_emga[1/T])[A/m]	A/m	Magnetic field, norm
Select by group		HB(normB_emqa[1/1])[A/m]		magnetic neid, norm

FIGURE 1.124 2D\_ACGS\_1 model Subdomain Settings (1, 6) edit window

Subdomains Groups	Physics	Infinite Elements	Forces Init	Element Color
bdomain selection	-Material propertie		rorces inte	Element Color
	and a second			
	Library material:	Samarium Cobalt (	Radial, outward)	Load
	22207/2 22	e/Expression	Unit	Description
	ΔV 0		V	Potential difference
	L 1		m	Length
	J <sup>e</sup> z 0		A/m <sup>2</sup>	External current density
	σο		S/m	Electric conductivity
roup:	H ↔ BB	$= \mu_0 \mu_r H + B_r$		Constitutive relation
Select by group	μ <sub>r</sub> 1		1	Relative permeability
Active in this domain	B <sub>r</sub> 0.8	4[T]*x/sqrt(x/ 0.84[T	Thu (continue)	Remanent flux density

**FIGURE 7.125** 2D\_ACGS\_1 model Subdomain Settings (2) edit window

$A_{z}/\partial t + \nabla \times (\mu_{0}^{-1}\mu_{r}^{-1})\nabla \times A_{z}$	$(z - B_r) = \sigma \Delta V / L + J^c_z$	
Subdomains Groups	Physics Infinite Elements Forces	Init Element Color
ubdomain selection	Material properties and sources	
1	Library material: Samarium Cobalt (Radial, in	ward)
3	Quantity Value/Expression	Unit Description
5	ΔV 0	V Potential difference
G 7	L 1	m Length
5	J <sup>e</sup> _ 0	A/m <sup>2</sup> External current density
	υ 0	S/m Electric conductivity
	$H \leftrightarrow B$ $B = \mu_0 \mu_r H + B_r$	Constitutive relation
iroup:	μ <sub>r</sub> 1	1 Relative permeability
Select by group	B <sub>r</sub> -0.84[T]*x/sqrt() -0.84[T]*y/sqrt	to T Remanent flux density

**FIGURE 7.126** 2D\_ACGS\_1 model Subdomain Settings (4) edit window

Select subdomains 7 and 8. Enter NN\*Vi/(Rc\*A) in the External current density edit window. See Figure 7.128. Click the Apply button.

Click the Forces tab. Select subdomains 1, 2, and 4. Enter F as the name of the variable. Click the Apply button. See Figure 7.129. Click OK.

Subdomains Groups	Phy	ysics Infinite Elements For	ces Init	Element Color
ubdomain selection	Material prop	erties and sources		
	Library mate	rial:		Load
	Quantity	Value/Expression	Unit	Description
	ΔV	0	v	Potential difference
	L	1	m	Length
	۶ <sup>۳</sup>	0	A/m <sup>2</sup>	External current density
	σ	0	S/m	Electric conductivity
roup:	H ⊷ B	$\mathbf{B} = \mu_0 \mu_r \mathbf{H}$		Constitutive relation
	μ	1	1	Relative permeability
Select by group Active in this domain				

**FIGURE 7.127** 2D\_ACGS\_1 model Subdomain Settings (3, 5, 7, 8) edit window

Subdomains Groups	Ph	ysics Infinite Elements For	ces Init Element Color
ubdomain selection	-Material prop	erties and sources	
;	Library mate	rial:	Load
roup:	Quantity $\Delta V$ L $\int_{z}^{e}$ $\sigma$ H $\leftrightarrow$ B	Value/Expression	Unit     Description       V     Potential difference       m     Length       A/m <sup>2</sup> External current density       S/m     Electric conductivity       Constitutive relation
Select by group	μ <sub>r</sub>	1	1 Relative permeability

FIGURE 7.128 2D\_ACGS\_1 model Subdomain Settings (7, 8) edit window

## Physics Boundary Settings: Perpendicular Induction Currents, Vector Potential (emqa)

Using the menu bar, Select Physics > Boundary Settings. Select boundaries 1–4, 7, 8, 15, 16, 19, and 20 in the Boundary selection window.

Subdomains Groups	P	hysics Infinite Elemen	ts Forces	Init Element Color
ubdomain selection	Electromagn	netic force variables		
	Name	Point		
	F	0 0		
roup: 🔹 🗘	surface stre for the tota Examples: name_force	ne above, COMSOL Multip ess-tensor boundary vari electromagnetic force a ex_emqa for the total force rex_emqa for the total to	ables and scal nd torque. e in the x dire	ar variables ction.

FIGURE 7.129 2D\_ACGS\_1 model Subdomain Settings (1, 2, 4) Forces edit window

Boundaries Groups Pairs	Conditions Weak Constr. Color/Style
Boundary selection	Boundary sources and constraints Boundary condition Type of periodicity: Antiperiodicity Periodic pair index: 0 1 Change source and destination order

**FIGURE 7.130** 2D\_ACGS\_1 model Boundary Settings (1–4, 7, 8, 15, 16, 19, 20) Conditions page

On the Conditions page, select "Periodic condition" from the Boundary condition pull-down list. Also select "Antiperiodicity" from the Type of periodicity pull-down list. See Figure 7.130.

Click the Weak Constr. tab. Select boundaries 1–4, 7, 8, 15, 16, 19–21, and 23 in the Boundary selection window. Uncheck the Use weak constraints check box. Click the Apply button. See Figure 7.131.

Boundaries Groups Pairs	h	_0	Conditions Weak Constr. Color	/Style
roup:	Weak constra Use wea Predefined e Coefficient wcshape wcgporder wcinit	k constra lements: Value	Lagrange - Quadratic	Description Shape functions Integration order Initial value

FIGURE 7.131 2D\_ACGS\_1 model Boundary Settings (1–4, 7, 8, 15, 16, 19–21, 23) Weak Constr. page

Boundaries Groups Pairs	Conditions Weak Constr. Color/Style
Pair 1 (identity)	Boundary sources and constraints Boundary condition: Sector antisymmetry  n <sub>sect</sub> Number of sectors

**FIGURE 7.132** 2D\_ACGS\_1 model Boundary Settings Pairs selection (1)

Click the Pairs tab. Select Pair 1. Select "Sector antisymmetry" from the Boundary condition pull-down list. Enter 8 in the Number of sectors edit window. See Figure 7.132. Click OK.

#### **Physics Settings: Periodic Point Conditions**

Using the menu bar, select Physics > Periodic Conditions > Periodic Point Conditions. Select point 4 in the Point selection window. On the Source page, enter lm1 (Lagrange Multiplier 1) in the first row of the Expression edit window. Press Return. See Figure 7.133.

	Source Destination	Source Vertices Destination Vertices
oint selection	Expression	Constraint name
1	Im1	pconstr1
3		
5		
5		
,		
	¥	
io		
Select by group	Vector element co	nstraint

**FIGURE 7.133** 2D\_ACGS\_1 model Periodic Point Conditions (4) Source page

00	Perioc	lic Point Condition	ons
	Source Destination	Source Vertices	Destination Vertices
Constraint name: pcc	onstr1		
Vertex selection:	Source ver	tices:	
1			ientation of the source domains
2			ely the destination domains is
3			nined by matching source and ation vertices in the order they are
4		added	
5			
6			
7			
8			
9			
10	1		

**FIGURE 7.134** 2D\_ACGS\_1 model Periodic Point Conditions, Source Vertices (4) page

NOTE When the modeler presses the Return key, pconstr1 appears in the Constraint name window.

Click the Source Vertices tab. Click the >> button to select point 4 as a source vertex. See Figure 7.134.

Click the Destination tab. Select point 11. Check the Use selected points as destination check box. Enter -lm1. See Figure 7.135.

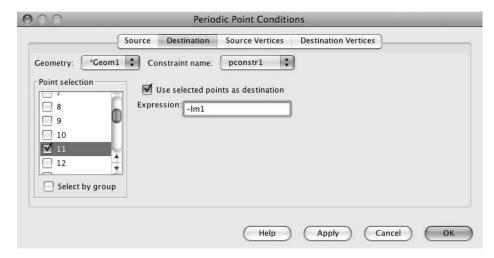


FIGURE 7.135 2D\_ACGS\_1 model Periodic Point Conditions, Destination page (11)

Source Destination	Source Vertices	Destination Vertices
constr1		
Destinatio	n vertices:	
		ientation of the source domains
		ely the destination domains is
		nined by matching source and
	destin	ation vertices in the order they a
	added	•
Ă.		
	constr1	Destination vertices:

FIGURE 7.136 2D\_ACGS\_1 model Periodic Point Conditions, Destination Vertices (11) page

Click the Destination Vertices tab. Click the >> button to select point 11 as a destination vertex. See Figure 7.136. Click OK.

## **Physics Settings: Global Equations**

Using the menu bar, select Physics > Global Equations. In the Global Equations edit windows, enter the information shown in Table 7.25. Click OK. See Figure 7.137.

## Table 7.25 Global Equations Edit Window

Name	Equation f(u,ut,utt,t)	lnit (u)	lnit (ut)	Description	Figure Number
omega	omegatt-(M+Tz)/(I+I0)	0	0	Rotation angle	7.137

Name (u)	Equation f(u,ut,utt,t)	Init (u)	Init (ut)	Description
mega	omegatt-(M+Tz)/(I+I0)	0	0	Rotation angle
			-	
			-	1

**FIGURE 7.137** 2D\_ACGS\_1 model Global Equations edit window

Predefined mesh sizes:     F	iner 🗘	Cance
O Custom mesh size		Appl
Maximum element size:		
Maximum element size scaling factor:	0.55	(Help
Element growth rate:	1.25	
Mesh curvature factor:	0.25	
Mesh curvature cutoff:	0.0005	
Resolution of narrow regions.	1	
Optimize quality		
efinement method: Regular		
ennement metriou. Kegular		

**FIGURE 7.138** 2D\_ACGS\_1 model Free Mesh Parameters edit window

#### **Mesh Generation**

From the toolbar, select Mesh > Free Mesh Parameters > Global. Select Predefined mesh sizes > Finer (from the pull-down list). See Figure 7.138.

Click the Boundary tab. Select boundaries 2, 5–8, 10–12, 14, 19–21. Enter 0.005 in the Maximum element size window. See Figure 7.139.

Global Su	bdomain Boundary Point Advanced	
Boundary selection 9 10 11 12 13 14 15 16 17 18 19 20 21 Select by group Select Remaining	Parameters         Distribution           Boundary mesh parameters         Maximum element size:         0.005           Element growth rate:         Mesh curvature factor:         Mesh curvature factor:           Mesh curvature cutoff:         Mesh curvature cutoff:         Mesh curvature factor:	(Car Ap He
Select Meshed		

**FIGURE 7.139** 2D\_ACGS\_1 model Free Mesh Parameters, Boundary page

## 坐

#### FIGURE 7.140 2D\_ACGS\_1 model Copy Mesh button

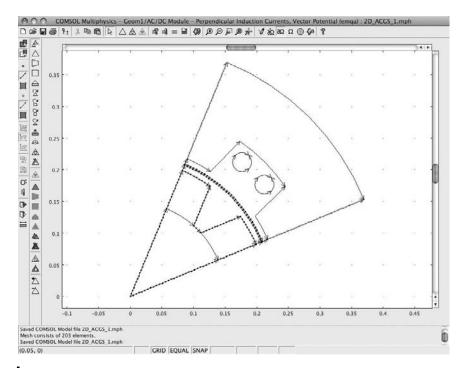
#### Click the Mesh Selected button.

Select boundaries 1 and 2. Click the Copy Mesh button on the Mesh toolbar. See Figure 7.140.

Note The purpose of copying the mesh from one edge to the other edge is to ensure that the proper phase relationship is maintained across the mesh. Otherwise, there might be a mismatch between the edges, which would distort the mesh and possibly result in solution problems.

Select boundaries 3 and 7. Click the Copy Mesh button. Select boundaries 4 and 8. Click the Copy Mesh button. Select boundaries 15 and 19. Click the Copy Mesh button. Select boundaries 16 and 20. Click the Copy Mesh button, and then click OK. See

Figure 7.141.



**FIGURE 7.141** 2D\_ACGS\_1 model copied meshed boundaries

# 8

**FIGURE 7.142** 2D\_ACGS\_1 model Mesh Remaining (Free) button

Click the Mesh Remaining (Free) button on the Mesh toolbar. See Figure 7.142. Figure 7.143 shows the meshed model.

#### Solving the Static 2D\_ACGS\_1 Model

Using the menu bar, Select Solve > Solver Parameters.

Note The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

	General	Stationary	Adaptive	Advanced
ransient 🛟	<u></u>			
Auto select solver	r system solver	Direct (UMFPA	ск)	•
10 m m m m m m m m m m m m m m m m m m m	onditioner:			*
Adaptive mesh refinement	x symmetry:	Automatic		÷

FIGURE 7.144 2D\_ACGS\_1 model Solver Parameters edit window

Select Stationary in the Solver selection window. See Figure 7.144. Click OK. Using the menu bar, select Solve > Solver Manager > Solve For. Unselect the

variable omega under ODE(OD). See Figure 7.145. Click OK.

Select Solve > Solve Problem. Click the Save button.

Select File > Save As. Enter 2D\_ACGS\_2.mph in the Save As edit window. See Figure 7.146.

Click the Save button. See Figure 7.147.

**NOTE** The 2D\_ACGS\_1 model (static) solution was built to gain experience in the creation of a complex geometrical sector model. It was saved as 2D\_ACGS\_2.mph and will act as the initial estimate for the 2D\_ACGS\_2 model (transient) solution.

## 2D AC Generators, Static Sector Model: Summary and Conclusions

The static 2D AC generator sector models has now been built. This model allows the modeler to solve the AC generator model through the use of a simpler geometric representation. It avoids some of the potential problems that might be observed at run

000	Solver Manager
	Initial Value Solve For Output Script
Solve for variables:	
ODE (0D)	
📄 omega	
▼ Perpendicula Az Im1 Moving Mesh	r Induction Currents, Vector Potential (emqa (ALE) (ale)
	) 4 4
	Help Solve Apply Cancel OK

**FIGURE 7.145** 2D\_ACGS\_1 model with omega unselected

9_Models	•
2D_EIS_2_Constants.txt	Date Modified Monday, March 16, 2009 5:41 PM Saturday, March 14, 2009 4:16 PM Monday, March 16, 2009 6:39 AM Tuesday, March 16, 2009 7:42 AM Wednesday, March 18, 2009 7:42 AM Wednesday, March 18, 2009 4:48 PM Friday, March 6, 2009 7:54 AM Wednesday, March 18, 2009 2:40 PM Wednesday, March 18, 2009 2:40 PM Saturday, March 18, 2009 2:40 PM Wednesday, March 18, 2009 2:42 PM Wednesday, March 18, 2009 2:43 PM Wednesday, March 18, 2009 2:43 PM
2D_EIS_2.mph 2D_EIS_2_Constants.txt 2D_EIS_2_SE.txt	Saturday, March 7, 2009 Wednesday, March 18, 20

**FIGURE 7.146** 2D\_ACGS\_2 model Save As edit window

FIGURE 7.147 2D\_ACGS\_2 model initial solution

time. This model generates low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. This model demonstrates the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of a rotating machine for the conversion of mechanical energy to electrical energy.

## 2D AC Generator Sector Model (2D\_ACGS\_2): Transient

The numerical solution model presented in this subsection (2D\_ACGS\_2) follows directly from the earlier 2D\_ACGS\_1 model, which we just built. In this subsection, the transient 2D generator sector model (2D\_ACGS\_2) utilizes the static 2D\_ACGS\_1 model as the initial solution to the transient problem. The new version avoids all the complex geometrical building by starting with the earlier saved solution.

To start building the 2D\_ACGS\_2 model (transient) solution, activate the COMSOL Multiphysics software. In the Model Navigator, click the Open tab. Select "2D\_ACGS\_2.mph." See Figure 7.148.

Click OK. See Figure 7.149.

00	Model Navigator
	New Model Library User Models Open Settings
	9_Models
Description:	Name       Date Modified         2D_ACG_1.mph       Monday, March 16, 2         2D_ACG_1_Geo       Monday, March 16, 2         2D_ACG_1_Geo       Monday, March 16, 2         2D_ACG_1_Geo       Monday, March 16, 2         2D_ACG_2.mph       Tuesday, March 16, 2         2D_ACGS_1.mph       Saturday, March 17,         2D_ACGS_1_Con       Yednesday, March 12,         2D_ACGS_2.mph       Saturday, March 6, 200         2D_EIS_1.mph       Friday, March 6, 200         2D_EIS_1_SE.txt       Wednesday, March 1         2D_EIS_2.mph       Saturday, March 7, 2         2D_EIS_2.mph       Saturday, March 1         2D_EIS_2.mph       Saturday, March 1         2D_EIS_2.sE.txt       Wednesday, March 1         2D_EIS_2.SE.txt       Wednesday, March 1
	File Format: COMSOL Multiphysics Mo

FIGURE 7.148 2D\_ACGS\_2 Model Navigator, initial solution selection

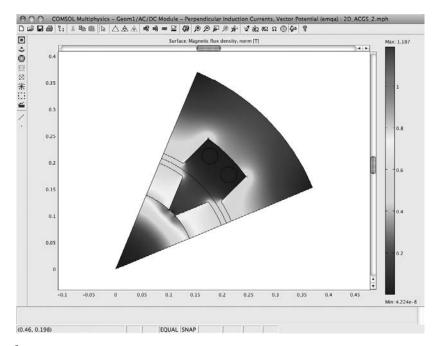


FIGURE 7.149 2D\_ACGS\_2 Model Navigator, initial solution

NOTE Because the initial solution to the 2D\_ACG\_2 model (transient) has already been built and verified, the modeler can proceed directly to implementing the necessary transient solver setup parameters.

#### Solving the Transient 2D\_ACGS\_2 Model

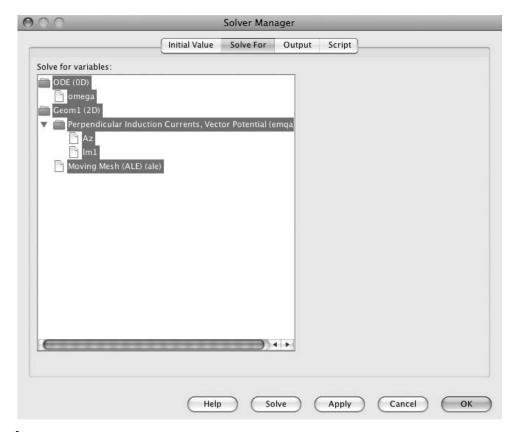
Using the menu bar, select Solve > Solver Parameters. select "Transient" from the Analysis pull-down list.

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

Select "Time dependent" in the Solver selection window. In the Times edit window, enter linspace(0,2.6,131). For later versions of COMSOL Multiphysics software enter range(0,2.6/130,2.6). In the Absolute tolerance edit window, enter Az 1e-3 lm1 5e3 omega 0.015. Click the Apply button. See Figure 7.150.

000	Solver Parameters	
Analysis: Transient		me Stepping Advanced
Auto select solver	Time stepping	linspace(0,2.6,131)
Stationary Time dependent Eigenvalue Parametric	Relative tolerance: Absolute tolerance:	0.01 Az 1e-3 lm1 5e3 omega 0.015
Stationary segregated Parametric segregated	Linear system solver Linear system solver: Direct (UI Preconditioner:	MFPACK)
	Matrix symmetry: Automati	c Settings
	Help	Apply Cancel 0

FIGURE 7.150 2D\_ACGS\_2 model Solver Parameters edit window



**FIGURE 7.151** 2D\_ACGS\_2 model Solver Manager, Solve For page

NOTE It is important to enter the solver parameters *exactly* as specified: otherwise, the modeler will see error messages.

Click OK. Using the menu bar, select Solve > Solver Manager > Solve For. See Figure 7.151.

Select all variables. Click OK.

Using the menu bar, select Solve > Restart. See Figure 7.152.

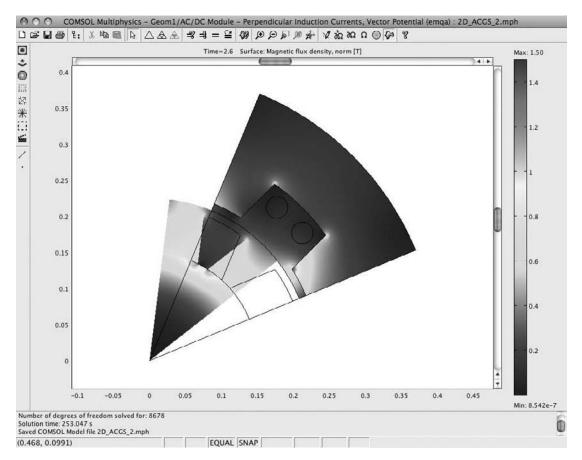
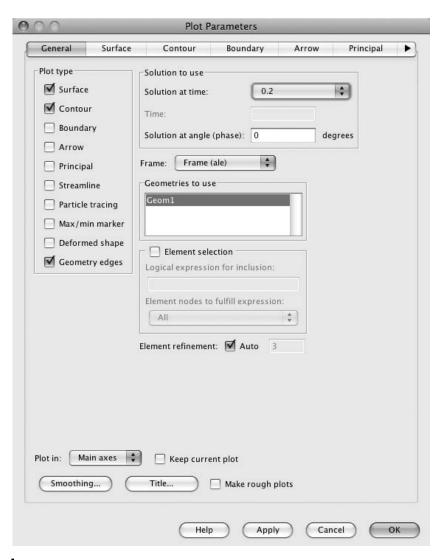


FIGURE 7.152 2D\_ACGS\_2 model solution default plot, final frame

#### **Postprocessing and Visualization**

Select Postprocessing > Plot Parameters > General. Check the Surface and Contour check boxes under Plot type. Select "Solution at time 0.2 seconds" from the Solutions to use pull-down list. See Figure 7.153.



**FIGURE 7.153** 2D\_ACGS\_2 model plot Parameters, General tab

Click the Surface tab. Select "Magnetic flux density, norm." See Figure 7.154.

Click the Contour tab. Enter Az in the Expression edit window. Enter 15 in the Levels edit window. Click the Uniform color radio button. Click the Color button, and select black. See Figure 7.155.

Surface plot	
- Surface plot	Surface Data Height Data
	Surface Data Height Data
Predefined quantities:	Magnetic flux density, norm 🗘 🤇 Range
Expression:	normB_emqa 🗹 Smooth
Unit:	T
oloring and fill	
	olated 🗘 Fill style: Filled 🗘
Coloring: Interpo	olated 🛟 Fill style: Filled 🛟
Coloring: Interpo	
Coloring and fill Coloring: Interpo Gurface color O Colormap:	jet Colors: 1024 Color scale
Coloring: Interpo	
Coloring: Interpo Surface color Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo iurface color Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo iurface color Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo urface color O Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo urface color O Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo urface color O Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo urface color O Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo urface color O Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo urface color O Colormap:	jet 🗘 Colors: 1024 🗹 Color scale
Coloring: Interpo urface color O Colormap:	jet 🗘 Colors: 1024 🗹 Color scale

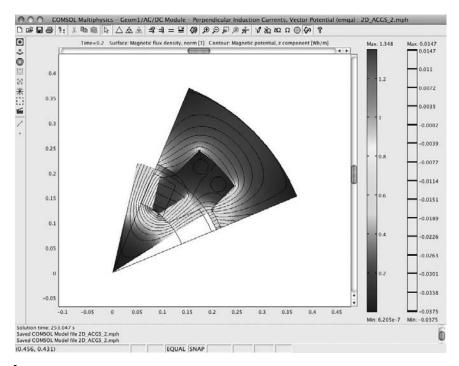
**FIGURE 7.154** 2D\_ACGS\_2 model Plot Parameters, Surface tab

0.0	Plot P	arameters		
General Surface	Contour	Boundary	Arrow	Principal
Contour plot	ntour Data Heigl	nt Data Color	Data	
	nour Data Heigi	it Data Color	Data	
Predefined quantities:	Magnetic potent	tial, z com 🛟	)	
Expression:	Az		Smoot	n
Unit:	Wb/m	\$	)	
Contour levels				
~	ber of levels	Vector with iso	levels	
Levels: 💿 15	0			-
Labels				
Contour color				
Colormap:	cool 🛟	Colors: 1024	Co	lor scale
Uniform color:	Color			
Filled				

**FIGURE 7.155** 2D\_ACGS\_2 model Plot Parameters, Contour tab

Click OK. See Figure 7.156.

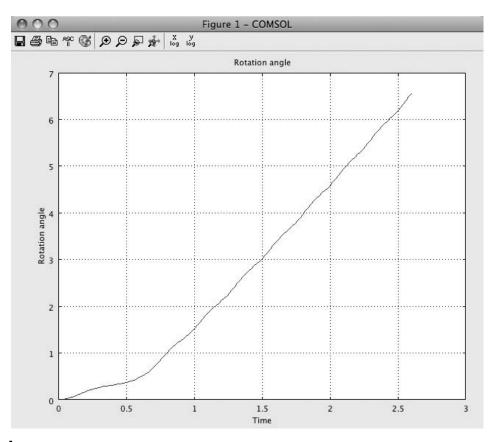
To view the rotation angle as a function of time, select Postprocessing > Global Variables Plot. Select "Rotation angle" from the Predefined quantities list. Click the > button. See Figure 7.157.



**FIGURE 7.156** 2D\_ACGS\_2 model magnetic flux density and magnetic potential, *z* component

Predefined quantities -		Quantities to plot
Rotation angle Total torque for the e Electromagnetic force Electromagnetic force Electromagnetic torqu	(	Rotation angle
xpression:		
omega		
0 0.02 0.04 0.06	utput times	Auto     Expression     Line Settings     Title/Axis
0.1	ţ	Plot in: New figure
0.08 0.1 0.12 Times:	\$	Plot in: New figure

FIGURE 7.157 2D\_ACGS\_2 model Global Variables Plot edit window



**FIGURE 7.158** 2D\_ACGS\_2 model global variables plot

Click the Apply button. See Figure 7.158. Click OK.

#### **Postprocessing Animation**

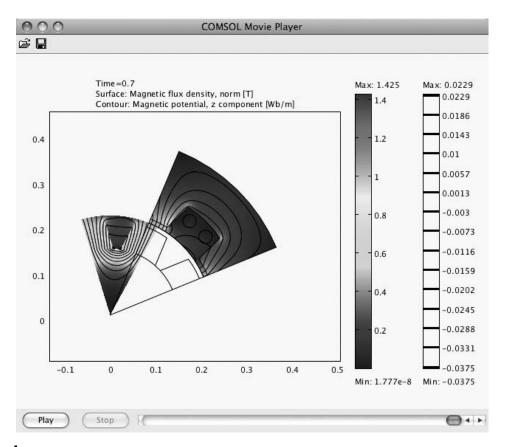
Select Postprocessing > Plot Parameters > Animate. Select the solutions from 0.0 to 0.7 in the Solutions to use window. See Figure 7.159.

Click the Start Animation button. See Figure 7.160.

Alternatively, you can play the file Movie7\_ACGS\_2.avi that was supplied with this book.

Solutions to use Select via: Stored output times 🛟 U.64 0.66
0.66
0.66
0.66
0.68
0.7
0.72
0.74
0.76
0.8
0.82
Times:
times.
Start Animation

**FIGURE 7.159** 2D\_ACGS\_2 model Plot Parameters, Animate tab



**FIGURE 7.160** 2D\_ACGS\_2 model animation, final frame

#### 2D AC Generators, Static and Transient Models: Summary and Conclusions

The two 2D AC generator sector models—static and transient—have now been built. These models generate low-frequency (60 Hz) current and voltage, as would be typically found on the power transmission grid. These models demonstrate the use of both hard (not easily magnetized) and soft (easily magnetized) nonlinear magnetic materials in the construction of a rotating machine for the conversion of mechanical energy to electrical energy.

The modeling and physics concepts employed in this section of Chapter 7 include addition of an ordinary differential equation (ODE), mechanical to electrical energy conversion, hard and soft nonlinear magnetic materials, mesh mapping, moving mesh (ale), and geometric assembly (pair creation across a boundary).

#### References

- 1. http://en.wikipedia.org/wiki/Electrical\_resistivity\_tomography
- 2. http://en.wikipedia.org/wiki/Industrial\_process\_imaging
- 3. http://en.wikipedia.org/wiki/Electrical\_impedance\_tomography
- 4. http://en.wikipedia.org/wiki/Alternating\_current
- P. Manwaring et al., Arbitrary Geometry Patient Interfaces for Breast Cancer Detection and Monitoring with Electrical Impedance Tomography, *Proceedings* of the 30th Annual International IEEE EMBS Conference, 2008, pp. 1178–1180.
- 6. X. Chen et al., Lung Ventilation Monitoring Incorporating Prior Information by Electrical Impedance Tomography, *Proceedings of I2MTC 2008: IEEE International Instrumentation and Measurement Technology Conference*, 2008, pp. 1531–1536.
- 7. D. S. Holder, Electrical Impedance Tomography of Brain Function, *Proceedings* of Automation Congress 2008, pp. 1–6.
- 8. http://en.wikipedia.org/wiki/Generators
- 9. http://en.wikipedia.org/wiki/Ordinary\_differential\_equation
- 10. http://en.wikipedia.org/wiki/Electrical\_impedance
- 11. http://en.wikipedia.org/wiki/Ohm%27s\_law
- 12. http://en.wikipedia.org/wiki/Oliver\_Heaviside
- 13. http://en.wikipedia.org/wiki/Arthur\_Kennelly
- 14. W. T. Scott, *The Physics of Electricity and Magnetism* (2nd ed.), John Wiley and Sons, 1966, Chapter 9.2–9.4, pp. 461–485.
- 15. http://en.wikipedia.org/wiki/Angular\_frequency
- 16. http://en.wikipedia.org/wiki/Reactance\_(electronics)
- 17. http://en.wikipedia.org/wiki/Skin\_depth
- 18. http://en.wikipedia.org/wiki/Permittivity
- 19. http://en.wikipedia.org/wiki/Michael\_Faraday
- 20. http://en.wikipedia.org/wiki/Electromagnetic\_induction
- 21. http://en.wikipedia.org/wiki/Thomas\_Alva\_Edison
- 22. http://en.wikipedia.org/wiki/George\_Westinghouse
- 23. http://en.wikipedia.org/wiki/Nikola\_Tesla
- 24. http://en.wikipedia.org/wiki/War\_of\_the\_currents
- 25. http://en.wikipedia.org/wiki/Joule%27s\_Law

- 26. http://en.wikipedia.org/wiki/Transformer
- 27. *COMSOL Multiphysics Modeling Guide*, Using Assemblies, Version 3.4, October 2007, COMSOL AB, Stockholm, Sweden, pp. 351–367.
- 28. http://en.wikipedia.org/wiki/Ordinary\_differential\_equation

#### Exercises

- 1. Build, mesh, and solve the basic 2D electric impedance sensor model problem presented in this chapter.
- 2. Build, mesh, and solve the advanced 2D electric impedance sensor model problem presented in this chapter.
- 3. Build, mesh, and solve the static 2D AC generator model (2D\_ACG\_1) problem presented in this chapter.
- 4. Build, mesh, and solve the transient 2D AC generator model (2D\_ACG\_2) presented in this chapter.
- 5. Build, mesh, and solve the static 2D AC generator sector model (2D\_ACGS\_1) presented in this chapter.
- 6. Build, mesh, and solve the transient 2D AC generator sector model (2D\_ACGS\_2) presented in this chapter.
- 7. Explore other materials as applied in the 2D electric impedance sensor models.
- 8. Explore other materials as applied in the 2D AC generator models.
- 9. Explore adding more turns to the 2D AC generator models.
- 10. Explore how the 2D electric impedance sensor model might be used to discover voids in boats, airplanes, bridges, and other areas.

# 8

# 3D Modeling

## In This Chapter

3D Modeling Guidelines for New COMSOL® Multiphysics® Modelers
3D Modeling Considerations
3D Coordinate System
Electrical Resistance Theory
Thin Layer Resistance Modeling Basics
3D Thin Layer Resistance Model: Thin Layer Approximation
3D Thin Layer Resistance Model, Thin Layer Approximation:
Summary and Conclusions
3D Thin Layer Resistance Model: Thin Layer Subdomain
3D Thin Layer Resistance Models: Summary and Conclusions
Electrostatic Modeling Basics
3D Electrostatic Potential Between Two Cylinders
3D Electrostatic Potential Between Two Cylinders: Summary and Conclusions
3D Electrostatic Potential Between Five Cylinders
3D Electrostatic Potential Between Five Cylinders: Summary and Conclusions
Magnetostatic Modeling Basics
3D Magnetic Field of a Helmholtz Coil
3D Magnetic Field of a Helmholtz Coil: Summary and Conclusions
3D Magnetic Field of a Helmholtz Coil with a Magnetic Test Object
3D Magnetic Field of a Helmholtz Coil with a Magnetic Test Object:
Summary and Conclusions

# **3D** Modeling Guidelines for New COMSOL<sup>®</sup> Multiphysics<sup>®</sup> Modelers

### **3D Modeling Considerations**

In this chapter on 3D modeling, all the basic material on 2D modeling presented in the earlier chapters will be assumed, utilized, and expanded. In the earlier chapters, models were built and solved using static, quasi-static, and transient methods. In this chapter,

the methods employed will be either static or quasi-static. The level of model solution complexity (difficulty) is increased through the development of models that explore applied physics at a more realistic and difficult level. In the three 3D models developed in this chapter, three modeling concept areas are explored: large dimensional differences, electrostatic field mapping, and magnetostatic field mapping. Each of these areas has broad industrial and scientific modeling applicability and potential levels of complexity.

The 3D models in this chapter implicitly assume, in compliance with the laws of physics, that the energy flow, the materials properties, the environment, and any other conditions and variables of interest are homogeneous, isotropic, or constant, unless otherwise specified (e.g., time dependent), throughout the entire domain of interest, both within the model and, through the boundary conditions, in the environs of the model.

Three models and variations are presented here: the 3D thin layer resistance models, the 3D electrostatic potential between cylinders models, and the 3D magnetic field of a helmholtz coil models. The first two models are developed using application modes from the basic COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software. The Helmholtz coil model requires the AC/DC Module. Each of these three models introduces the modeler to different modeling aspects in the employment of the basic COMSOL Multiphysics software and the AC/DC Module to explore a range of diverse design, test, and engineering problems.

The 3D thin layer resistance models explore the modeling of a technology that is widely employed in both research and applied development for science and industry. Both thin<sup>1,2</sup> and thick<sup>3</sup> layers (coatings) are widely applied. Layers, such as those modeled here or other layers that may inadvertently or unknowingly form, can significantly modify the overall performance (behavior) of a device structure.

When building a model, the modeler should perform at least a first estimate review of the conditions to which the modeled structure will be exposed under the normal (extreme) conditions of use. In that review, a variety of questions should be asked, including these possibilities: Will corrosion films form? Will any of the structure melt? Will any of the films exhibit a structural transition? Will any of the films exhibit an electronic/magnetic properties shift?

#### **3D Coordinate System**

In a steady-state solution to a 3D model, parameters can vary only as a function of position in space (x), space (y), and space (z) coordinates. Such a 3D model represents the parametric condition of the model in a time-independent mode (quasi-static).

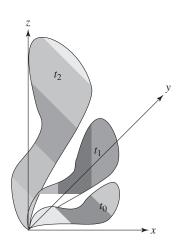


FIGURE 8.1 3D coordinate system, plus time

In a transient solution model, parameters can vary both by position in space (x), space (y), space (z) and in time (t). See Figure 8.1.

The transient solution model is essentially a sequential collection of (quasi-static) solutions, except that one or more of the dependent variables [f(x, y, z, t)] has changed with time. The space coordinates (x), (y), and (z) typically represent a distance coordinate throughout which the model is to calculate the change of the specified observables (i.e., temperature, heat flow, pressure, voltage, current) over the range of values  $(x_{\min} \le x \le x_{\max})$ ,  $(y_{\min} \le y \le y_{\max})$ , and  $(z_{\min} \le z \le z_{\max})$  The time coordinate (t) represents the range of values  $(t_{\min} \le t \le t_{\max})$  from the beginning of the observation period  $(t_{\min})$  to the end of the observation period  $(t_{\max})$ .

#### **Electrical Resistance Theory**

A well-known example of the application of thin layer technology is the touch screen,<sup>4</sup> which is widely used in computers, personal digital assistants (PDAs), electronic lock pads, and other devices. The fundamental concept of the touch screen is relatively simple. The underlying touch screen principle starts with Ohm's law.

NOTE Ohm's law was discovered by Georg Ohm and published in 1827:

$$I = \frac{V}{R} \tag{8.1}$$

where I = current in amperes (A) V = voltage (electromotive force) in volts (V)R = resistance in ohms

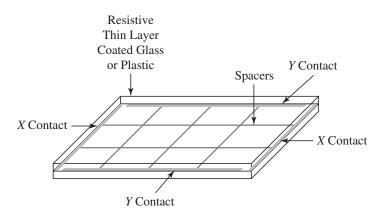


FIGURE 8.2 3D touch screen geometry

This technology utilizes the concept of an electrical resistance<sup>5</sup> divider<sup>6</sup> (voltage divider) to locate the point of contact (touch). When implemented in four-wire touch screen technology, this technology employs two thin layer orthogonal voltage dividers. In the touch screen technology, the thin layer resistive sheets are coated onto an insulating glass or plastic substrate. The substrates are mounted with the thin resistive layers facing each other and separated by an array of thin insulating dots, insulating bars, or a similar porous insulating spacer array. See Figure 8.2.

When pressure (touch) is applied to the screen, a point of contact forms at that location. See Figure 8.3.

The voltage  $V_{XT}$  is measured as shown in Figure 8.3 (measurement circuitry not shown).  $V_{YT}$  is similarly measured sequentially. The X location of the contact point is

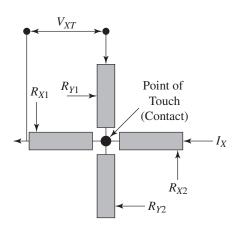


FIGURE 8.3 3D touch screen divider circuit

determined as follows:

$$V_{XT} = I_X * R_{X1} : V_{\text{TOTAL}} = I_X * (R_{X1} + R_{X2})$$
(8.2)

where

 $I_X = X$  current in amperes (A)  $V_{XT} =$  voltage drop (electromotive force) in volts (V)  $V_{\text{TOTAL}} =$  total voltage (electromotive force) along X in volts (V)  $R_{X1} =$  divider resistance 1 in ohms  $R_{X2} =$  divider resistance 2 in ohms

since

$$R = \frac{\rho L}{A} \tag{8.3}$$

where R = resistance in ohms

 $\rho$  = resistivity in ohm\*m

L =length of the resistive material (m)

A =cross-sectional area in meters squared (m<sup>2</sup>)

Thus the length (distance) to the contact point is

$$\frac{V_{XT}}{V_{\text{TOTAL}}} * (L) = \frac{R_{X1}}{R_{X1} + R_{X2}} * (L) = \frac{L_{X1}}{L_{X1} + L_{X2}} * (L)$$
(8.4)

where

 $V_{XT}$  = voltage drop (electromotive force) in volts (V)

 $V_{\text{TOTAL}}$  = total voltage (electromotive force) along X in volts (V)

 $R_{X1}$  = divider resistance 1 in ohms

 $R_{X2}$  = divider resistance 2 in ohms

 $L_{X1}$  = resistor length 1 in meters (m)

 $L_{X2}$  = resistor length 2 in meters (m)

 $L = L_{X1} + L_{X2}$  in meters (m)

#### Thin Layer Resistance Modeling Basics

The first example presented here, the 3D thin layer resistance model, thin layer approximation (3D\_TLR\_1 model), explores the use of the thin layer approximation in the solution of a direct current conduction model. In the problem explored in both this model and the model to follow, the current balance throughout the domains is described as follows:

$$V \cdot \left(-\sigma \nabla V\right) = 0 \tag{8.5}$$

where V = electric potential (electromotive force) in volts (V)

 $\Delta = \text{gradient}$ 

 $\sigma$  = electrical conductivity in siemens per meter (S/m)

In the thin layer approximation model, it is assumed that the x and y components of the current density vector are sufficiently small in the thin layer that only the z component makes a contribution. Thus

$$-\sigma * \frac{d^2 V}{dz^2} = 0 \tag{8.6}$$

where V = electric potential (electromotive force) in volts (V)  $\sigma =$  electrical conductivity in siemens per meter (S/m)

By substitution, it can be seen that

$$V = \alpha * z + \beta \tag{8.7}$$

where  $\alpha = \text{constant (V/m)}$  $\beta = \text{constant (V)}$ 

Equation 8.7 is one possible solution for Equation 8.6.

NOTE Considering the following:

$$\frac{dV}{dz} = \alpha : \frac{d^2V}{dz^2} = 0 \tag{8.8}$$

Assuming that

$$V_{\text{lower}} = V_1 \colon V_{\text{upper}} = V_2 \tag{8.9}$$

then for z = 0:

$$\beta = V_1 \tag{8.10}$$

For  $z = \delta$ :

$$\alpha = \frac{V_2 - V_1}{\delta} \tag{8.11}$$

where  $\delta$  = thickness of the thin layer in meters (m).

Because

$$J_Z = -\sigma * \frac{dV}{dz} = -\sigma * \alpha = -\sigma * \left(\frac{V_2 - V_1}{\delta}\right)$$
(8.12)

and there are no sources or sinks,  $J_z$  is the current flow through the system.

NOTE The use of the thin layer approximation is applicable to any problem in which flow is described by the divergence of a gradient flux (i.e., diffusion, heat conduction, flow through porous media under Darcy's law).

The application of the thin layer approximation is especially valuable to the modeler when the differences in domain thickness are so great that the mesh generator is unable to properly mesh the model or creates more elements than the modeling platform can handle (the "run out of memory" problem). In those cases, this approximation may enable a model to be solved that would otherwise fail.

#### **3D Thin Layer Resistance Model: Thin Layer Approximation**

The following numerical solution model (3D\_TLR\_1 model) is derived from a model that was originally developed by COMSOL as a Multiphysics Electromagnetics demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL Multiphysics Model Library.

To start building the 3D\_TLR\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. See Figure 8.4. Click OK.

989	Ma	del Navigator			
New	Model Library	User Models	Open	Settings	
Space dimension:	3D	\$	E la stra		
Application Modes	85.47.58.	M	Electro	omagnetics	
<ul> <li>COMSOL Multiphy</li> <li>Acoustics</li> </ul>	sics				
<ul> <li>Convection at</li> </ul>	nd Diffusion		0		
🔻 🚞 Electromagne					
Conductiv	re Media DC			1 State	
<ul> <li>Fluid Dynamic</li> </ul>		U.	Deceminate		
▶ 🚞 Heat Transfer			Descriptio Electrosta	n: tics of conducting	material
Structural Methods           Image: Structural Methods           Image: Structural Methods	chanics				
Deformed Met	sh	¥ .			
▶ 🚞 Electro-Therr	nal Interaction	•			
Dependent variables:	V				
Application mode name:	dc				
Element:	Lagrange - Qu	uadra 🛟	_	Multiphysics	

FIGURE 8.4 3D\_TLR\_1 Model Navigator setup

Name	Style	Base	Length (X, Y, Z)	Axis Base Point (X, Y, Z)	Figure Number
BLK1	Solid	Corner	(1, 1, 0.1)	(0, 0, 0)	8.5
BLK2	Solid	Corner	(1, 1, 0.1)	(0, 0, 0.1)	8.6

Table 8.1	Geometry	Components
-----------	----------	------------

#### **Geometry Modeling**

Using the menu bar, select Draw > Block. In the Block edit window, enter the information shown in Table 8.1. Click OK after filling in the parameters of each separate block in the Block edit window. See Figures 8.5 and 8.6.

		Block	_	_	12
Style	Base	Axis base point		tation an	
Solid	🕑 Corner	x: 0	α:	0	(degrees
O Face	O Center	y: 0			
Length		z: 0			
X: 1		Axis direction vector			
Y: 1		Cartesian coordina	tes 🔘	Spherica	l coordinates
Z: 0.1		x: 0	θ:	0	(degrees)
		y: 0	φ:	0	(degrees)
lame: BLH	(1	z: 1	1		

FIGURE 8.5 3D\_TLR\_1 model BLK1 edit window

000	_		Block	-		-
Style Solid Face	Base Corner Center		s base point		Rotation ar x: 0	gle (degrees)
Length X: 1 Y: 1		1000	0.1 s direction vector- ) Cartesian coordin	nates (	) Spherica	l coordinates
Z: 0.1		x:	0	Θ:	0	(degrees)
Name: BLK2		y: z:	0 1 Help Ap	φ:	Cancel	(degrees)

**FIGURE 8.6** 3D\_TLR\_1 model BLK2 edit window

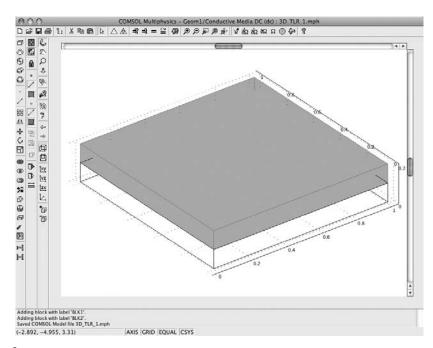


FIGURE 8.7 3D\_TLR\_1 model BLK2

Click the Zoom Extents button. See Figure 8.7.

Select File > Save As. Enter 3D\_TLR\_1 in the Save As edit window. See Figure 8.8. Click the Save button.

Using the menu bar, select Draw > Work-Plane Settings. See Figure 8.9.

10_Models	\$
Name 🔺	Date Modified
[P_Cylinders_1a.mph [P_Ellipsoids_2a.mph [P_Sph_Ellipsoids_3a.mph [Coil_1a.mph [Coil_3a.mph [Coil_3a.mph 2.P_E_P_1a.mph [Thin_Film_Resistance_1a.mph [Thin_Film_Resistance_7a.mph	Wednesday, April 1, 2009 8:02 AM Wednesday, April 1, 2009 2:15 PM Wednesday, April 1, 2009 2:15 PM Thursday, April 2, 2009 8:37 AM Thursday, April 2, 2009 8:37 AM Friday, April 2, 2009 7:49 PM Friday, April 1, 2009 5:45 PM Tuesday, March 31, 2009 3:36 PM Tuesday, March 31, 2009 5:05 PM

FIGURE 8.8 3D\_TLR\_1 model Save As edit window

	a — — —		ine Setting		
Quick	Face Parallel	Edge Angle	Vertices	Advanced	_
Plane					ОК
● х-у z:	0				Cance
() y-z ×:	0				Apply
() z-x γ:	0				Help
					Preview

**FIGURE 8.9** 3D\_TLR\_1 model Work-Plane Settings edit window

**NOTE** The use of the Work-Plane Settings is intended to make specific 2D planes available to the modeler to facilitate the creation of essentially 2D objects in the 3D geometry.

100						_	-	-	_							-
1				'	,		, <del>–</del>							r	,	
1	10.													18. 1	2	
0.9 - •	÷	×	1	- 41	÷	31 <sup>1</sup>		142	2	*		4	-	ж. Эй	э¥	
0.8 -		ŝ	2		*	8.			ŝ	*		e.	÷.	÷	3	
0.7	10			(19)) (19))	*	21.	5		e)	2		e.		20	1	
0.6 -						811			8	$\mathbf{x}$	3	÷	0	$\sim$	×	
0.5	*	12	-	141	÷			4	23	Ξ.			3	22	9	
0.4 - •	4		1	•	4	2	•	•		-				2	3	
0.3 - •		17		30		1	<u>.</u>		1	5	÷	2		2	a.	
0.2	+	+	2	(4)	÷	(e)	÷	e.	R	+	(4)			÷	+	
0.1	142			390	÷.	31 I		C42	17	30	241	1		2	х.	
0			-	- 100						0			_			
-0.3	-0.2	-0.1	0	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1	1.1	1.2	

Click OK, using the default settings. Click the Zoom Extents button. See Figure 8.10.

**FIGURE 8.10** 3D\_TLR\_1 model Geom2 work-plane

Size		Rotat	ion angle	
Radius:	0.6	α:	0	(degrees)
Position				
Base:	Center	Style:	Solid	\$
x:	0	Name:	C1	
	1	_		

FIGURE 8.11 3D\_TLR\_1 model Circle edit window

Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.6, set the Base as Center, and set x equal to 0 and y equal to 1. See Figure 8.11. Click OK.

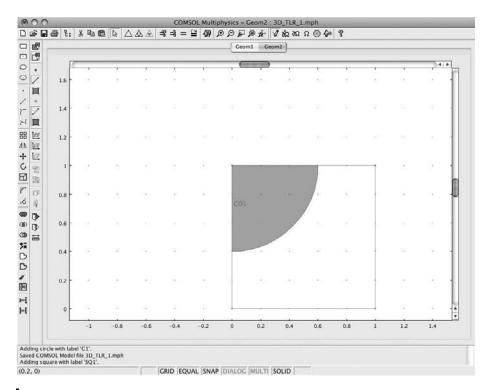
Using the menu bar, select Draw > Specify Objects > Square. Enter a width of 1, set the Base as Corner, and set x equal to 0 and y equal to 0. See Figure 8.12. Click OK.

Using the menu bar, select Draw > Create Composite Object.

Enter C1\*SQ1 in the Set formula edit window. See Figure 8.13.

Size		Rotat	ion angle	
Width:	1	] α:	0	(degrees)
Positio	n			
Base:	Corner	Style:	Solid	•
x:	0	Name:	SQ1	
y:	0	1		

**FIGURE 8.12** 3D\_TLR\_1 model Square edit window



**FIGURE 8.14** 3D\_TLR\_1 model CO1 (intersection of circle and square)

NOTE The formula X \* Y creates the intersection of X and Y.

Click OK. See Figure 8.14.

Using the menu bar, select Draw > Embed. See Figure 8.15. Click OK.

Objects to eml	bed:
Embed to geor	materia
Geom1	¢
	1000
Embedded obj	ject name:

FIGURE 8.15 3D\_TLR\_1 model Embed edit window

0	0.0	Move	
Dis	placement		ОК
x:	0		UK
y:	0		Cancel
z:	0.2		

FIGURE 8.16 3D\_TLR\_1 model Move edit window

As is obvious, the quarter-circle electrode needs to be moved to the upper surface of the upper block. Using the menu bar, select Draw > Modify > Move. Enter x = 0, y = 0, and z = 0.2. See Figure 8.16.

Click OK. See Figure 8.17.

Select EMB1 and BLK2 (click on EMB1 and then Shift-click on BLK2). See Figure 8.18. Using the menu bar, select Draw > Coerce To > Solid.

Using the menu bar, select Draw > Create Pairs. Select BLK1 and CO1. See Figure 8.19.

Click OK. See Figure 8.20.

#### Physics Subdomain Settings: Conductive Media DC (dc)

Having established the geometry for the 3D\_TLR\_1 model of two blocks, an electrode, and an identity paired interface, the next step is to define the fundamental

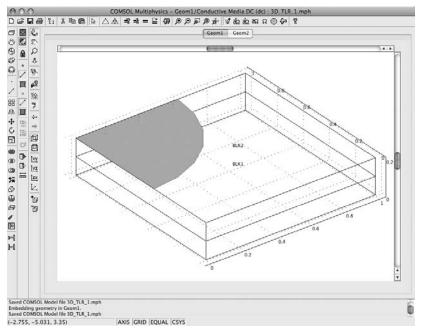


FIGURE 8.17 3D\_TLR\_1 model EMB1 on top of block BLK2

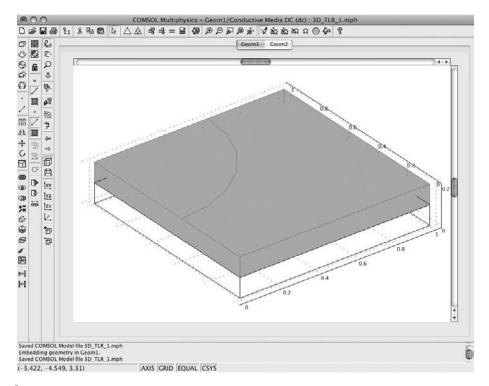


FIGURE 8.18 3D\_TLR\_1 model with EMB1 and BLK1 selected

Physics conditions. Using the menu bar, select Physics > Subdomain Settings. Select subdomains 1 and 2 in the Subdomain selection window (the only available subdomains). In the Subdomain edit windows, enter the information shown in Table 8.2; also see Figure 8.21. Click OK.

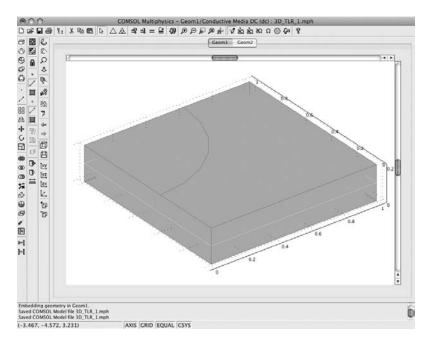
Create Pairs	
Object selection: BLK1 CO1	OK Cancel Apply
Pair type: Identity pair 🛟	Help
Repair tolerance: 1.0E-4	

FIGURE 8.19 3D\_TLR\_1 model Create Pairs edit window

Name	Expression	Description
σ	1	Electrical conductivity

**Subdomain Edit Window** 

Table 8.2



#### **FIGURE 8.20** 3D\_TLR\_1 model identity pair

Subdomains Groups		Physics Init	Element Color	L
ubdomain selection	Material properties and	l sources		
1	Library material:	:	ad)	
	Quantity	Value/Expression	Unit	Description
	Je	0 0	0 A/m <sup>2</sup>	External current density
	Qj	0	A/m <sup>3</sup>	Current source
	Conductivity relation:	Conductivity	:	
	σ	1	5/m	Electric conductivity
Group:				
Select by group				
Active in this domain				

FIGURE 8.21 3D\_TLR\_1 model Subdomain Settings edit window

Boundary	<b>Boundary Condition</b>	Value/Expression	Figure Number
1, 2, 4–10, 12, 13	Electric insulation	—	8.22
3	Inward current flow	0.3	8.23
11	Ground	_	8.24

#### Table 8.3 Boundary Settings – Conductive Media DC (dc) Edit Window

#### Physics Boundary Settings: Conductive Media DC (dc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 8.3. See Figures 8.22, 8.23, and 8.24.

Click the Pairs tab. Select "Contact resistance" from the Boundary condition pulldown list. For the indicated quantity, select or enter the given value as shown in Table 8.4; also see Figure 8.25. Click OK.

#### Table 8.4 Boundary Settings – Conductive Media DC (dc) Edit Window, Pairs

Quantity	Value/Expression	Description
σ	1e-2	Electrical conductivity
d	0.02	Thickness in meters

ation	
I = 0	
Boundaries Groups Pairs	Conditions Color
oundary selection	Boundary sources and constraints
;	Library material:
5	
5	Boundary condition: Flectric insulation
10	
12	
iroup:	
Select by group	
Interior boundaries	

FIGURE 8.22 3D\_TLR\_1 model Boundary Settings (1, 2, 4–10, 12, 13) edit window

Boundaries Groups Pairs	L	Conditions Cold	r
Boundary selection	Boundary sources ar	id constraints	_
1	Library material:	( Load )	
2	-		
4	Boundary condition:	Inward current flow	
5	Quantity	Value/Expression	Unit Description
6	J <sub>n</sub>	0.3	A/m <sup>2</sup> Normal current densit
8			
9 .			
Group:			
Select by group			
Interior boundaries			

FIGURE 8.23 3D\_TLR\_1 model Boundary Settings (3) edit window

**NOTE** This is the most important step in this model, as it implements the thin layer approximation. By using the identity pair-contact resistance approximation in this model, the modeler has eliminated the necessity of building and using a third domain as the interface layer.

Boundaries Groups P	lairs	Conditions	olor ]	
lair selection Pair-1 (identity)	Boundary sources ar Library material: Boundary condition: Quantity σ d	Load	Unit I	<b>Description</b> Electric conductivity Thickness
Active pair				

FIGURE 8.25 3D\_TLR\_1 model boundary pairs value edit window

#### **Mesh Generation**

From the toolbar, select Mesh > Initialize Mesh. See Figure 8.26.

#### Solving the 3D\_TLR\_1 Model

Using the menu bar, select Solve > Solve Problem. See Figure 8.27.

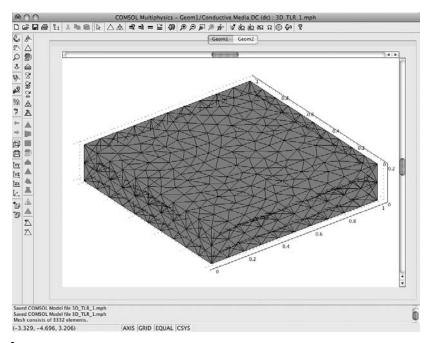
**NOTE** The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

#### Postprocessing and Visualization

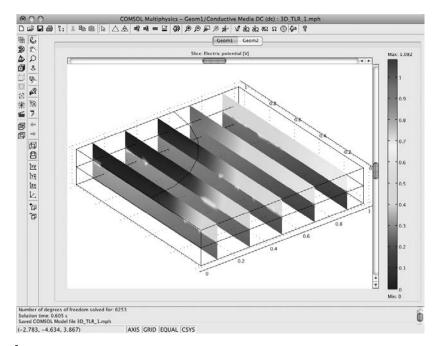
The default plot shows a slice plot of the electric potential (V) distribution in volts. To visualize the solution as a boundary plot, the Plot Parameters will need to be modified.

Select Postprocessing > Plot Parameters > General. In the Plot type list, unselect the Slice check box. In the Plot type list, select the Boundary check box.

Click the Boundary tab. Select Conductive Media DC (dc) > Electric potential (V). Unselect the Smooth check box. See Figure 8.28.







**FIGURE 8.27** 3D\_TLR\_1 model solution, default slice plot

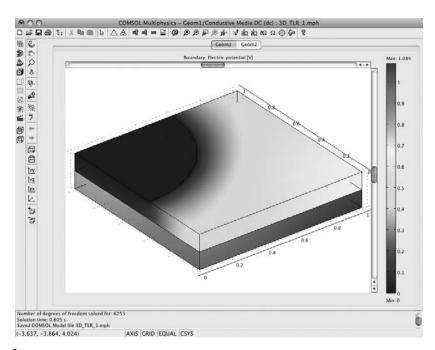
	Plot	Parameters		
General Slice	Isosurface	Subdomain	Boundary	Edge
Boundary plot				
Boundary data				
Predefined quantities:	Electric potent	ial	Range	
Expression:	V		Smooth	
Unit:	v		•	
Coloring and fill				
Coloring: Interp	olated	Fill style: (	Filled	
Boundary color				
Olormap:	jet 🗘	Colors: 1024	Color sca	ale
O Uniform color:	Color			

FIGURE 8.28 3D\_TLR\_1 model solution, Plot Parameters edit window, Boundary tab

Click OK. See Figure 8.29.

#### **TLR Voltage Measured Across the Layer**

To visualize the voltage across the thin layer resistance, select Postprocessing > Cross-Section Plot Parameters > General. Click the Line/Extrusion plot radio button among the Plot type selection choices. Click the Line/Extrusion tab. Select "Electric Potential" from the y-axis data pull-down list. Click the Expression radio button. Click the Expression button and enter z in the edit window. See Figure 8.30.



**FIGURE 8.29** 3D\_TLR\_1 model solution, boundary plot

Click OK. For the cross-section line data, select or enter the given value as shown in Table 8.5. See Figure 8.31.

Click OK. See Figure 8.32.

#### 3D Thin Layer Resistance Model, Thin Layer Approximation: Summary and Conclusions

The 3D thin layer resistance model, thin layer approximation, has now been built and solved. This model employs the thin layer approximation to solve a model by replacing the center domain with a contact resistance identity pair. Such an approximation

0.0	X-Axis Data
x-axis data	
Predefined quantities:	z-coordinate
Expression:	k
Unit:	m 🛟

FIGURE 8.30 3D\_TLR\_1 model, X-Axis Data edit window

Value/Expression
0.5
0.5
0.0
0.5
0.5
0.2

 Table 8.5
 Cross-Section Line Data Parameters

has a broad applicability: It can be used in any problem in which flow is described by the divergence of a gradient flux (i.e., diffusion, heat conduction, flow through porous media under Darcy's law).

The application of the thin layer approximation is especially valuable to the modeler when the differences in domain thickness are so great that the mesh generator fails to properly mesh the model or creates more elements than the modeling platform can

Line plot y-axis data	C	) Extrusio	n plot		
Predefined quantities:	Electric p	•			
Expression:	V				
Unit:	V			•	
x-axis data	Cro	oss-section	n line data		
O Arc-length	\$ x0:	0.5	x1:	0.5	
• Expression	) y0:	0.5	y1:	0.5	
	z0:	0.0	z1:	0.2	
	Line	e resolutio	n:	200	

FIGURE 8.31 3D\_TLR\_1 model, Cross-Section Plot Parameters edit window

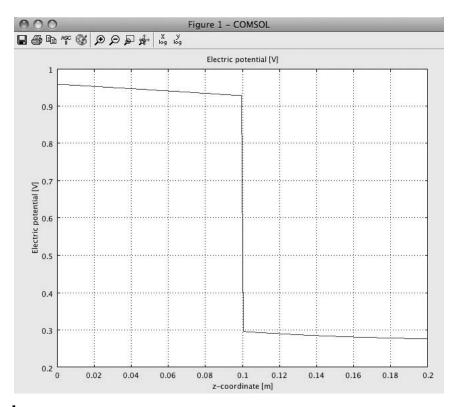


FIGURE 8.32 3D\_TLR\_1 model, cross-section electric potential plot

handle (the "run out of memory" problem). In those cases, this approximation may enable a model to be solved that would otherwise fail.

#### **3D Thin Layer Resistance Model: Thin Layer Subdomain**

The following numerical solution model (3D\_TLR\_2 model) is similar to the previous model. However, in this case, the center layer is a full domain.

To start building the 3D\_TLR\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Electromagnetics > Conductive Media DC. See Figure 8.33. Click OK.

#### **Geometry Modeling**

Using the menu bar, select Draw > Block. In the Block edit window, enter the information shown in Table 8.6. Click OK after filling in the parameters of each separate block in the Block edit window. See Figures 8.34, 8.35, and 8.36.

Name	Style	Base	Length (X, Y, Z)	Axis Base Point (X, Y, Z)	Figure Number
BLK1	Solid	Corner	(1, 1, 0.1)	(0, 0, 0)	8.34
BLK2	Solid	Corner	(1, 1, 0.02)	(0, 0, 0.1)	8.35
BLK3	Solid	Corner	(1, 1, 0.1)	(0, 0, 0.12)	8.36

## Table 8.6 Geometry Components

	Model Library User Models	Open Settings
Space dimension:	D 🛟	Electromagnetics
Application Modes	n in	Liectromagneuts
<ul> <li>COMSOL Multiphy:</li> <li>Acoustics</li> </ul>	sics	
Convection an	d Diffusion	
V 🚔 Electromagnet	C. Martine Mark	
Conductive	Media DC	3.0
Electrostat		
Fluid Dynamics		Description:
▶ 🛄 Heat Transfer		Electrostatics of conducting materials
<ul> <li>Structural Mec</li> <li>PDE Modes</li> </ul>	nanics	
Deformed Mes	h	
Electro-Therm	Contraction of the second seco	
Dependent variables:	v	
Application mode name:	dc	
Element:	Lagrange - Quadra	Multiphysics
5121001010 E		muniphysics

FIGURE 8.33 3D\_TLR\_2 Model Navigator setup

Style	Base	Axis base point	Re	otation an	ale
Solid Solid	Corner	x: 0 y: 0		0	(degrees)
Length		z: 0			
X: 1		Axis direction vector	<u>.</u>		
Y: 1		Cartesian coordinates	0	Spherica	l coordinates
Z: 0.1		x: 0	θ:	0	(degrees)
Name: BLK1		y: 0 z: 1	φ;	0	(degrees)

**FIGURE 8.34** 3D\_TLR\_2 model BLK1 edit window

Style	Base	Ax	is base point	F	lotation a	ngle
Solid	Corner	x:	0		x: 0	(degrees
O Face	O Center	y:	0			
Length		z:	0.1			
K: 1		Ax	is direction vector			
Y: 1		6	) Cartesian coordir	nates (	) Spheric	al coordinates
Z: 0.02		x:	0	θ:	0	(degrees)
		y:	0	φ:	0	(degrees)
ame: BLK2		z:	1			
Name: BLK2			1			

FIGURE 8.35 3D\_TLR\_2 model BLK2 edit window

Click the Zoom Extents button. See Figure 8.37.

Select File > Save As. Enter 3D\_TLR\_2 in the Save As edit window. See Figure 8.38. Click the Save button.

Using the menu bar, select Draw > Work-Plane Settings. See Figure 8.39.

NOTE As noted earlier, the use of the Work-Plane Settings is intended to make specific 2D planes available to the modeler to facilitate the creation of essentially 2D objects in the 3D geometry.

200		Block	_	_	
Style Solid Face	Base Corner Center	Axis base point x: 0 y: 0		ctation an	ngle (degrees
Length		z: 0.12			
X: 1		Axis direction vector			
Y: 1		Cartesian coordin	ates C	) Spherica	al coordinates
Z: 0.1		x: 0	θ:	0	(degrees)
Name: BLK3	0	y: 0 z: 1	φ:	0	(degrees)

FIGURE 8.36 3D\_TLR\_2 model BLK3 edit window

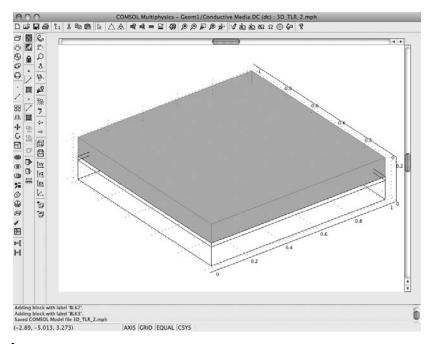


FIGURE 8.37 3D\_TLR\_2 model

Click OK, using the default settings. Click the Zoom Extents button. See Figure 8.40. Using the menu bar, select Draw > Specify Objects > Circle. Enter a radius of 0.6, set the base to Center, and set x equal to 0 and y equal to 1. See Figure 8.41. Click OK.

10_Models	•
Name	Date Modified
3D_TLR_1.mph	Friday, April 10, 2009 12.28 PM
E_P_Cylinders_1a.mph E_P_Ellipsoids_2a.mph	Wednesday, April 1, 2009 8:02 AM Wednesday, April 1, 2009 2:15 PM
E P Sph Ellipsoids 3a.mph	Wednesday, April 1, 2009 2:15 PM Wednesday, April 1, 2009 2:48 PM
H Coil 1a.mph	Thursday, April 2, 2009 8:37 AM
H_Coil_2a.mph	Thursday, April 2, 2009 7:49 PM
H_Coil_3a.mph	Friday, April 3, 2009 5:45 PM
Q_P_E_P_1a.mph	Wednesday, April 1, 2009 5:59 PM
Thin_Film_Resistance_1a.mph	Tuesday, March 31, 2009 3:36 PM
Thin Film Resistance 2a.mph	Tuesday, March 31, 2009 5:05 PM

FIGURE 8.38 3D\_TLR\_2 model Save As edit window

Plane	1		e	OK
• x-y z:	S		C	Cano
Ο γ-z ×: Ο z-x γ:			C	App
0 2-x y.	0		C	Hel
			C	Previe

**FIGURE 8.39** 3D\_TLR\_2 model Work-Plane Settings edit window

Size		Rotat	ion angle —	
Radius:	0.6	] α:	0	(degrees)
Positior				
Base:	Center 🛟	Style:	Solid	¢
<b>x</b> :	0	Name:	C1	
y:	1	1		

FIGURE 8.41 3D\_TLR\_2 model Circle edit window

Using the menu bar, select Draw > Specify Objects > Square. Enter a width of 1, set the base as Corner, and set x equal to 0 and y equal to 0. See Figure 8.42. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter C1\* SQ1 in the Set formula edit window. See Figure 8.43.

Size		Rotation angle
Width:	1	α: 0 (degrees)
Positio	n	
Base:	Corner	Style: Solid
<b>x</b> :	0	Name: SQ1
y:	0	

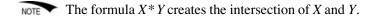
FIGURE 8.42 3D\_TLR\_2 model Square edit window

000 G	reate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
SQ1	Keep interior boundaries Repair Repair 1.0E-4

FIGURE 8.43 3D\_TLR\_2 model Create Composite Object edit window

	<b>۲</b>	X	Pa 63	$\square$		송 랴	= 🗎	(H) @	PPP	1	1 200 5	ΩΩ()	0.60	8				
2	_	_					_		Geom1	Ge	om2	_		_				_
7		-					,		_	-								-
•	1	23	12	it.	1	024		10	75	-				- 23		12	۰.	.
1	0.9	e et	2	it.	-						-	21	3		10	85.5	12	
ī	0.8										/ .				28.5			
16.1 16.1 16.1	0.7	• •	(ł	с÷	C01					/			1			240	(4)	
- DH 200	0.6	- 2	æ	9					./		30	30.3			2		10	
<u>e</u>	0.5	4	iii	+					<u> </u>	÷		÷	(a)	(a) (	50) (	- 24	191	
•	0.4		22	79	-			22	$\hat{\mathbf{x}}$	92	9	÷	(a) (		31	240	195	-
5	0.3	- 14	19	- 32		141		47	22	i.	-	аř			30	(10)		
	0.2	:02	12	14		949		21	20	4	ų.	4	64 I I		1941	14	14	-
	0.1	2555	52	12		020		27	26	21	$\overline{a}$	91	21	12	2	$\sim$	125	-
	0		1	ii.	_	197		- 22		2	-	- 3-	3	- 22	-	÷	-	
		-0.3	-0.2	-0.1	0	0.1	0.2	0.3	0.4	0.5	0.6	0.7	0.8	0.9	1	1.1	1.2	1.3
	le with la are with																	

**FIGURE 8.44** 3D\_TLR\_2 model CO1 (intersection of circle and square)



Click OK. See Figure 8.44.

Using the menu bar, select Draw > Embed. See Figure 8.45. Click OK.

As is obvious, the quarter-circle electrode needs to be moved to the upper surface of the upper block. Using the menu bar, select Draw > Modify > Move. Enter x = 0, y = 0, and z = 0.22. See Figure 8.46.

Objects to emb	oed:	
C01		Ţ
		l
		l
		1
Embed to geor	netry:	
Geom1		\$
Geom1 Embedded obj	ect name:	\$

**FIGURE 8.45** 3D\_TLR\_2 model Embed edit window

0.0	Move	
placement		ок
0		
0		Cancel
0.22		1
	0	placement 0 0

FIGURE 8.46 3D\_TLR\_2 model Move edit window

Click OK. See Figure 8.47.

Select EMB1 and BLK3 (click on EMB1 and then shift-click on BLK3). See Figure 8.48. Using the menu bar, select Draw > Coerce To > Solid.

## Physics Subdomain Settings: Conductive Media DC (dc)

Having established the geometry for the 3D\_TLR\_2 model of three blocks and an electrode, the next step is to define the fundamental Physics conditions. Using the menu bar, select Physics > Subdomain Settings. In the Subdomain edit windows, enter the information shown in Table 8.7. Click OK. See Figures 8.49 and 8.50.

Subdomain	Name	Expression	Description	Figure Number
1, 3	σ	1	Electrical conductivity	8.49
2	σ	1e-2	Electrical conductivity	8.50



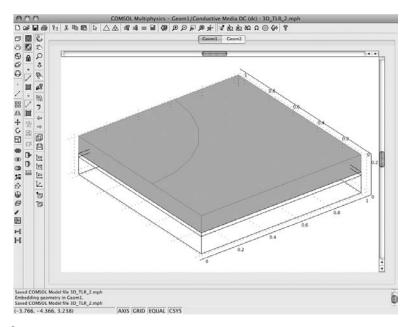


FIGURE 8.48 3D\_TLR\_2 model with EMB1 and BLK3 selected

T-1-1-00

### Physics Boundary Settings: Conductive Media DC (dc)

Using the menu bar, select Physics > Boundary Settings. For the indicated boundaries, select or enter the given boundary condition and value as shown in Table 8.8. Click OK. See Figures 8.51, 8.52, and 8.53.

Table 8.8	Boundary Settings – Conductive Media DC (dc) Edit Window

Boundary	Boundary Condition	Value/Expression	Figure Number
1, 2, 4, 5, 7, 8, 10, 12–17	Electric insulation	_	8.51
3	Inward current flow	0.3	8.52
11	Ground	_	8.53

Subdomains Groups			ment Color	
Ibdomain selection	Library material:	sources	)	
	Quantity J <sup>e</sup> Q <sub>j</sub> Conductivity relation: σ	Value/Expression 0 0 0 Conductivity 1	A/m <sup>2</sup>	Description External current density Current source Electric conductivity
oup: 🔹				

**FIGURE 8.49** 3D\_TLR\_2 model Subdomain Settings (1, 3) edit window

Subdomains Groups		Physics Init Elem	nent Color	
ubdomain selection	Material properties and Library material:	d sources		
	<b>Quantity</b> J <sup>e</sup> Q <sub>j</sub> Conductivity relation: σ	Value/Expression     0   0     0   0     Conductivity	A/m <sup>2</sup>	Description External current density Current source Electric conductivity
roup: 🔹 🗘				

**FIGURE 8.50** 3D\_TLR\_2 model Subdomain Settings (2) edit window

Boundaries Groups Boundary selection 10 11 12 13 14 15 16 17 Troup: Select hy group Interior boundaries	Conditions Color Boundary sources and constraints Library material: Boundary condition: Electric insulation

**FIGURE 8.51** 3D\_TLR\_2 model Boundary Settings (1, 2, 4, 5, 7, 8, 10, 12–17) edit window

NOTE In this model, the thin layer approximation has been replaced by subdomain 2.

### **Mesh Generation**

From the toolbar, select Mesh > Free Mesh Parameters > Subdomain. Select subdomain 2. Enter 0.02 in the Maximum element size edit window. See Figure 8.54. Click the Mesh Selected button.

Boundaries Groups		Conditions Color	]
Boundary selection	Boundary sources a Library material: Boundary condition Quantity J <sub>n</sub>	toad	Unit Description A/m <sup>2</sup> Normal current densit
Group:			

FIGURE 8.52 3D\_TLR\_2 model Boundary Settings (3) edit window

Boundary Settings - Conductive Media DC (dc)
Conditions Color Boundary sources and constraints Library materiat inad Boundary condition: Ground

FIGURE 8.53 3D\_TLR\_2 model Boundary Settings (11) edit window

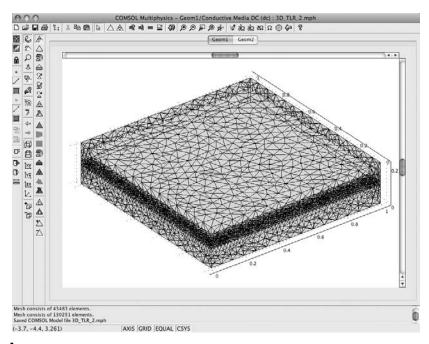
Click the Select Remaining button. (Subdomains 1 and 3 will be highlighted.) Click the Mesh Selected button. Click OK. See Figure 8.55.

## Solving the 3D\_TLR\_2 Model

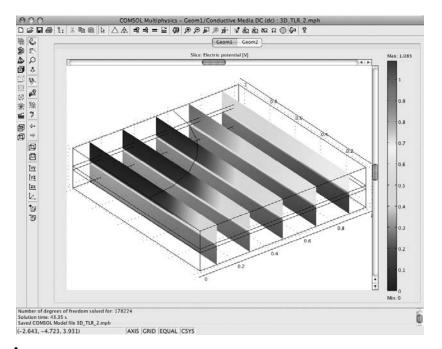
Using the menu bar, select Solve > Solve Problem. See Figure 8.56.

			FI	ee Mes	h Parai	neters			
Gl	bal	Subdu	main Boun	Jary	Edge	Puint	Advance	ed	
Subdomain 1 2 3	selecti	on I	Subdomai Maximur Element	n elemer	nt size:	0.02			Car (Ap
									He
									He
Select	by grou	η							
Select R									_ ( <u>H</u>

FIGURE 8.54 3D\_TLR\_2 model Free Mesh Parameters edit window







**FIGURE 8.56** 3D\_TLR\_2 model solution, default slice plot

10	Plot	Parameters		
General Slice	Isosurface	Subdomain	Boundary	Edge
Plot type Slice Isosurface Subdomain Roundary Edge Arrow Principal	Solution to us Solution at tim Time. Solution at an Frame: Geometries to Geom1	gle (phase): 0	2	grees
Principal     Streamline     Particle tracing     Max/min marker     Deformed shape     Geometry edges	Element Logical expre	selection ssion for inclusion s to fulfill express		
fot in: Main axes	🕄 🗌 Keep cu	nent: 🗹 Auto	3	
Smoothing)	Title	Make rough	Cancel	) ( ок

FIGURE 8.57 3D\_TLR\_2 model Plot Parameters edit window, General tab

The COMSOL Multiphysics software automatically selects the solver best suited for the particular model based on the overall evaluation of the model. The modeler can, of course, change the chosen solver and the parametric settings. It is usually best to try the selected solver and default settings first to determine how well they work. Then, once the model has been run, the modeler can do a variation on the model parameter space to seek improved results.

#### Postprocessing and Visualization

The default plot shows a slice plot of the electric potential (V) distribution in volts. To visualize the solution as a boundary plot, the Plot Parameters will need to be modified.

Select Postprocessing > Plot Parameters > General. In the Plot type list, unselect the Slice check box. In the Plot type list, select the Boundary check box. See Figure 8.57.

Subdomain	Boundary	
	/	Edge
\$	Range	
	Smooth	
\$		
ill style: Fi	illed 🛟	
lors: 1024	🗹 Color sca	ale

FIGURE 8.58 3D\_TLR\_2 model solution Plot Parameters edit window, Boundary tab

Click the Boundary tab. Select Conductive Media DC (dc) > Electric potential (V). Unselect the Smooth check box. See Figure 8.58.

Click OK. See Figure 8.59.

## TLR Voltage Measured Across the Layer

To visualize the voltage across the thin layer resistance (subdomain 2), select Postprocessing > Cross-Section Plot Parameters > General. Click the Line/Extrusion plot radio button among the Plot type selection choices.

Click the Line/Extrusion tab. Select "Electric Potential" from the y-axis data pulldown list. Click the Expression radio button. Click the Expression button and enter z in the edit window. See Figure 8.60. Click OK.

**FIGURE 8.59** 3D\_TLR\_2 model solution, boundary plot

For the cross-section line data, select or enter the given value as shown in Table 8.9. See Figure 8.61.

Click OK. See Figure 8.62.

### **3D Thin Layer Resistance Models: Summary and Conclusions**

The 3D thin layer resistance model, thin layer approximation, and the 3D thin layer resistance model, thin layer subdomain, have now been built and solved. A direct

000	X-Axis Data
x-axis data	
Predefined quantities:	z-coordinate
Expression:	z
Unit:	m 🛟
	Cancel OK

FIGURE 8.60 3D\_TLR\_2 model X-Axis Data edit window

Quantity	Value/Expression
×0	0.5
у0	0.5
zO	0.0
x1	0.5
y1	0.5
z1	0.2

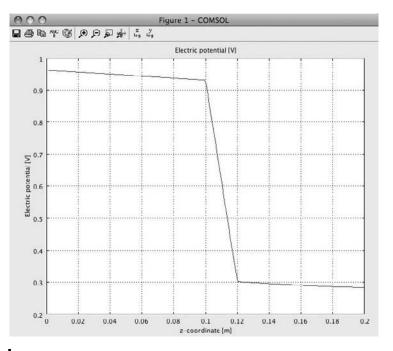
### Table 8.9 Cross-Section Line Data Parameters

comparison can be made of the model solutions by comparing the results obtained from the cross-section plots. See Figures 8.63 and 8.64.

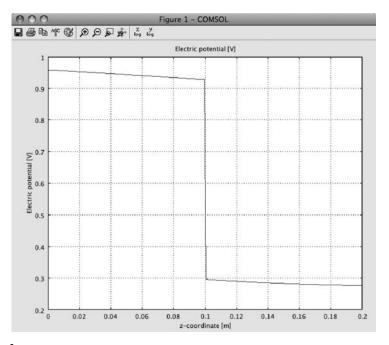
As can be seen from the examination of the plots, the only substantial difference between the two solutions is the electrical potential difference across subdomain 2. Thus the modeler can choose the implementation that best suits his or her system and time constraints, without suffering excessive inaccuracies based on the approximation method.

• Line plot y-axis data	Extrusion plot
Predefined quantities:	Electric potential
Expression: Unit:	v 🗘
x-axis data	Cross-section line data
O Arc-length	* x0: 0.5 x1: 0.5
Expression	
	z0: 0.0 z1: 0.2
	Line resolution: 200

FIGURE 8.61 3D\_TLR\_2 model Cross-Section Plot Parameters edit window



**FIGURE 8.62** 3D\_TLR\_2 model, cross-section electric potential plot



**FIGURE 8.63** 3D\_TLR\_1 model, cross-section electric potential plot, thin layer approximation

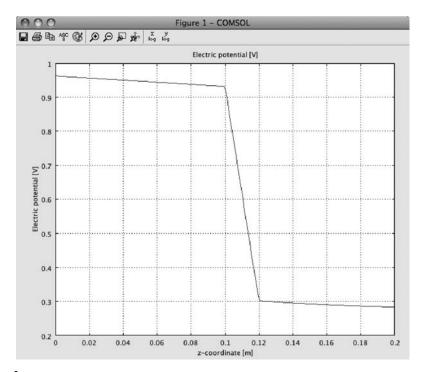


FIGURE 8.64 3D\_TLR\_2 model, cross-section electric potential plot, thin layer subdomain

## **Electrostatic Modeling Basics**

The study of static electricity, a well-known and widely observed phenomenon, has a long history. Thales of Miletus<sup>7</sup> recorded the first known scientific observations using amber in approximately the sixth century BC. Additional serious documented scientific work on static electricity did not occur until Otto von Guericke<sup>8</sup> invented the first electrostatic generator<sup>9</sup> around 1663.

The physics of static electricity was not well understood until the work of Charles-Augustin de Coulomb,<sup>10</sup> Johann Carl Friedrich Gauss,<sup>11</sup> and others<sup>12</sup> explored electrostatics and mathematics of physics in the late 1700s to early 1800s AD. Based on that work, the electrostatic scalar potential (V) is known to be related to the electric field vector ( $\mathbf{E}$ ) as follows:

$$\mathbf{E} = -\nabla V \tag{8.13}$$

where  $\mathbf{E} = \text{electric field vector (V/m)}$ 

 $\nabla$  = divergence operator (1/m)

V = scalar electric potential (V)

NOTE The divergence operator is as follows:

$$\nabla = \hat{i}\frac{\partial}{\partial x} + \hat{j}\frac{\partial}{\partial y} + \hat{k}\frac{\partial}{\partial z}$$

where i, j, k are the unit vectors in the x, y, z directions, respectively.

Using Gauss's law,13

$$\nabla \cdot (\varepsilon \mathbf{E}) = \rho \tag{8.14}$$

where  $\mathbf{E} = \text{electric field vector (V/m)}$   $\nabla = \text{divergence operator (1/m)}$   $\varepsilon = \text{permittivity}$  $\rho = \text{space charge density}$ 

$$-\nabla \cdot (\varepsilon \nabla V) = \rho \tag{8.15}$$

and

$$-\nabla \cdot (\varepsilon \nabla V) = -\varepsilon \nabla^2 V = \rho \tag{8.16}$$

where  $\nabla^2$  is the Laplacian operator.

Note The Laplacian operator  $^{14,15}$  is as follows:

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2}$$

It is a differential operator, shown above in the scalar form, named after Pierre-Simon de Laplace.<sup>16</sup> The Laplacian operator is widely employed in the physics of electromagnetics, wave propagation, heat flow, fluid flow, and quantum mechanics, to name a few areas.

A large body of subsequent work has led to both scientific and engineering applications (e.g., ranging from X-ray tubes and particle accelerators to paint sprayers and dust precipitators).

The 3D electrostatic potential models presented in this section are examples of methods that can be used by the modeler to explore electrostatic potentials in different geometric configurations.

New	Model Library	User Models	Open	Settings
Space dimension:	3D	\$	Flacter	omagnetics
	nd Diffusion tics re Media DC tics :s chanics		Description	P
Dependent variables: Application mode name. Element:	V es Lagrange - Ou			
riement	Lagrange - Qu	aura 🔹	<u> </u>	Multiphysics

FIGURE 8.65 3D\_ESP\_1 Model Navigator setup

## **3D Electrostatic Potential Between Two Cylinders**

The following numerical solution model (3D\_ESP\_1 model) is derived from a model that was originally developed by COMSOL as a Multiphysics Electromagnetics demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL Multiphysics Model Library.

To start building the 3D\_ESP\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Electromagnetics > Electrostatics. See Figure 8.65. Click OK.

### **Geometry Modeling**

Using the menu bar, select Draw > Sphere. In the Sphere edit window, enter the information shown in Table 8.10. Click OK after filling in the parameters of each separate solid in the appropriate edit window. See Figures 8.66, 8.67, and 8.68.

Solid	Name	Style	Radius	Height	Axis Base Point (X, Y, Z)	Axis Direction Vector (X, Y, Z)	Figure Number
Sphere	SPH1	Solid	2		(0, 0, 0)	(0, 0, 1)	8.66
Cylinder	CYL1	Solid	0.1	0.4	(-0.4, 0, -0.2)	(0, 0, 1)	8.67
Cylinder	CYL2	Solid	0.1	0.4	(0.4, 0, -0.2)	(0, 0, 1)	8.68

#### Table 8.10 Geometry Components

**FIGURE 8.66** 3D\_ESP\_1 model Sphere SPH1 edit window

**FIGURE 8.67** 3D\_ESP\_1 model Cylinder CYL1 edit window

**FIGURE 8.68** 3D\_ESP\_1 model Cylinder CYL2 edit window

Save As: 3D_ESP_1.mp	h
Name       3D_TLR_1.mph       3D_TLR_2.mph	Date Modified Friday, April 10, 2009 12:28 PM Friday, April 10, 2009 5:28 PM
File Format: COMSOL	Multiphysics Mo + Cancel Save

FIGURE 8.69 3D\_ESP\_1 model Save As edit window

Click the Zoom Extents button. Select File > Save As. Enter 3D\_ESP\_1.mph in the Save As edit window. See Figure 8.69. Click the Save button.

Using the menu bar, select Draw > Create Composite Object. Enter SPH1-CYL1-CYL2 in the Set formula edit window. See Figure 8.70. Click OK.

000 c	reate Composite Object
Object type Solids Faces Curves Points	Shortcuts Union Intersection Select All Help
Object selection: SPH1 CYL1 CYL2	Set formula: SPH1-CYL1-CYL2 Keep interior boundaries Keep interior edges Repair Repair tolerance: 1.0E-4

FIGURE 8.70 3D\_ESP\_1 model Create Composite Object edit window

FIGURE 8.71 3D\_ESP\_1 model Subdomain Settings edit window

### Physics Subdomain Settings: Electrostatics (es)

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 (the only subdomain available). Click the radio button  $D = \varepsilon_0 \varepsilon_r E$ . Enter 0 in the Space charge density ( $\rho$ ) edit window. Enter 1 in the Relative permittivity ( $\varepsilon_r$ ) edit window. See Figure 8.71. Click OK.

The numerical value of the permittivity for free space ( $\varepsilon_0$ ) in SI units is 8.854\*10<sup>12</sup> F/m. That value is the default value for permittivity incorporated into the COMSOL Multiphysics software. The permittivity of a material is the product of  $\varepsilon_0$  and  $\varepsilon_r$ , which in this case is 1.

#### **Physics Boundary Settings: Electrostatics (es)**

Using the menu bar, select Physics > Boundary Settings. Select or enter the settings as indicated in Table 8.11. Click OK. See Figures 8.72, 8.73, and 8.74.

Boundary	Settings	Value	Figure Number
1-4, 11-14	Zero charge/symmetry		8.72
5–10	Electric potential	1V	8.73
15–20	Electric potential	-1V	8.74

#### Table 8.11 Boundary Settings

**FIGURE 8.72** 3D\_ESP\_1 model Boundary Settings (1–4, 11–14) edit window

## **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Select "Coarser" from the Predefined mesh sizes pull-down list. See Figure 8.75. Click OK.

Using the menu bar, select Mesh > Initialize Mesh. See Figure 8.76.

quation = V <sub>0</sub>				
Boundaries Groups Boundary selection	Boundary sources and Boundary condition: Quantity Y <sub>0</sub>	Conditions Cold d constraints Electric potential + Value/Expression -1	Unit	Description Electric potential

FIGURE 8.74 3D\_ESP\_1 model Boundary Settings (15–20) edit window

## Solving the 3D\_ESP\_1 Model

**NOTE** Electrostatics problems can be complex and difficult. The modeler can, of course, accept the COMSOL software default settings. However, in this case, depending on the modeler's platform, it will probably be best to choose an iterative solver (GMRES) and an appropriate preconditioner (Algebraic multigrid). These choices will reduce both the memory required and the time to solution.

6	Global	C.1.1						<u> </u>
	Global	Subdomain	Bou	ndary	Edge	Point	Advanced	ОК
🕑 Pred	lefined m	esh sizes:	C	oarser		•		Cane
() Cus	itom mes	h size						Appl
Maximui	n elemen	t size:						(Help
Maximui	n elemen	t size scaling fa	ctor:	1.9				CHeit
Element	arowth r	ate:		1.7				
Mesh cu	rvature fa	actor:		0.8				
Mesh cu	rvature c	utoff:		0.05				
Resoluti	on of nar	row regions:		0.3				
🖌 Optin	nize qual	ity						
afineme	nt metho	d: Longest						

**FIGURE 8.75** 3D\_ESP\_1 model Free Mesh Parameters edit window

**FIGURE 8.76** 3D\_ESP\_1 model mesh

Select Solve > Solver Parameters. Select "GMRES" from the Linear system solver pull-down list. Select "Algebraic multigrid" from the Preconditioner pull-down list. See Figure 8.77.

Click OK. Select Solve > Solve Problem.

#### Postprocessing and Visualization

The default solution plot is the slice plot. See Figure 8.78.

Using the menu bar, select Postprocessing > Plot Parameters > General. Uncheck the Slice check box, and check the Boundary and Streamline check boxes. See Figure 8.79.

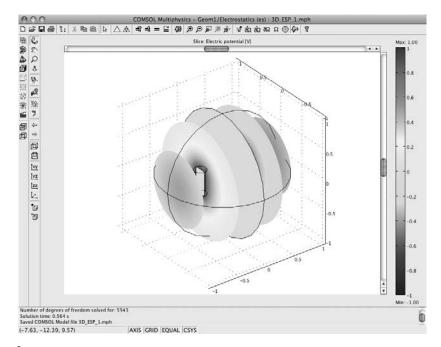
Click the Boundary tab. Click the Apply button.

Click the Streamline tab. Enter 30 in the Number of start points window. Click OK.

To see the streamline plot, the modeler will need to suppress the boundaries of the sphere. Using the menu bar, select Options > Suppress > Suppress Boundaries. Select boundaries 1–4 and 11–14. See Figure 8.80. Click OK.

000	Solver Parameters
Analysis:  Auto select solver  Solver:  Stationary  Time dependent Eigenvalue	General     Stationary     Adaptive     Advanced       Linear system solver     Elinear system solver     Image: System solver       Preconditioner:     Algebraic multigrid     Image: System solver       Quality of multigrid hierarchy:     3
Parametric Stationary segregated Parametric segregated	Memory efficiency Precond. quality Settings Matrix symmetry: Automatic
	Help Apply Cancel OK

FIGURE 8.77 3D\_ESP\_1 model Solver Parameters edit window



**FIGURE 8.78** 3D\_ESP\_1 model default solution plot

Solution to use   Slice   Slice   Isosurface   Subdomain   Subdomain   Solution at angle (phase):   0   degrees   Frame:     Principal   Streamline
Isosurface       Time:         Subdomain       Solution at angle (phase):         Ø Boundary       Solution at angle (phase):         Edge       Frame:         Arrow       Geometries to use         Principal       Geom1         Streamline       Streamline
✓ Streamline
Particle tracing  Max/min marker  Deformed shape  Geometry edges Element nodes to fulfill expression:  All

**FIGURE 8.79** 3D\_ESP\_1 model Plot Parameters selection window

FIGURE 8.81 3D\_ESP\_1 model streamline plot with suppressed boundaries

Using the menu bar, select Postprocessing > Plot Parameters > Streamline. Click the Apply button, and then click OK. See Figure 8.81.

## 3D Electrostatic Potential Between Two Cylinders: Summary and Conclusions

The 3D electrostatic potential model presented here demonstrates one of the methods that can be used by the modeler to explore electrostatic potentials in different geometric configurations. This technique can be applied to both scientific and engineering applications (e.g., ranging from X-ray tubes and particle accelerators to paint sprayers and dust precipitators).

## **3D Electrostatic Potential Between Five Cylinders**

To start building the 3D\_ESP\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select COMSOL Multiphysics > Electromagnetics > Electrostatics. See Figure 8.82. Click OK.

New	Model Library User Mode	els Open Settings
Application Modes COMSOL Multiphy COMSOL Multiphy Convection ar Convecti	nd Diffusion tics e Media DC tics s :hanics	Electromagnetics Description: Electrostatics of dielectric materials.
Dependent variables: Application mode name:	V es	
Element:	Lagrange - Quadra 🗘	Multiphysics

**FIGURE 8.82** 3D\_ESP\_2 Model Navigator setup

## **Geometry Modeling**

Using the menu bar, select Draw > Sphere. In the Sphere edit window, enter the information shown in Table 8.12. Click OK after filling in the parameters of each separate solid in the appropriate edit window. See Figures 8.83–8.88.

## Table 8.12 Geometry Components

Solid	Name	Style	Radius	Height	Axis Base Point (X, Y, Z)	Axis Direction Vector (X, Y, Z)	Figure Number
Sphere	SPH1	Solid	4		(0, 0, 0)	(0, 0, 1)	8.83
Cylinder	CYL1	Solid	0.1	0.4	(-0.4, 0, -0.2)	(0, 0, 1)	8.84
Cylinder	CYL2	Solid	0.1	0.4	(0, 0, -0.2)	(0, 0, 1)	8.85
Cylinder	CYL3	Solid	0.1	0.4	(0.4, 0, -0.2)	(0, 0, 1)	8.86
Cylinder	CYL4	Solid	0.1	0.4	(0, -0.4, -0.2)	(0, 0, 1)	8.87
Cylinder	CYL5	Solid	0.1	0.4	(0, 0.4, -0.2)	(0, 0, 1)	8.88

000	Sphere		
Style Solid	Axis base point x. 0	Rotatio	n angle (degrees)
() Face	у: 0		
Sphere parameters Radius. 4	2: 0		
Name: SPH1	Cartesian coordina	ites 🔘 Sphe	erical coordinates
	x: 0	θ: 0	(degrees)
	у: 0	φ: 0	(degrees)
	z: 1		
	Help App	ly Can	ocel ОК

**FIGURE 8.83** 3D\_ESP\_2 model Sphere SPH1 edit window

**FIGURE 8.84** 3D\_ESP\_2 model Cylinder CYL1 edit window

**FIGURE 8.85** 3D\_ESP\_2 model Cylinder CYL2 edit window

# FIGURE 8.86 3D\_ESP\_2 model Cylinder CYL3 edit window

Style	Axis base point	Rotat	ion angle
🕑 Sulid	x. 0	α. [	) (degrees
⊖ Face	y: -0.4		
Cylinder parameters	z: -0.2		
Radius. 0.1	Axis direction vector		
Height: 0.4	Cartesian coordinate	es 🔘 Sp	herical coordinates
lame: CYL4	x: 0	θ: 0	(degrees)
tame. CTC4	y: 0	φ: 0	(degrees)
	z: 1		

**FIGURE 8.87** 3D\_ESP\_2 model Cylinder CYL4 edit window

000	Cylinder	
Style Sulid Face	Axis base point         Rotation           x:         0         a:         0           y:         0.4	angle (degrees
Cylinder parameters	z:0.2	
Radius. 0.1 Height: 0.4	Axis direction vector Cartesian coordinates O Spheri	cal coordinates
Name: CYL5	x:         0         θ:         0           y:         0         φ:         0	(degrees)
	z. 1	
	Help Apply Cance	е Ок

**FIGURE 8.88** 3D\_ESP\_2 model Cylinder CYL5 edit window

000	Save As
Save As: 3D	ESP_2.mph
10_	Models 🗘
Name	Date Modified
3D_ESP_1.mph 3D_ESP_2.mph 3D_TLR_1.mph 3D_TLR_2.mph 3D_TLR_2.mph	Monday, April 13, 2009 5:38 PM Tuesday, April 14, 2009 10:14 AM Friday, April 10, 2009 12:28 PM Friday, April 10, 2009 5:28 PM
File Format:	
(New Folder	(Cancel) (Save

FIGURE 8.89 3D\_ESP\_2 model Save As edit window

Click the Zoom Extents button. Select File > Save As. Enter 3D\_ESP\_2.mph in the Save As edit window. See Figure 8.89. Click the Save button.

Using the menu bar, select Draw > Create Composite Object. Enter SPH1-CYL1-CYL2-CYL3-CYL4-CYL5 in the Set formula edit window. See Figure 8.90. Click OK.

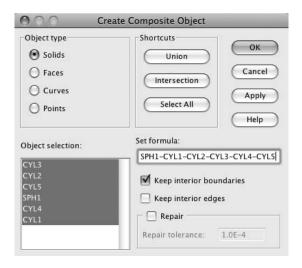


FIGURE 8.90 3D\_ESP\_2 model Create Composite Object edit window

Subdomains Groups		Physics Init	Element Color
ubdomain selection	-Material propert	ies and sources	
1	Library material		Load
	Constitutive re		
	$\bigcirc \mathbf{D} = \mathbf{\varepsilon}_{0}\mathbf{\varepsilon}_{r}\mathbf{E}$	$\bigcirc \mathbf{D} = \mathbf{\epsilon}_0 \mathbf{E} + \mathbf{P}$	$\bigcirc$ D = $\varepsilon_0 \varepsilon_r E + D_r$
	Quantity	Value/Expression	Unit Description
	ρ	0	C/m <sup>3</sup> Space charge densi
Group:	F,	1	1 Relative permittivity
Select by group		1.	Relative permitting
Active in this domain			

FIGURE 8.91 3D\_ESP\_2 model Subdomain Settings edit window

### Physics Subdomain Settings: Electrostatics (es)

Using the menu bar, select Physics > Subdomain Settings. Select subdomain 1 (the only subdomain available). Click the radio button  $\mathbf{D} = \varepsilon_0 \varepsilon_r \mathbf{E}$ . Enter 0 in the Space charge density ( $\rho$ ) edit window. Enter 1 in the Relative permittivity ( $\varepsilon_r$ ) edit window. See Figure 8.91. Click OK.

**NOTE** The numerical value of the permittivity for free space ( $\varepsilon_0$ ) in SI units is 8.854\*10<sup>12</sup> F/m. That value is the default value for permittivity incorporated into the COMSOL Multiphysics software. The permittivity of a material is the product of  $\varepsilon_0$  and  $\varepsilon_r$ , which in this case is 1.

#### **Physics Boundary Settings: Electrostatics (es)**

Using the menu bar, select Physics > Boundary Settings. Select or enter the settings as indicated in Table 8.13. Click OK. See Figures 8.92, 8.93, and 8.94.

Table 8.13	<b>Boundary Settings</b>
------------	--------------------------

Boundary	Settings	Value	Figure Number
1–4, 23,	Zero charge/symmetry	_	8.92
24, 28, 29			
5–14, 19–22,	Electric potential	1V	8.93
25, 26, 31–38			
15–18,	Electric potential	-1V	8.94
27, 30			

FIGURE 8.92 3D\_ESP\_2 model Boundary Settings (1–4, 23, 24, 28, 29) edit window

## **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Select "Coarser" from the Predefined mesh sizes pull-down list. See Figure 8.95. Click OK.

Using the menu bar, select Mesh > Initialize Mesh. See Figure 8.96.

• V <sub>0</sub>			_	
Boundaries Groups	Boundary sources an Boundary condition: Quantity V <sub>0</sub>	Conditions Color d constraints Electric potential Value/Expression -1	Unit V	<b>Description</b> Electric potential
Select by group				

**FIGURE 8.94** 3D\_ESP\_2 model Boundary Settings (15–18, 27, 30) edit window

Global Subdomain Bou	ndary Edge Point	Advanced	
Predefined mesh sizes:     C	oarser 🗘		Can
O Custom mesh size		10	Ар
Maximum element size:			He
Maximum element size scaling factor:	1.9		Crie
Element growth rate:	1.7		
Mesh curvature factor:	0.8		
Mesh curvature cutoff:	0.05		
Resolution of narrow regions:	0.3		
🗹 Optimize quality		1/1	
efinement method: Longest 🛟			

**FIGURE 8.95** 3D\_ESP\_2 model Free Mesh Parameters edit window

**FIGURE 8.96** 3D\_ESP\_2 model mesh

#### Solving the 3D\_ESP\_2 Model

**NOTE** Electrostatics problems can be complex and difficult. The modeler can, of course, accept the COMSOL software default settings. However, in this case, depending on the modeler's platform, it will probably be best to choose an iterative solver (GMRES) and an appropriate preconditioner (Algebraic multigrid). These choices will reduce both the memory required and the time to solution.

Select Solve > Solver Parameters. Select "GMRES" from the Linear system solver pull-down list. Select "Algebraic multigrid" from the Preconditioner pull-down list. See Figure 8.97.

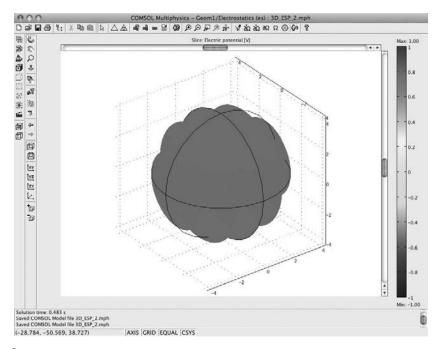
Click OK. Select Solve > Solve Problem.

#### **Postprocessing and Visualization**

The default solution plot is the slice plot. See Figure 8.98.

000	Solver Parameters
Analysis:  Auto select solver  Solver:  Stationary	General Stationary Adaptive Advanced
Time dependent Eigenvalue Parametric Stationary segregated Parametric segregated	Quality of multigrid hierarchy: 3 Memory efficiency Precond. quality Settings
Adaptive mesh refinement	Matrix symmetry: Automatic
	Help Apply Cancel OK

**FIGURE 8.97** 3D\_ESP\_2 model Solver Parameters edit window



**FIGURE 8.98** 3D\_ESP\_2 model default solution plot

Solution to use         Slice         Isosurface         Subdomain         Solution at angle (phase):         Boundary
Edge     Frame:       Arrow     Geometries to use       Principal     Geom1
Y Streamline         Particle tracing         Max/min marker         Deformed shape         Geometry edges    Element nodes to fulfill expression:          All

**FIGURE 8.99** 3D\_ESP\_2 model Plot Parameters selection window

Using the menu bar, select Postprocessing > Plot Parameters > General. Uncheck the Slice check box, and check the Boundary and Streamline check boxes. See Figure 8.99.

Click the Boundary tab. Click the Apply button.

Click the Streamline tab. Enter 29 in the Number of start points window. Click OK.

To see the streamline plot, the modeler will need to suppress the boundaries of the sphere. Using the menu bar, select Options > Suppress > Suppress Boundaries. Select boundaries 1–4, 23, 24, 28, and 29. See Figure 8.100. Click OK.

Using the menu bar, select Postprocessing > Plot Parameters > Streamline. Click the Apply button, and then click OK. See Figure 8.101.

Suppress Boundaries				
Boundary selection:				
ssion				
n				

FIGURE 8.100 3D\_ESP\_2 model Suppress Boundaries selection window

# **3D Electrostatic Potential Between Five Cylinders: Summary and Conclusions**

The 3D electrostatic potential models presented here demonstrate one of the methods that can be used by the new modeler to explore electrostatic potentials in different geometric configurations. The 3D\_ESP\_2 model is typical of those that might be found in a particle beam analyzer or similar engineering or scientific device. This modeling technique can be applied widely to both scientific and engineering applications (e.g., ranging from X-ray tubes and particle accelerators to paint sprayers and dust precipitators).

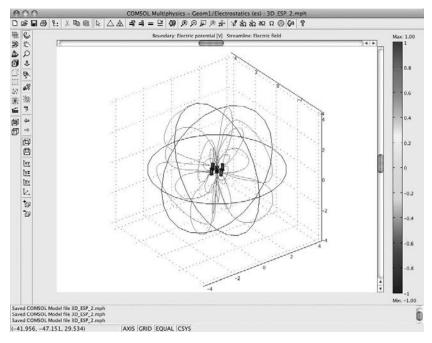


FIGURE 8.101 3D\_ESP\_2 model streamline plot with suppressed boundaries

#### Magnetostatic Modeling Basics

The fundamental equations governing electromagnetic phenomena are Maxwell's equations,<sup>17</sup> first published in 1873 by James Clerk Maxwell.<sup>18</sup> Maxwell's equations, as written for free charge and as commonly seen in scientific papers and textbooks, in SI units are

$$\nabla \cdot \mathbf{D} = \rho \tag{8.17}$$

$$\nabla \cdot \mathbf{B} = 0 \tag{8.18}$$

$$\nabla \times \mathbf{E} = -\frac{\partial B}{\partial t} \tag{8.19}$$

$$\nabla \times \mathbf{H} = \mathbf{J} + \frac{\partial D}{\partial t}$$
(8.20)

where 
$$\mathbf{E} =$$
 electric field vector in volts per meter (V/m)

- $\mathbf{D}$  = electric flux density in coulombs per square meter (C/m<sup>2</sup>)
- $\mathbf{B}$  = magnetic field vector in tesla (T)

 $\mathbf{H}$  = magnetizing field vector in amperes per meter (A/m)

 $\mathbf{J}$  = free current density in amperes per square meter (A/m<sup>2</sup>)

 $\rho$  = free charge density in coulombs per cubic meter (C/m<sup>3</sup>)

To solve any of the potential electromagnetic problems, the modeler also needs to assume continuity (no sinks or sources—that is, "What goes in, comes out"). The equation of continuity is

$$\nabla \cdot \mathbf{J} = -\frac{\partial \rho}{\partial t} \tag{8.21}$$

The modeler also needs to define the properties of the medium(s) throughout which the electromagnetic wave is traveling. These equations are called the constitutive relationships for the medium:

$$\mathbf{D} = \boldsymbol{\varepsilon}_0 \mathbf{E} + \mathbf{P} \tag{8.22}$$

$$\mathbf{B} = \boldsymbol{\mu}_0 (\mathbf{H} + \mathbf{M}) \tag{8.23}$$

$$\mathbf{J} = \boldsymbol{\sigma} \mathbf{E} \tag{8.24}$$

where

 $\mathbf{E}$  = electric field vector in volts per meter (V/m)

 $\mathbf{D}$  = electric flux density vector in coulombs per square meter (C/m<sup>2</sup>)

 $\mathbf{P}$  = electric polarization vector in coulombs per square meter (C/m<sup>2</sup>)

- $\mathbf{B}$  = magnetic field vector in tesla (T)
- $\mathbf{H}$  = magnetizing field vector in amperes per meter (A/m)
- $\mathbf{M}$  = magnetization vector in amperes per meter (A/m)
- $\mathbf{J}$  = free current density in amperes per square meter (A/m<sup>2</sup>)
- $\varepsilon_0$  = permittivity of vacuum in farads per meter (F/m)
- $\mu_0$  = permeability of vacuum in henries per meter (H/m)
- $\sigma$  = electric conductivity in siemens per meter (S/m)

In a magnetostatic model, such as the Helmholtz coil, all parameters are stable and do not fluctuate. If they do fluctuate, it is at a slow rate.

**NOTE** If the parameters of the model do fluctuate, a good measure of the validity of the model is that the dimensions of the model should be at least 10 times smaller than the wavelength of the fluctuation. Consider, for example, 60 Hz. The wavelength is calculated as follows:

$$\lambda = \frac{c}{f} = \frac{3 \times 10^8}{60} = 5 \times 10^6 \,\mathrm{m} \tag{8.25}$$

where  $\lambda$  = wavelength in meters (m) c = speed of light in meters per second (m/s) f = frequency in cycles per second (cycle/s)

In the case of a magnetostatic model, the relationships between the potentials and the fields are as follows:

$$\nabla \times (\mu^{-1} \nabla \times \mathbf{A}) = \mathbf{J}^{\mathbf{e}}$$
(8.26)

$$\mathbf{B} = \nabla \times \mathbf{A} \tag{8.27}$$

$$\mathbf{H} = \boldsymbol{\mu}^{-1} \mathbf{B} \tag{8.28}$$

$$\boldsymbol{\mu} = \boldsymbol{\mu}_0 \, \boldsymbol{\mu}_{\mathrm{r}} \tag{8.29}$$

where  $\mathbf{A} =$  magnetic vector potential in volt-seconds per meter (V · s/m)

- $\mathbf{B}$  = magnetic field vector in tesla (T)
- $\mathbf{H}$  = magnetizing field vector in amperes per meter (A/m)
- $J^e$  = externally applied current density in amperes per square meter (A/m<sup>2</sup>)
- $\mu_0$  = permeability of vacuum in henries per meter (H/m)

$$\mu_{\rm r}$$
 = relative permeability

Note The numerical value of the permeability for free space ( $\mu_0$ ) in SI units is exactly  $4\pi \times 10^{-7}$  H/m. That value is the default value for permeability incorporated into the COMSOL Multiphysics software. The permeability of a material is the product of  $\mu_0$  and  $\mu_r$ , which in this case is 1.

Because the electromagnetic potentials do not uniquely define a solution, within a gauge transformation<sup>19</sup> it is necessary to choose a gauge (transformation).<sup>20</sup> This technique is called gauge fixing. In this case, the gauge chosen is called the Coulomb gauge. The condition of the Coulomb gauge is that  $\nabla \cdot \mathbf{A} = 0$ .

To avoid numerical instability in the model,  $\nabla \cdot \mathbf{A}$  is numerically adjusted to zero by using a type of special pre- and post-smoother called an SOR gauge.<sup>21</sup>

### **3D Magnetic Field of a Helmholtz Coil**

The following numerical solution model (3D\_HC\_1 model) is derived from a model that was originally developed by COMSOL as an AC/DC Module Electrical Components demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL AC/DC Module Model Library.

To start building the 3D\_HC\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Magnetostatics. See Figure 8.102. Click OK.



FIGURE 8.102 3D\_HC\_1 Model Navigator setup

Name	Expression	Description	Figure Number
JO	1[A/m^2]	Coil current density	8.103

#### Table 8.14 Constants Edit Window

## Constants

Using the menu bar, select Options > Constants. Enter the constant shown in Table 8.14. Click OK. See Figure 8.103.

Select File > Save As. Enter 3D\_HC\_1.mph in the Save As edit window. See Figure 8.104. Click the Save button.

Name	Expression	Value	Description
JO	1[A/m^2]	1[A/m <sup>2</sup> ]	Coil current density

**FIGURE 8.103** 3D\_HC\_1 model Constants edit window

OO Save As: 3	Save As D_HC_1.mph
Name	)_Models
3D_ESP_1.mph 3D_ESP_2.mph 3D_TLR_1.mph 3D_TLR_2.mph	Monday, April 13, 2009 5:38 PM Tuesday, April 14, 2009 5:35 PM Friday, April 10, 2009 12:28 PM Friday, April 10, 2009 5:28 PM
File Format	t: COMSOL Multiphysics Mo 🛟
New Folder	Cancel Save

FIGURE 8.104 3D\_HC\_1 model Save As edit window

C					
Quick	Face Parallel	Edge Angle	Vertices	Advanced	
lane					ОК
🛈 х-у z:	0				Canc
() y-z x:	0				Appl
() z-x y:	0				Help
					Previe

**FIGURE 8.105** 3D\_HC\_1 model Work-Plane Settings edit window

#### **Geometry Modeling**

0.175

Help

y:

Each Helmholtz coil is created in cross section by drawing squares in the 2D work-plane. The modeler then creates a solid coil by revolution (revolve) of the 2D work-plane geometry into the 3D geometry.

Using the menu bar, select Draw > Work-Plane Settings. See Figure 8.105. Click OK.

Using the menu bar, select Draw > Specify Objects > Square. In the Square edit window, enter the information shown in Table 8.15. Click OK after filling in the parameters of each separate square in the appropriate edit window. See Figures 8.106 and 8.107.

Name	Width	Base	х	у	Figure Numbe
SQ1	0.05	Corner	-0.425	0.175	8.106
SQ2	0.05	Corner	-0.425	-0.225	8.107
	540	lare			
-	Squ	-Rotation angle-			
Size Width: 0.05	340	201	(degrees)		
100 m		-Rotation angle-	(degrees)		

ок

Table 8.15	Geometry	Components
------------	----------	------------

FIGURE 8.106 3D\_HC\_1 model Square SQ1 edit window

Apply

Cancel

Size		Rotat	ion angle	
Width:	0.05	α:	0	(degrees)
Positio	n			
Base:	Corner	\$ Style:	Solid	\$
x:	-0.425	Name:	SQ2	
y:	-0.225			

FIGURE 8.107 3D\_HC\_1 model Square SQ2 edit window

Click the Zoom Extents button. Select SQ1 and SQ2. Using the menu bar, select Draw > Revolve. See Figure 8.108.

Click OK. See Figure 8.109.

Using the menu bar, select Draw > Sphere. Enter 1 in the Radius edit window. See Figure 8.110.

Click OK, and then click the Zoom Extents button. Click on the display window background outside the sphere. See Figure 8.111.

**FIGURE 8.109** 3D\_HC\_1 model Helmholtz coil pair

000	Sphere		
Style Solid	Axis base point x: 0	Rotation angle α: 0 (deg	rees
⊖ Face	y: 0		
Sphere parameters	z: 0		
Radius: 1	Axis direction vector		
Name: SPH1	Cartesian coordinat	es 🔘 Spherical coordinate	es
	x: 0	θ: 0 (degre	es)
	y: 0	φ: 0 (degre	es)
	z: 1		

**FIGURE 8.110** 3D\_HC\_1 model Sphere edit window

FIGURE 8.111 3D\_HC\_1 model sphere and Helmholtz coil pair

### Physics Subdomain Settings: Magnetostatics (emqa)

Using the menu bar, select Physics > Subdomain Settings. Select the subdomain(s) and enter the expression indicated in Table 8.16. Click OK. See Figures 8.112 and 8.113.

The numerical value of the permeability for free space  $(\mu_0)$  in SI units is exactly  $4\pi \times 10^{-7}$  H/m. That value is the default value for permeability incorporated into the COMSOL Multiphysics software. The permeability of a material is the product of  $\mu_0$  and  $\mu_r$ , which in this case is 1.

Subdomain	Quantity	Value/Expression	Figure Number
1	J <sup>e</sup>	000	8.112
2, 3	J <sup>e</sup>	$-J0*z/sqrt(x^2+z^2) 0$	8.113
		$J0^*x/sqrt(x^2+z^2)$	

#### Table 8.16 Subdomain Settings

FIGURE 8.112 3D\_HC\_1 model Subdomain Settings (1) edit window

## Physics Boundary Settings: Magnetostatics (emqa)

Using the menu bar, select Physics > Boundary Settings. Verify or enter the default boundary setting (Magnetic insulation) for spherical boundaries 1-4, 21, 22, 31, and 32. See Figure 8.114. Click OK.

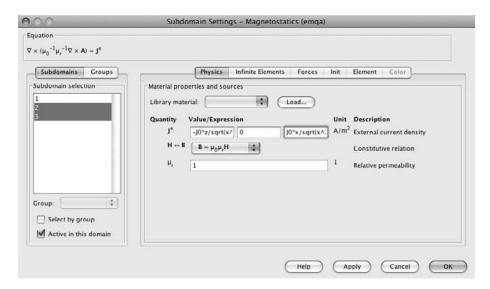


FIGURE 8.113 3D\_HC\_1 model Subdomain Settings (2, 3) edit window

FIGURE 8.114 3D\_HC\_1 model Boundary Settings (1–4, 21, 22, 31, 32) edit window

### **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Select "Coarser" from the Predefined mesh sizes pull-down list. See Figure 8.115.

Click the Subdomain tab. Select subdomains 2 and 3. Enter 0.05 in the Maximum element size edit window. See Figure 8.116.

o			6
Predefined mesh sizes:	oarser 🛟		Car
() Custom mesh size		1	Ap
Maximum element size:			He
Maximum element size scaling factor:	1.9		Cine
Element growth rate:	1.7		
Mesh curvature factor:	0.8		
Mesh curvature cutoff:	0.05		
Resolution of narrow regions:	0.3		
Optimize quality		1	
efinement method: Longest 🛟			

FIGURE 8.115 3D\_HC\_1 model Free Mesh Parameters edit window, Global tab

00			Free M	esh Para	meters		
	Global	Subdomain	Boundary	Edge	Point	Advanced	Ок
Subdo	main selecti		ubdomain mes Maximum elen Element growt	ient size:	0.05		Cancel Apply Help
_	elect by gro lect Remaini elect Meshe	ing					

FIGURE 8.116 3D\_HC\_1 model Free Mesh Parameters edit window, Subdomain tab

Click the Remesh button, and then click OK. See Figure 8.117.

### Solving the 3D\_HC\_1 Model

Using the menu bar, select Solve > Solve Problem.

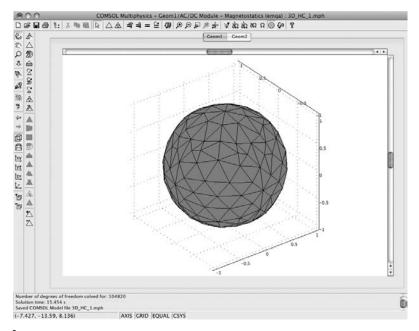
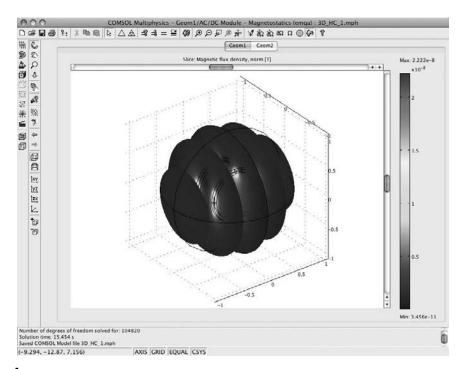


FIGURE 8.117 3D\_HC\_1 model mesh



**FIGURE 8.118** 3D\_HC\_1 model default solution plot

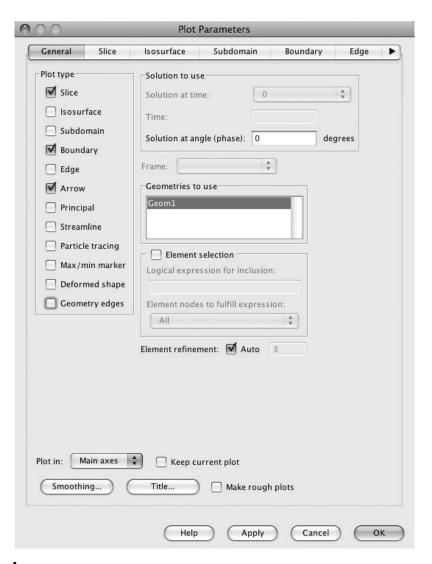
### Postprocessing and Visualization

The default solution plot is the slice plot. See Figure 8.118.

To see the model solution inside the sphere, the modeler needs to suppress the boundaries of the sphere. Using the menu bar, select Options > Suppress > Suppress Boundaries. Select boundaries 1-4, 21, 22, 31, and 32 in the Boundary selection window. See Figure 8.119. Click OK.

$0 \circ 0$	Suppress Boundaries
Boundary selec	ction:
27	Select Current Suppression
28	Sectedariantsappression
29	
30	Invert Suppression
31	
32	Suppress None
33	
34	
35	×
36	*

**FIGURE 8.119** 3D\_HC\_1 model plot Suppress Boundaries selection window



**FIGURE 8.120** 3D\_HC\_1 model Plot Parameters selection window, General tab

Using the menu bar, select Postprocessing > Plot Parameters > General. Check the Slice, Boundary, and Arrow check boxes. Uncheck the Geometry edges check box. See Figure 8.120. Click the Apply button.

Click the Slice tab. Select or verify the Predefined quantities: Magnetic flux density, norm. Enter 0 in the x and y levels edit windows and 1 in the z levels edit window. See Figure 8.121. Click the Apply button.

xpression: Init:	normB_emqa		Smooth	
lice positioning Nur	nber of levels	Vector with coore	linates	
levels: 💿 🛛	C			
levels: 💿 🛛 0	0			
levels: 💿	C	)		
Coloring and fill	terpolated 🛟	Fill style:	Filled	
lice color				
<ul> <li>Colormap:</li> </ul>	jet	Colors: 1024	Color sca	le
O Uniform color	Color			

**FIGURE 8.121** 3D\_HC\_1 model Plot Parameters selection window, Slice tab

Click the Boundary tab. Enter 1 in the Boundary data Expression edit window. Click the Uniform color radio button and choose black. Click OK. See Figure 8.122. Click the Apply button.

Click the Arrow tab. Select "Magnetic field" from the Predefined quantities pull-down list. Enter 24 in the x points window, 10 in the y points window, and 1 in the z points window. Uncheck the Scale factor Auto check box. Enter 0.5 in the Scale factor edit window. See Figure 8.123. Click the Apply button.

	Plot	Parameters		_
General Slice	Isosurface	Subdomain	Boundary	Edge
🗹 Boundary plot				
Boundary data				
Predefined quantities	:	\$	Range	
Expression:	1		Smooth	
Unit:	1	1		
Unic			9	
Coloring and fill				
Coloring:	rpolated	Fill style:	Filled	
Boundary color				
O Colormap:	jet 🔹	Colors: 1024	Color sca	le
🖲 Uniform color:	Color			
		. An		

**FIGURE 8.122** 3D\_HC\_1 model Plot Parameters selection window, Boundary tab

Click OK. See Figure 8.124.

### **Cross-Section Field Analysis**

To obtain a graphical plot of the magnetic field, use the Cross-Section Plot feature. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters > General. Click the Line/Extrusion plot radio button in the Plot type selection window. Click the Line/Extrusion tab. Select "Magnetic flux density, norm" from the Predefined quantities pull-down list in the y-axis data selection window. Select "x" from the x-axis data

Arrow plot			Plot arrows on:	Subdomains 🖨
l	Subdom	ain Data Bou	ndary Data Edge	Data
Predefined qu	antities:	Magnetic fiel	id 🗘	
x component: y component:		Hx_emqa		
		Hy_emqa		
z component	8	Hz_emqa		]
Unit:		A/m	*	
z points: 💿 Arrow paramete	1 ers	0		
Arrow type:	Cone	\$	Scale factor: 🗌 A	uto 0.5
	Propo	rtional 🛟	Color	

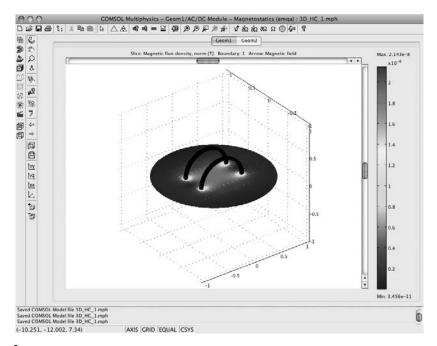
FIGURE 8.123 3D\_HC\_1 model Plot Parameters selection window, Arrow tab

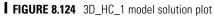
pull-down list. In the Cross-section line data edit windows, enter -0.8 for x0 and 0.8 for x1. See Figure 8.125.

Click the Apply button. See Figure 8.126. Click OK. See Figure 8.127.

# 3D Magnetic Field of a Helmholtz Coil: Summary and Conclusions

The 3D magnetic field of a Helmholtz coil model demonstrates the magnetic field uniformity of a Helmholtz coil pair. The magnetostatic modeling technique can be applied to a diverse collection of scientific and engineering applications (e.g., ranging from magnetometers and Hall effect sensors to biomagnetic and medical studies).





Line plot	C	) Extrusio	n plot	
y-axis data Predefined quantities:	Magnetic	flux densi	ty, norm	•
Expression:	normB_emo	la		
Unit:	Т			•
x-axis data		ss-section	line data	
• ×	x0:	-0.8	×1:	0.8
O Expression.	. yo:	0	y1:	0
	z0.	0	z1.	0
	Line	resolution	1.	200
Line Settings	) (Surfac	e Settings		

I FIGURE 8.125 3D\_HC\_1 model Cross-Section Plot Parameters edit window

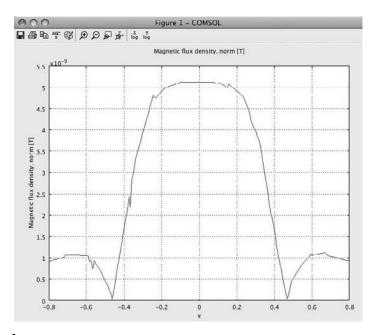


FIGURE 8.126 3D\_HC\_1 model cross-section plot

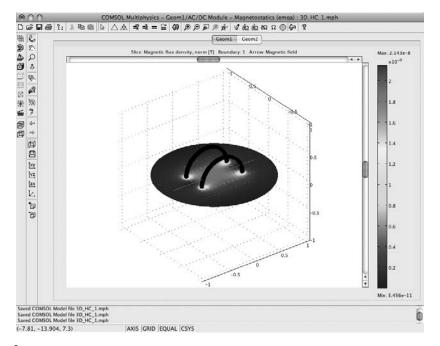


FIGURE 8.127 3D\_HC\_1 model solution plot with cross-section plot line

0 0	Model Navigato	r	-	
New	Model Library User Models	Open	Settings	
Space dimension:	3D 🛟		Module	
Conducti Electrosta Electrosta Magnetos Magnetos	ductive Media DC <i>ve</i> Media DC titos titos, Generalized tatics, Ungauged AV tatics, No Currents , Electric	Descriptio Magnetos	C	ting and
Dependent variables:	Ax Ay Az psi			
Application mode name: Element:	emqa Vector - Quadratic		Multiphysics	

**FIGURE 8.128** 3D\_HC\_2 Model Navigator setup

# 3D Magnetic Field of a Helmholtz Coil with a Magnetic Test Object

The following numerical solution model (3D\_HC\_2 model) is derived from a model that was originally developed by COMSOL as an AC/DC Module Electrical Components demonstration model. That model was developed for distribution with the Multiphysics software as a COMSOL AC/DC Module Model Library.

To start building the 3D\_HC\_2 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "3D" from the Space dimension pull-down list. Select AC/DC Module > Statics > Magnetostatics. See Figure 8.128. Click OK.

### Constants

Using the menu bar, select Options > Constants. Enter the constant shown in Table 8.17. Click OK. See Figure 8.129.

Name	Expression	Description	Figure Number
JO	1[A/m^2]	Coil current density	8.129

Table 8.17 Constants Edit Winde	ow
---------------------------------	----

Name	Expression	Value	Description
10	1[A/m^2]	1[A/m <sup>2</sup> ]	Coil current density

FIGURE 8.129 3D\_HC\_2 model Constants edit window

Select File > Save As. Enter 3D\_HC\_2.mph in the Save As edit window. See Figure 8.130. Click the Save button.

#### **Geometry Modeling**

Each Helmholtz coil is created in cross section by drawing squares in the 2D work-plane. The modeler then creates a solid coil by revolution (revolve) of the 2D work-plane geometry into the 3D geometry.

0 0 0 Save As: 3D_H	Save As
10_Mc	
Name 3D_ESP_1.mph 3D_ESP_2.mph 3D_HC_1.mph 3D_TLR_1.mph 3D_TLR_2.mph	<ul> <li>Date Modified</li> <li>Monday, April 13, 2009 5:38 PM</li> <li>Tuesday, April 14, 2009 5:35 PM</li> <li>Friday, April 17, 2009 2:10 PM</li> <li>Friday, April 10, 2009 12:28 PM</li> <li>Friday, April 10, 2009 5:28 PM</li> </ul>
File Format:	COMSOL Multiphysics Mo
New Folder	Cancel Save

FIGURE 8.130 3D\_HC\_2 model Save As edit window

Quick Face	Parallel Edge A	ngle Vertices	Advanced	
ane				OK
• x-y z: 0				Cane
Ο γ-z x: 0				App
O z-x y: 0				
				Hel
				Previe

**FIGURE 8.131** 3D\_HC\_2 model Work-Plane Settings edit window

Using the menu bar, select Draw > Work-Plane Settings. See Figure 8.131. Click OK.

Using the menu bar, select Draw > Specify Objects > Square. In the Square edit window, enter the information shown in Table 8.18. Click OK after filling in the parameters of each separate square in the appropriate edit window. See Figures 8.132 and 8.133.

#### Table 8.18Geometry Components

Name	Width	Base	х	у	Figure Number
SQ1	0.05	Corner	-0.425	0.175	8.132
SQ2	0.05	Corner	-0.425	-0.225	8.133

Size		Rotat	ion angle	
Width:	0.05	α:	0	(degrees)
Positio	n			
Base:	Corner	\$ Style:	Solid	\$
x:	-0.425	Name:	SQ1	
<b>y</b> :	0.175			

FIGURE 8.132 3D\_HC\_2 model Square SQ1 edit window

FIGURE 8.133 3D\_HC\_2 model Square SQ2 edit window

Click the Zoom Extents button. Select SQ1 and SQ2. Using the menu bar, select Draw > Revolve. See Figure 8.134.

Click OK. See Figure 8.135.

Using the menu bar, select Draw > Ellipsoid. Enter x = 0.05, y = 0.3, and z = 0.05 in the Length semiaxes edit windows. Enter x = 0, y = 0 and z = 1 in the Cartesian coordinates edit windows. See Figure 8.136. Click OK.

Using the menu bar, select Draw > Sphere. Enter 1 in the Radius edit window. See Figure 8.137. Click OK.

Click the Zoom Extents button. Click on the display window background outside the sphere. See Figure 8.138.

000	Revolve
Objects to revolve:	Angles of revolution
5Q1 5Q2	α1: 0 (degrees)
3Q2	α2: 360 (degrees)
	Revolution axis
	Point on axis:
	x: 0
	y: 0
	Axis direction through:
Revolve to geometry:	Second point O Angle from x-axis
Geom1	
Revolved object name:	x:         0         θ:         90         (degrees)
REV1	y: 1

FIGURE 8.134 3D\_HC\_2 model Revolve edit window

**FIGURE 8.135** 3D\_HC\_2 model Helmholtz coil pair

000	Ellipsoid		_
Style Solid	Axis base point x: 0 y: 0	Rotation angle	legrees
Length semiaxes	z: 0		
X: 0.05 Y: 0.3	Axis direction vector	ates 🔘 Spherical coordi	nates
Z: 0.05	x: 0	θ: 0 (de	grees)
lame: ELP1	y: 0 z: 1	φ: 0 (de	grees)

FIGURE 8.136 3D\_HC\_2 model Ellipsoid edit window

000	Sphere		
Style Solid	Axis base point x: 0	Rotation α: 0	angle (degrees
○ Face	у: 0		
Sphere parameters	z: 0		
Radius: 1	Axis direction vector		
Name: SPH1	Cartesian coordinate	es 🔘 Spher	ical coordinates
	x: 0	θ: 0	(degrees)
	y: 0	φ: 0	(degrees)
	z: 1		

**FIGURE 8.137** 3D\_HC\_2 model Sphere edit window

Subdomain	Quantity	Value/Expression	Figure Number
1	Je	000	8.139
	μ <sub>r</sub>	1	
2, 3	Je	-J0*z/sqrt(x^2+z^2) 0 J0*x/sqrt(x^2+z^2)	8.140
	μ <sub>r</sub>	1	
4	Je	000	8.141
	μ <sub>r</sub>	15000	

#### Physics Subdomain Settings: Magnetostatics (emqa)

Using the menu bar, select Physics > Subdomain Settings. Select the Subdomain(s) and enter the expression indicated in Table 8.19. Click OK. See Figures 8.139, 8.140, and 8.141.

The numerical value of the permeability for free space ( $\mu_0$ ) in SI units is exactly  $4\pi \times 10^{-7}$  H/m. That value is the default value for permeability incorporated into the COMSOL Multiphysics software. The permeability of a material is the product of  $\mu_0$  and  $\mu_r$ , which in this case is 1. FIGURE 8.140 3D\_HC\_2 model Subdomain Settings (2, 3) edit window

## Physics Boundary Settings: Magnetostatics (emqa)

Using the menu bar, select Physics > Boundary Settings. Verify or enter the default boundary setting (Magnetic insulation) for spherical boundaries 1-4, 25, 26, 37, and 40. See Figure 8.142. Click OK.

× A = 0	
Boundaries Croups Boundary selection Boundary selection Boundary selection Croup: Select by group Interior boundaries	Conditions Color Boundary sources and constraints Boundary condition: Magnetic insulation

FIGURE 8.142 3D\_HC\_2 model Boundary Settings (1–4, 25, 26, 37, 40) edit window

#### **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Select "Coarser" from the Predefined mesh sizes pull-down list. See Figure 8.143.

Click the Subdomain tab. Select subdomains 2 and 3. Enter 0.05 in the Maximum element size edit window. See Figure 8.144.

Global Subdomain Bou	ndary Edge Point	Advanced	
Predefined mesh sizes:     C	oarser 🛟		Can
O Custom mesh size			Ap
Maximum element size:			He
Maximum element size scaling factor:	1.9		Спе
Element growth rate:	1.7		
Mesh curvature factor:	0.8		
Mesh curvature cutoff:	0.05		
Resolution of narrow regions:	0.3		
🗹 Optimize quality			
efinement method: 🛛 Longest 🛟			

FIGURE 8.143 3D\_HC\_2 model Free Mesh Parameters edit window, Global tab

Glubal	Subdomain	Boundary	Edge I	Point	Advanced		OK
subdomain sele 1 2 3 4	N	bdomain mesh faximum elemei lement growth i	nt size:	s 0.05		]	Cano App Hel
Select by g							

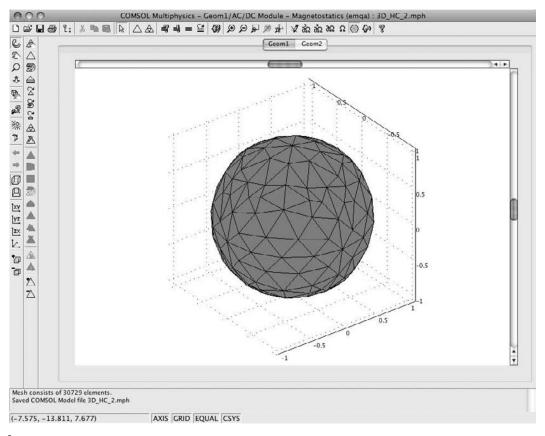
FIGURE 8.144 3D\_HC\_2 model Free Mesh Parameters edit window, Subdomain (2, 3) tab

Select subdomain 4. Enter 0.03 in the Maximum element size edit window. See Figure 8.145.

Click the Remesh button, and then click OK. See Figure 8.146.

20			Free M	esh Para	meters		_	-
-	Global	Subdomain	Boundary	Edge	Puint	Advanced		
Subdor 1 2 3 4	nain select	N	bdomain mes Maximum elem Element growtl	ent size:	0.03			Cano App Hel
Se	lect by gro	oup						
_	ect Remain elect Mesh							
_								

FIGURE 8.145 3D\_HC\_2 model Free Mesh Parameters edit window, Subdomain (4) tab





#### Solving the 3D\_HC\_2 Model

Using the menu bar, select Solve > Solve Problem.

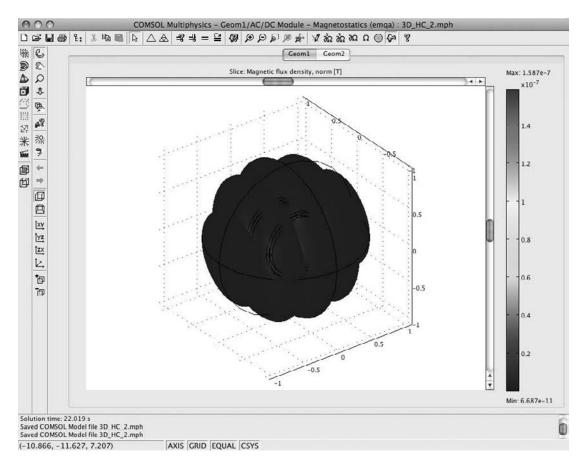
#### **Postprocessing and Visualization**

The default solution plot is the slice plot. See Figure 8.147.

To see the model solution inside the sphere, the modeler needs to suppress the boundaries of the sphere. Using the menu bar, select Options > Suppress > Suppress Boundaries. Select boundaries 1-4, 25, 26, 37, and 40 in the Boundary selection window. See Figure 8.148. Click OK.

Using the menu bar, select Postprocessing > Plot Parameters > General. Check the Slice, Boundary, and Arrow check boxes. Uncheck the Geometry edges check box. See Figure 8.149. Click the Apply button.

663



**FIGURE 8.147** 3D\_HC\_2 model default solution plot

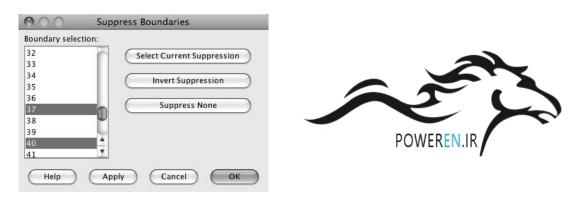


FIGURE 8.148 3D\_HC\_2 model plot Suppress Boundaries selection window

General Slice	Isosurface Subdomain Boundary Edge
Plot type	Solution to use
Slice	Solution at time: 0
Isosurface	Time:
Subdomain	Solution at angle (phase): 0 degrees
M Boundary	Solution at angle (phase).
Edge	Frame:
Arrow	Geometries to use
Principal	Geom1
Streamline	
Particle tracing	
Max/min marker	Element selection
Deformed shape	Logical expression for inclusion.
Geometry edges	Element nodes to fulfill expression:
	All
ot in: Main axes	Element refinement: 🗹 Auto 3

FIGURE 8.149 3D\_HC\_2 model Plot Parameters selection window, General tab

Click the Slice tab. Select or verify the Predefined quantities: Magnetic flux density, norm. Enter 0 in the x levels edit window, 0 in the y levels edit window, and 1 in the z levels edit window. See Figure 8.150. Click the Apply button.

Click the Boundary tab. Enter 1 in the Boundary data Expression edit window. Click the Uniform color radio button, and choose black. Click OK. See Figure 8.151. Click the Apply button.

Click the Arrow tab. Select "Magnetic field" from the Predefined quantities pulldown list. Enter 24 in the x points window, 10 in the y points window, and 1 in the z points window. Uncheck the Scale factor Auto check box. Enter 0.5 in the Scale factor edit window. See Figure 8.152. Click the Apply button.

Expression: Unit:	normB_emqa			\$	Smooth		
Slice positionin	g						
x levels: 💿		r of levels	0	Vector with	coordinate	25	
	0		0				
y levels: 💿 z levels: 💿	0		0				
Coloring and fi Coloring: (		olated	\$	Fill style:	Fil	led	\$
Slice color © Colormap O Uniform c		jet Color		Colors:	1024	Colo	r scale
				_			

FIGURE 8.150 3D\_HC\_2 model Plot Parameters selection window, Slice tab

Click OK. See Figure 8.153.

## **Cross-Section Field Analysis**

To obtain a graphical plot of the magnetic field, use the Cross-Section Plot feature. Using the menu bar, select Postprocessing > Cross-Section Plot Parameters > General. Click the Line/Extrusion plot radio button in the Plot type selection window. Click the Line/Extrusion tab. Select "Magnetic flux density, norm" from the Predefined quantities pull-down list in the y-axis data selection window. Select "x" from the x-axis data 1

) ()		Plot	Parameters		-
General	Slice	Isosurface	Subdomain	Boundary	Edge
🗹 Boundar	y plot				
Boundary da	ta				
Predefined q	uantities:		\$	Range	
Expression:		1		Smooth	
Unit:			1	7	
				2	
Coloring and			6		-
Coloring:	Interp	olated	Fill style:	Filled	
Boundary co					
O Colorm	ap:	jet 🛟	Colors: 1024	Color sca	ile
O Uniform	n color: (	Color			

**FIGURE 8.151** 3D\_HC\_2 model Plot Parameters selection window, Boundary tab

pull-down list. In the Cross-section line data edit windows, enter -0.8 for x0 and 0.8 for x1. See Figure 8.154.

Click the Apply button. See Figure 8.155. Click OK. See Figure 8.156.

#### 3D Magnetic Field of a Helmholtz Coil with a Magnetic Test Object: Summary and Conclusions

The 3D magnetic field of a Helmholtz coil with a magnetic test object model demonstrates the magnetic field concentration, when a high relative permeability object lies within the field of the Helmholtz coil. This magnetostatic modeling technique can be applied to a diverse collection of scientific and engineering test, measurement, and design applications.

Arrow plot			Plot arrows on:	Subdoma	ins 🛊
	Subdom	ain Data Bou	ndary Data Edge	Data	
Predefined qu	antities:	Magnetic fiel	d 😂	)	
x component		Hx_emqa			
y component:		Hy_emqa			
z component:		Hz_emqa		]	
Unit:		A/m	\$	)	
y points: 🕑	10	0			
Arrow paramete	1	100	<u></u>		
Arrow type:	Cone	•	Scale factor:	Auto 0.5	
Arrow length	Propo	rtional 🗘	Color		

FIGURE 8.152 3D\_HC\_2 model Plot Parameters selection window, Arrow tab

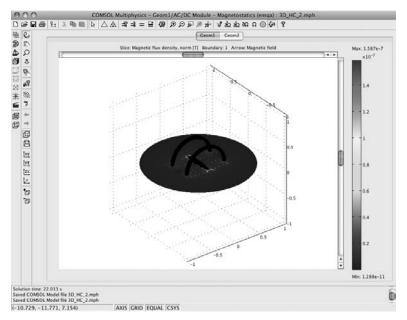
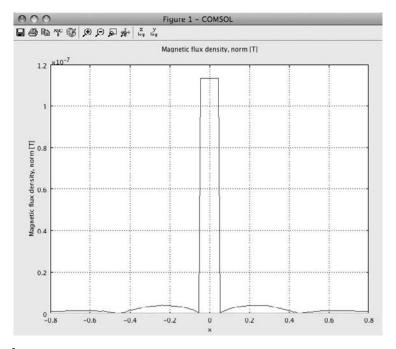


FIGURE 8.153 3D\_HC\_2 model solution plot

lot type Line plot	O Extrusion plot	
-axis data		
redefined quantities:	Magnetic flux density, no	rm 🗘
xpression:	ormB_emqa	
Jnit:	т	:
-axis data	Cross-section line of	lata
⊙ (x 😫	x0: -0.8	<1: 0.8
O Expression	) yo: 0 y	/1: 0
	z0: 0	£1. 0
	Line resolution:	200
Line Settings)	(Surface Settings)	

**FIGURE 8.154** 3D\_HC\_2 model Cross-Section Plot Parameters edit window



**FIGURE 8.155** 3D\_HC\_2 model cross-section plot

FIGURE 8.156 3D\_HC\_2 model solution plot with cross-section plot line

# References

- 1. http://en.wikipedia.org/wiki/Thin\_film
- 2. http://en.wikipedia.org/wiki/Layer\_(electronics)
- 3. http://en.wikipedia.org/wiki/Coating
- 4. http://focus.ti.com/lit/an/slyt209a/slyt209a.pdf
- 5. http://en.wikipedia.org/wiki/Electrical\_resistance
- 6. http://en.wikipedia.org/wiki/Voltage\_divider
- 7. http://en.wikipedia.org/wiki/Thales\_of\_Miletus
- 8. http://en.wikipedia.org/wiki/Otto\_von\_Guericke
- 9. http://en.wikipedia.org/wiki/Electrostatic\_generator
- 10. http://en.wikipedia.org/wiki/Charles-Augustin\_de\_Coulomb
- 11. http://en.wikipedia.org/wiki/Carl\_Friedrich\_Gauss
- 12. http://en.wikipedia.org/wiki/Electrostatics

- 13. http://en.wikipedia.org/wiki/Gauss%27s\_Law
- 14. http://en.wikipedia.org/wiki/Laplacian\_operator
- 15. http://en.wikipedia.org/wiki/Vector\_Laplacian
- 16. http://en.wikipedia.org/wiki/Pierre-Simon\_de\_Laplace
- 17. http://en.wikipedia.org/wiki/Maxwell%27s\_Equations
- 18. http://en.wikipedia.org/wiki/James\_Clerk\_Maxwell
- 19. http://en.wikipedia.org/wiki/Gauge\_Transformation
- 20. http://en.wikipedia.org/wiki/Coulomb\_gauge
- COMSOL AC/DC Module User's Guide, Version 3.4, October 2007, COSMOL AB, Stockholm, Sweden, p. 92

#### Exercises

- 1. Build, mesh, and solve the 3D thin layer resistance model, thin layer approximation problem presented in this chapter.
- 2. Build, mesh, and solve the 3D thin layer resistance model, thin layer subdomain problem presented in this chapter.
- 3. Build, mesh, and solve the 3D electrostatic potential between two cylinders problem presented in this chapter.
- 4. Build, mesh, and solve the 3D electrostatic potential between five cylinders problem presented in this chapter.
- 5. Build, mesh, and solve the 3D magnetic field of a Helmholtz coil (static) problem presented in this chapter.
- 6. Build, mesh, and solve the 3D magnetic field of a Helmholtz coil with a magnetic test object problem presented in this chapter.
- 7. Explore other materials as applied in the 3D thin layer resistance models.
- 8. Explore other materials as applied in the model of a 3D magnetic field of a Helmholtz coil with a magnetic test object.
- 9. Explore adding more and/or different magnetic test objects to the 3D magnetic field of a Helmholtz coil model.
- 10. Explore the different geometries in the 3D thin layer resistance models.

# 9

# **Perfectly Matched** Layer Models

# In This Chapter

Perfectly Matched Layer Modeling Guidelines and Coordinate Considerations PML Theory

PML Models

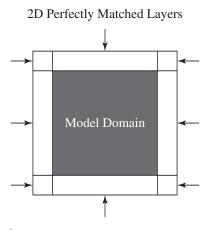
- 2D Dielectric Lens Model, with PMLs
- 2D Dielectric Lens Model, with PMLs: Summary and Conclusions
- 2D Dielectric Lens Model, without PMLs
- 2D Dielectric Lens Model, with and without PMLs: Summary and Conclusions
- 2D Concave Mirror Model, with PMLs
- 2D Concave Mirror Model, with PMLs: Summary and Conclusions
- 2D Concave Mirror Model, without PMLs
- 2D Concave Mirror Model, with and without PMLs: Summary and Conclusions

# Perfectly Matched Layer Modeling Guidelines and Coordinate Considerations

# **PML** Theory

One of the underlying fundamental difficulties in electromagnetic wave equation calculations (Maxwell's equations<sup>1</sup>) is dealing with a propagating wave interacting with boundaries (reflections). If the boundary of the model domain is terminated in the typical fashion,<sup>2</sup> unwanted reflections are typically incorporated into the solution. Fortunately, there is a methodology that works sufficiently well to essentially eliminate reflection problems at the domain boundary. That methodology is the perfectly matched layer (PML).

The PML is an approximation methodology originally developed in 1994 by Jean-Pierre Berenger<sup>3</sup> for use with FDTD<sup>4</sup> (finite-difference time-domain) electromagnetic modeling calculations. The PML technique has since been adapted and applied to other calculational methodologies that have similar domain mediated needs (e.g., FEM and others).<sup>5</sup> This methodology can be applied to a large variety of diverse wave equation problems.<sup>6</sup> In this chapter, however, it is applied only to electromagnetic problems within the context of the COMSOL RF Module.<sup>7</sup>



**FIGURE 9.1** 2D Cartesian domain with PMLs

NOTE For a broader detailed application of the PML methodology to other types of wave problems, refer to the literature.

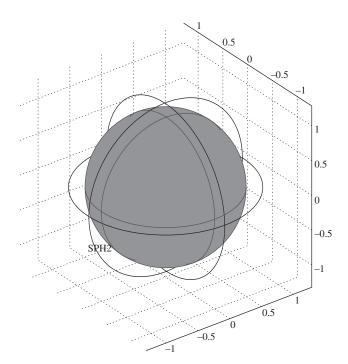
In the COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software, the PML technique is explicitly available in the RF Module, for application to electromagnetics modeling problems. The function of the PML methodology is to add anisotropic attenuating domains (layers) outside the modeled domain, so that the modeled domain has substantially reflectionless boundaries. Examples of modeling domains with PMLs include a 2D Cartesian domain with PMLs (Figure 9.1), a 3D Cartesian domain with PMLs (Figure 9.2), a 3D spherical domain with PMLs (Figure 9.3), and a 3D cylindrical domain with PMLs (Figure 9.4). The coordinate systems employed with the domain structures are those that are associated with their respective geometries.

Note To achieve the desired behavior of the wave equation PDE, the entire model domain, including the perfectly matched layers, is transformed to a complex coordinate system. For a Cartesian system (x, y, z), the transformation occurs as follows:

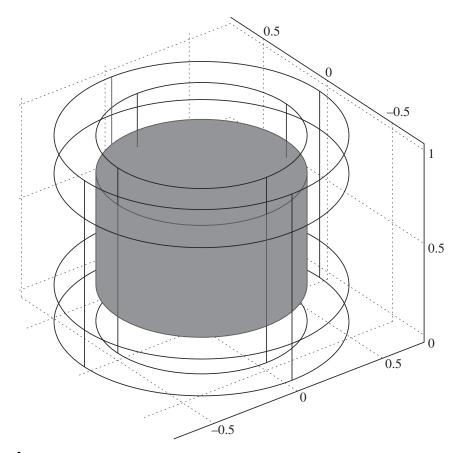
$$\frac{\partial}{\partial x} \to \frac{1}{1 + \frac{i\sigma(x)}{\omega}} \frac{\partial}{\partial x}; \quad \frac{\partial}{\partial y} \to \frac{1}{1 + \frac{i\sigma(y)}{\omega}} \frac{\partial}{\partial y}; \quad \frac{\partial}{\partial z} \to \frac{1}{1 + \frac{i\sigma(z)}{\omega}} \frac{\partial}{\partial z}$$
(9.1)

where  $\sigma(x, y, z)$  is a step function that is zero inside the solution domain and a positive real number or an appropriate function of the designated coordinate variable (*x*, *y*, *z*) outside the solution domain and inside the PML.





**FIGURE 9.3** 3D spherical domain with PMLs



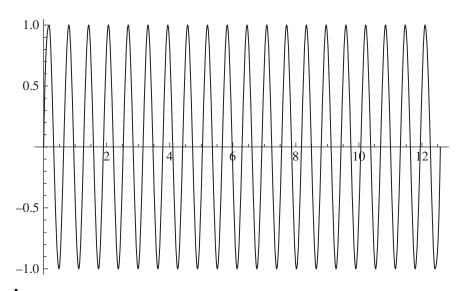
**FIGURE 9.4** 3D cylindrical domain with PMLs

The transformation of the PDE in this fashion results in a solution with a multiplicative term that is, in general, as follows:

$$F(x, y, z) = f(x, y, z) * e^{-\frac{k\sigma(x, y, z)}{\omega}}$$
(9.2)

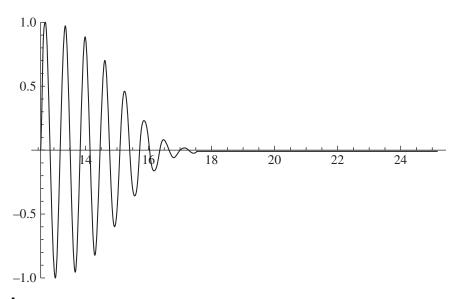
where  $F(x, y, z) = f(x, y, z) * e^{-0}$  (the solution inside the domain)  $F(x, y, z) = f(x, y, z) * e^{-\kappa\sigma(x, y, z)/\omega}$  (the decaying solution within the PML domain)

At the outer PML boundary, the preferred boundary condition is the scattering boundary. However, if the attenuation of the propagating wave at the outer boundary of the PML is sufficient, then the particular boundary condition invoked is largely irrelevant. In such a case, the amplitude of the reflected wave will be sufficiently small as not to contribute to the final solution.



**FIGURE 9.5** Wave equation solution example inside the modeling domain

Figures 9.5 and 9.6 show examples of wave equation solutions. For an example of the wave equation solution inside the modeling domain, see Figure 9.5. For an example of the wave equation solution inside the PML domain, see Figure 9.6.



**FIGURE 9.6** Wave equation solution example inside the PML domain

00	Model Navigat	or
Space dimension: Application Mode COMSOL Mult AC/DC Modu AC/DC Module MEMS Module RF Module F In-Plane V TE Wa	v Model Library User Model 2D 2D 3 s iphysics e Module Aves ves	
RF Module	ves rmonic propagation genfrequency analysis ansient propagation attered harmonic propagation	Description: Transverse electric (TE) wave propagation in the plane. Harmonic propagation analysis for the scattered wave.
Application mode nar	ne: rfweh	Multiphysics

FIGURE 9.7 2D\_PML\_DL\_1 Model Navigator setup

#### PML Models

#### 2D Dielectric Lens Model, with PMLs

The dielectric lens is a concept borrowed from optical physics. In this application, the principles of optics are applied to lower-frequency electromagnetic waves to focus the impinging wavefront into the region of a sensor. The act of focusing the wavefront effectively amplifies the magnitude of the impinging signal.

The following numerical solution model (2D\_PML\_DL\_1 model) is derived from a model that was originally developed by COMSOL as an RF Module tutorial model for the demonstration of the PML methodology. That model was developed for distribution with the RF Module software as part of the COMSOL RF Module Model Library.

To start building the 2D\_PML\_DL\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select RF Module > In-Plane Waves > TE Waves > Scattered harmonic propagation. See Figure 9.7. Click OK.

Name	Width	Height	Base	Х	Y	Figure Number
R1	2.4	0.2	Corner	-1.2	1.0	9.8
R2	2.4	0.2	Corner	-1.2	-1.2	9.9
R3	0.2	2.4	Corner	-1.2	-1.2	9.10
R3	0.2	2.4	Corner	1.0	-1.2	9.11

#### Table 9.1 Geometry Components

**NOTE** The Model Navigator command sequence (In-Plane Waves > TE Waves > Scattered harmonic propagation) selects a transverse electric field (*z*-direction) wave traveling in the plane (x, y-plane) of the modeling domain.

#### **Geometry Modeling**

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 9.1. Click OK after filling in the parameters of each separate rectangle in the Rectangle edit window. See Figures 9.8–9.11.

Size		Rotat	ion angle	
Width:	2.4	α:	0	(degrees)
Height:	0.2			
Position				
Base:	Corner	Style:	Solid	\$
<b>x</b> :	-1.2	Name:	R1	
y:	1.0			

FIGURE 9.8 2D\_PML\_DL\_1 model Rectangle (R1) edit window

Size		Rotation angle
Width:	2.4	α: 0 (degree
Height:	0.2	
Position		
Base:	Corner	Style: Solid
<b>x</b> :	-1.2	Name: R2
y:	-1.2	

FIGURE 9.9 2D\_PML\_DL\_1 model Rectangle (R2) edit window

000	1	Rectangle	-	
Size		Rotat	ion angle-	
Width:	0.2	α:	0	(degrees)
Height:	2.4			
Position				
Base:	Corner	\$ Style:	Solid	\$
x:	-1.2	Name:	R3	
	-1.2			

FIGURE 9.10 2D\_PML\_DL\_1 model Rectangle (R3) edit window

Click the Zoom Extents button before drawing the next rectangle. Figure 9.12 shows the PML rectangles of model 2D\_PML\_DL\_1.

Select File > Save As. Enter 2D\_PML\_DL\_1.mph in the Save As edit window. See Figure 9.13. Click the Save button.

Using the menu bar, select Draw > Specify Objects > Circle. Enter Radius = 0.5, Base = Center, x = -0.5, and y = 0. See Figure 9.14. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.5, Height = 1.0, Base = Corner, x = -1.0, and y = -0.5. See Figure 9.15. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter C1-R5 in the Set formula edit window. See Figure 9.16.

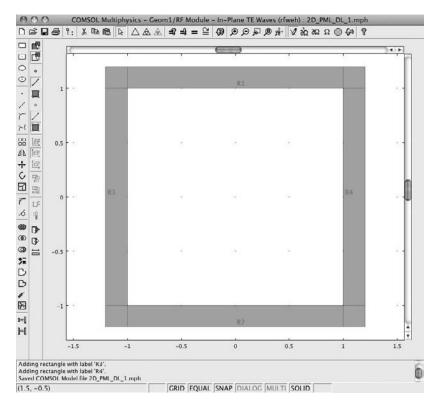


FIGURE 9.12 2D\_PML\_DL\_1 model PML rectangles

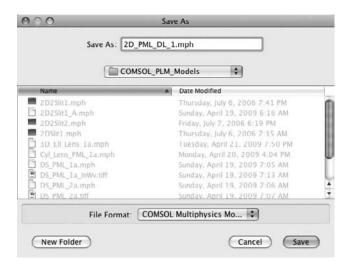


FIGURE 9.13 2D\_PML\_DL\_1 model Save As edit window

Size		Rotat	ion angle	
Radius	0.5	α:	0	(degrees
Positio	n			
Base:	Center	\$ Style:	Solid	\$
x:	-0.5	Name:	C1	
y:	0			

**FIGURE 9.14** 2D\_PML\_DL\_1 model Circle (C1) edit window

Size		Rotat	ion angle	
Width:	0.5	α:	0	(degrees)
Height:	1.0			
Position				
Base:	Corner	\$ Style:	Solid	\$
		Name:	R5	
x:	-1.0			

**FIGURE 9.15** 2D\_PML\_DL\_1 model Rectangle (R5) edit window

000 c	reate Composite Object
Solids     Curves     Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
R1	C1-R5
R2	
R3	Keep interior boundaries
R4	
C1	Repair
R5	Repair tolerance: 1.0E-4

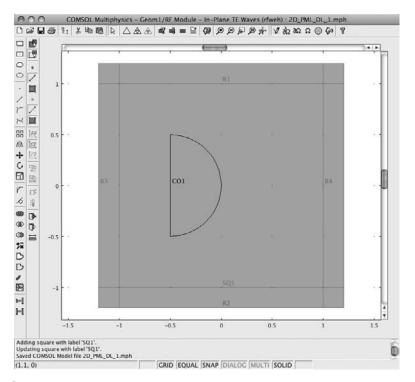
I FIGURE 9.16 2D\_PML\_DL\_1 model Create Composite Object edit window

**FIGURE 9.17** 2D\_PML\_DL\_1 model dielectric lens (CO1)

Click OK. See Figure 9.17.

Using the menu bar, select Draw > Specify Objects > Square. Enter Width = 2, Base = Center, x = 0, and y = 0. See Figure 9.18. Click OK.

Figure 9.19 shows the model domain plus PMLs. Having established the geometry for the 2D\_PML\_DL\_1 model, the next step is to define the fundamental Physics properties.



**FIGURE 9.19** 2D\_PML\_DL\_1 model domain plus PMLs

#### Physics Application Mode Properties: In-Plane TE Waves (rfweh)

Select Physics > Properties. Select "Free space wavelength" from the Specify wave using pull-down list. See Figure 9.20. Click OK.

Default element type:	Lagrange - Quadratic	
belaut element type.	Lugrange Quadrane	11.
Analysis type:	Harmonic propagation	•
Field type:	Scattered TE waves	•
Specify wave using:	Free space wavelength	
Specify eigenvalues using:	Figenfrequency	Å. T
Divergence condition:	Off	Å.
Weak constraints:	Off	
Vector element constraint:	Off	-0
Constraint type:	Ideal	4

**FIGURE 9.20** 2D\_PML\_DL\_1 model Application Mode Properties edit window

Name	Expression	Unit	Description
epsilon0_rfweh	8.854187817e-12	F/m	Permittivity of vacuum
mu0_rfweh	4*pi*1e-7	H/m	Permeability of vacuum
lambda0_rfweh	0.5	m	Free space wavelength
E0iz_rfweh	exp(-j*k0_rfweh*x)	V/m	Incident electric field, z component
<b>đ</b>			
Synchronize	e equivalent variables		oply Cancel OK

FIGURE 9.21 2D\_PML\_DL\_1 model Application Scalar Variables (lambda0\_rfweh) edit window

### Physics Application Scalar Variables: In-Plane TE Waves (rfweh)

Select Physics > Application Scalar Variables. Enter 0.5 in the lambda0\_rfweh edit window. See Figure 9.21. Click OK.

#### Physics Subdomain Settings: In-Plane TE Waves (rfweh)

Having established the basic Physics settings for the 2D\_PML\_DL\_1 model, the next step is to define the fundamental Physics subdomain settings. Select Physics > Subdomain Settings. Click the PML tab. Select subdomains 1–4, 6, and 8–10 (the PMLs). Select "Cartesian" from the Type of PML pull-down list. Click the Apply button. See Figure 9.22.

Subdomains Groups	Physi	cs PML	Init Element	Color
Subdomain selection	Type of PML: Absorbing in x direct Absorbing in y direct x <sub>0</sub> , y <sub>0</sub>	Sux_gue		Description <sup>m</sup> Width in x direction <sup>m</sup> Width in y direction <sup>m</sup> Center point
Group: + Select by group Active in this domain				

FIGURE 9.22 2D\_PML\_DL\_1 model Subdomain Settings, PML type selection

Subdomains Groups		Physics PML Init	Element Color	
1       2       3       4       5       6       7       8       9       Croup:       *       Select by group       Image: Select by group       Image: Select by group	Type of PML: Absorbing in x Absorbing in y x <sub>0</sub> , y <sub>0</sub>	Jux_guess_i	fweh <sup>m</sup> Width i	in x direction in y direction

FIGURE 9.23 2D\_PML\_DL\_1 model Subdomain Settings, x absorption

Select subdomains 1–3, and 8–10 (the vertical PMLs). Check the Absorbing in x direction check box. Click the Apply button. See Figure 9.23.

Select subdomains 1, 3, 4, 6, 8, and 10 (the horizontal PMLs). Check the Absorbing in y direction check box. Click the Apply button. See Figure 9.24.

Subdomains Groups	Physics	PML Init	t Element	Color
subdomain selection	Type of PML: Absorbing in x direction Absorbing in y direction x <sub>0</sub> , y <sub>0</sub>	Cartesian Value/Expres Sdx_guess_ Sdy_guess_ 0	rfweh	Description <sup>m</sup> Width in x direction <sup>m</sup> Width in y direction <sup>m</sup> Center point
Active in this domain				

**FIGURE 9.24** 2D\_PML\_DL\_1 model Subdomain Settings, y absorption

	$E_z = 0, \epsilon_r = n^2$	
Subdomains Groups	Physics PML	Init Element Color
ubdomain selection	Material properties	
2 7	Library material:	$(\cdot, \cdot)$
3	Library material:	(Load)
4	Quantity Value/Expression	Unit Description
6	<ul> <li>Specify material properties in terms of</li> </ul>	of refractive index
7	n	
3 🔍	1	<sup>1</sup> Refractive index
9 10 <b>*</b>	Specify material properties in terms of the second seco	of $\epsilon_r$ , $\mu_r$ , and $\sigma$
-	Fr 1	1 Relative permittivity
iroup:		
Select by group	0	5/m Electric conductivity
Active in this domain	μ, 1	1 Relative permeability

**FIGURE 9.25** 2D\_PML\_DL\_1 model Subdomain Settings, Physics tab, subdomain 5

Click the Physics tab. Select subdomain 5 (the model domain). Enter  $\varepsilon_r = 1$ ,  $\sigma = 0$ , and  $\mu_r = 1$ . Click the Apply button. See Figure 9.25.

Select subdomain 7 (the dielectric lens). Enter  $\varepsilon_r = 3$ ,  $\sigma = 0$ , and  $\mu_r = 1$ . Click the Apply button. See Figure 9.26. Click OK.

Subdomains Groups	Physics PML Init Element Color
subdomain selection	Material properties         Library material:         Quantity Value/Expression         Unit Description         Specify material properties in terms of refractive index         n       1         Refractive index         Specify material properties in terms of ε <sub>r</sub> , μ <sub>r</sub> , and σ
Group: (*) Select by group	Fr     3     1     Relative permittivity       σ     0     S/m     Electric conductivity       μr     1     Relative permeability

FIGURE 9.26 2D\_PML\_DL\_1 model Subdomain Settings, Physics tab, subdomain 7

Boundaries Groups	Conditions	Material Properties Pro	ort Far-Field Color/Style
Boundary selection	Boundary sources and con Boundary condition:		ondition
2 3	Quantity	Value/Expression	Unit Description
4 5	Wave type:	Plane wave 🛟	
2			
9			
11			
13			
15			
Group:			
Select by group			

FIGURE 9.27 2D\_PML\_DL\_1 model Boundary Settings

#### Physics Boundary Settings: In-Plane TE Waves (rfweh)

Having established the subdomain settings for the 2D\_PML\_DL\_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, select Physics > Boundary Settings. Select boundary 1. Check the Select by Group check box to select all the outer edges of the PMLs (boundaries). Select "Scattering boundary condition" from the Boundary condition pull-down list. See Figure 9.27. Click OK.

#### **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 7 (the dielectric lens). Enter 0.05 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 9.28.

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 9.29.

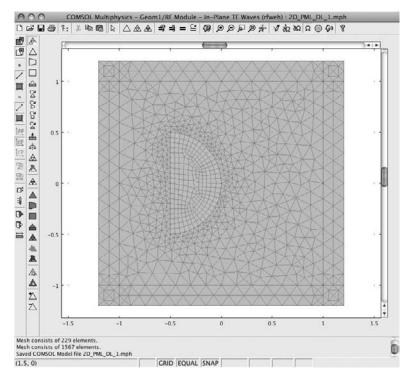
#### Solving the 2D\_PML\_DL\_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" in the Solver list. Enter lambda0\_rfweh in the Parameter name edit window. Enter linspace(0.5,1.5,11) in the Parameter values edit window. (For later versions of the COMSOL Multiphysics software, enter range( $0.5,\frac{1}{10},1.5$ ) in the Parameter values edit window.) See Figure 9.30.

Click OK. Using the menu bar, select Solve > Solve Problem.

	Free Mesh Parameters	
Global St	ubdomain Boundary Point Advanced	ОК
Subdomain selection	Subdomain mesh parameters Maximum element size: 0.05 Element growth rate: Method: Quad	Cane Appl

**FIGURE 9.28** 2D\_PML\_DL\_1 model subdomain Free Mesh Parameters



**FIGURE 9.29** 2D\_PML\_DL\_1 model mesh

Analysis:	General Par	ametric Stationary	Adaptive Advanced
Harmonic propagation		ametric Stationary	Adaptive Advanced
Auto select solver	Parameter Parameter name:	lambda0_	
Stationary	Parameter values:	linspace(0	.5,1.5,11)
Time dependent Eigenvalue	Linear system solver		
Parametric Stationary segregated	Linear system solver:	Direct (UMFPACK)	\$
Parametric segregated	Preconditioner:	[	¢ ]
	Matrix symmetry:	Automatic	¢
		Help Apply	Cancel

**FIGURE 9.30** 2D\_PML\_DL\_1 model Solver Parameters edit window

#### **Postprocessing and Visualization**

The default plot shows a surface plot of the scattered electric field, *z*-component (V/m). See Figure 9.31.

An alternative approach to viewing the effect of the dielectric lens on the plane wave within the modeling domain is to suppress the plot within the PMLs and visualize the electric field, *z*-component. Using the menu bar, select Options > Suppress > Suppress Subdomains. Select subdomains 1-4, 6, and 8-10. Click the Apply button. See Figure 9.32. Click OK.

Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select "Electric field, z component" from the Predefined quantities pull-down list. See Figure 9.33.

689

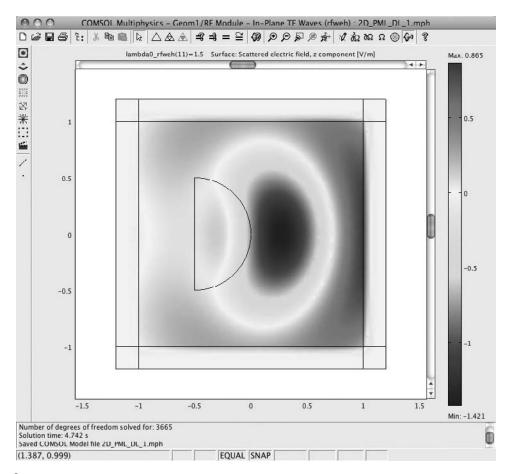


FIGURE 9.31 2D\_PML\_DL\_1 model solution, scattered electric field, z-component (V/m)

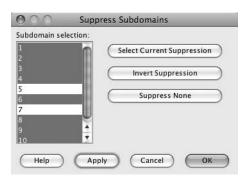


FIGURE 9.32 2D\_PML\_DL\_1 model Suppress Subdomains

0	Plot I	Parameters	_	
General Surface	Contour	Boundary	Arrow	Principal
Surface plot				
	Surface Data	Height Data		
Predefined quantities:	Electric field,	z component	CRai	nge
Expression:	Ez		🗹 Sr	nooth
Unit:	V/m		\$	

FIGURE 9.33 2D\_PML\_DL\_1 model postprocessing Plot Parameters, Surface tab

Click OK. See Figure 9.34.

Using the menu bar, select Postprocessing > Plot Parameters > Animate. Verify that all the Solutions to use are selected. See Figure 9.35. Click the Start Animation button.

#### 2D Dielectric Lens Model, with PMLs: Summary and Conclusions

The 2D dielectric lens model, with PMLs (2D\_PML\_DL\_1 model), has been built and solved. This model employs PMLs and a dielectric lens to explore the geometric behavior of transverse electric field RF waves in the presence of a focusing element. It can easily be observed by watching the animation that the position and

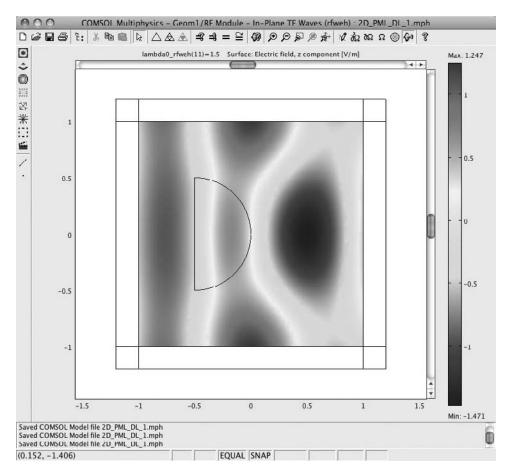


FIGURE 9.34 2D\_PML\_DL\_1 model electric field, z-component

intensity of the electric field, *z*-component varies greatly as a function of the free space wavelength.

## 2D Dielectric Lens Model, without PMLs

The following numerical solution model (2D\_NoPML\_DL\_1 model) is derived from the immediately preceding model in this chapter (2D\_PML\_DL\_1). The purpose in building this model is to empirically demonstrate the differences that are seen when PMLs are not employed.

To start building the 2D\_NoPML\_DL\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select RF Module > In-Plane Waves > TE Waves > Scattered harmonic propagation. See Figure 9.36. Click OK.

Streamline	Particle Tracing	Max/Min	Deform	Animate
Movie settings		-Solutions t	o use	
File type:	AVI	Select via:		0
Width (in pixels)	640	0.5		-
Height (in pixels):	480	0.6 0.7		
Frames per second:	10	8.0 0.9		
	( Advanced )	1		
Static / Eigenfunctio	n animation	1.2		Ę
Cycle type:	Full harmonic	1.4		۲
Number of frames:	11	Times:		
Reverse direction           Use camera setti	ngs from main window			
			Start	Animation

**FIGURE 9.35** 2D\_PML\_DL\_1 model Plot Parameters, Animate tab

0.0	Model Navigat	lor
Eigent	dule ss onic propagation frequency analysis lent propagation red harmonic propagation	A Open Settings
Dependent variables: Application mode name: Element:	scEz rfweh Lagrange - Quadra	Multiphysics

FIGURE 9.36 2D\_NoPML\_DL\_1 Model Navigator setup

000	Sav	e As
	Save As: 2D_NoPML_DI	1.mph
	11_Models	\$
Name	*	Date Modified
2D_PML_DL	1.mph	Wednesday, April 29, 2009 2:32 PM
	File Format: COMSOL	Multiphysics Mo
New Folder	D	Cancel Save

FIGURE 9.37 2D\_NoPML\_DL\_1 model Save As edit window

**NOTE** The Model Navigator command sequence (In-Plane Waves > TE Waves > Scattered harmonic propagation) selects a transverse electric field (*z*-direction) wave traveling in the plane (x, y-plane) of the modeling domain.

Select File > Save As. Enter 2D\_NoPML\_DL\_1.mph in the Save As edit window. See Figure 9.37. Click the Save button.

#### **Geometry Modeling**

Using the menu bar, select Draw > Specify Objects > Circle. Enter Radius = 0.5, Base = Center, x = -0.5, and y = 0. See Figure 9.38. Click OK.

Size		Rotation angle
Radius:	0.5	α: 0 (degree
Position		
Base:	Center 🛟	Style: Solid
x:	-0.5	Name: C1
y:	0	1

FIGURE 9.38 2D\_NoPML\_DL\_1 model Circle (C1) edit window

Size		Rotat	ion angle-	
Width:	0.5	α:	0	(degrees)
Height:	1.0			
Position				
Base:	Corner	\$ Style:	Solid	¢
x:	-1.0	Name:	R1	
у:	-0.5			

**FIGURE 9.39** 2D\_NoPML\_DL\_1 model Rectangle (R1) edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.5, Height = 1.0, Base = Corner, x = -1.0, and y = -0.5. See Figure 9.39. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter C1-R1 in the Set formula edit window. See Figure 9.40.

Click OK. See Figure 9.41.

Using the menu bar, select Draw > Specify Objects > Square. Enter Width = 2, Base = Center, x = 0, and y = 0. See Figure 9.42.

Click OK, and then click the Zoom Extents button. See Figure 9.43.

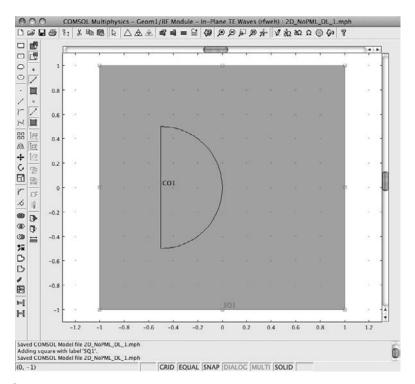
Having established the geometry for the 2D\_NoPML\_DL\_1 model, the next step is to define the fundamental Physics properties.

000 c	reate Composite Object
Solids     Curves     Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
R1	Keep interior boundaries Repair Repair tolerance: 1.0E-4

FIGURE 9.40 2D\_NoPML\_DL\_1 model Create Composite Object edit window

ا ت 1	36	E:	¥ 🖻 🛛	©    k   Z		ची ची ः	= =   0	9 9 9		ae=   w as	2 025 25	0 V1 1	۶ 	
	1	5					, (	)						4 1
•														- 1
	0.8	ŀ			8		8	35	1	19		æ	÷	
• /	0.6	ŀ		52				21	3	•	•		t)	
國國	0.4	-			ii.	+		13. 2			(4)	×		-
回見	0.2	-						\.		*	•		•	
	0	ŀ	1	÷		COļ			5		•	÷		
	-0.2	-	19		1.7	*		/.	3	16	( <b>4</b> )		72	
÷	-0.4	ŀ	÷					÷	•		+		2	-
	-0.6	ŀ					8	8	2		•	×		-
	-0.8	ł				7.	đ	•	÷	•		•	•	
	-1										-			
			-1	-0.8	-0.6	-0.4	-0.2	0	0.2	0.4	0.6	0.8	1	
ed CO	ircle wit MSOL N	lodel	el 'C1'. file 2D_No label 'K1'	PML_DL_1	.mph									

**FIGURE 9.41** 2D\_NoPML\_DL\_1 model dielectric lens (C01)



**FIGURE 9.43** 2D\_NoPML\_DL\_1 model domain

#### Physics Application Mode Properties: In-Plane TE Waves (rfweh)

Select Physics > Properties. Select "Free space wavelength" from the Specify wave using pull-down list. See Figure 9.44. Click OK.

Default element type	Lagrange - Quadratic	
	(	
Analysis type:	Harmonic propagation	101
Field type:	Scattered TE waves	\$
Specify wave using:	Free space wavelength	\$
Specify elgenvalues using:	Figenfrequency	Å.
Divergence condition:	oll	\$
Weak constraints:	0ff	\$
Vector element constraint:	Off	;
Constraint type:	Ideal	\$

FIGURE 9.44 2D\_NoPML\_DL\_1 model Application Mode Properties edit window

Applicatio	n Scal	ar Variables
Expression	Unit	Description
8.854187817e-12	F/m	Permittivity of vacuum
	H/m	Permeability of vacuum
0.5	m	Free space wavelength
exp(-j*k0_rfweh*x)	V/m	Incident electric field, z component
	8.854187817e-12 4*pi*1e-7 0.5	8.854187817e-12 F/m 4*pi*1e-7 H/m 0.5 <b>m</b>

**FIGURE 9.45** 2D\_NoPML\_DL\_1 model Application Scalar Variables edit window (lambda0\_rfweh)

#### Physics Application Scalar Variables: In-Plane TE Waves (rfweh)

Select Physics > Scalar Variables. Enter 0.5 in the lambda0\_rfweh edit window. See Figure 9.45. Click OK.

#### Physics Subdomain Settings: In-Plane TE Waves (rfweh)

Having established the basic Physics settings for the 2D\_NoPML\_DL\_1 model, the next step is to define the fundamental Physics subdomain settings. Select Physics > Subdomain Settings. Select subdomain 1 (the model domain). Enter  $\varepsilon_r = 1, \sigma = 0$ , and  $\mu_r = 1$ . Click the Apply button. See Figure 9.46.

iquation 7×( $\mu_r^{-1}$ ∇×E <sub>x</sub> ) – (ε <sub>r</sub> – jσ/ωε <sub>0</sub> )k	Subdomain Settings – In-Plane TE Waves (rfweh) $_{0}{}^{2}E_{x} = 0, \epsilon_{r} = n^{2}$	
Subdomains Groups Subdomain selection	Physics PML Init Elem Material properties Library material:	nent Color
2	Quantity Value/Expression Specify material properties in terms of refractive n I Specify material properties in terms of $\epsilon_i$ , $\mu_i$ , and	1 Refractive index
Group: 🔅	<sup>ε</sup> <sub>r</sub> 1 σ 0 μ <sub>r</sub> 1	1     Relative permittivity       S/m     Electric conductivity       1     Relative permeability

FIGURE 9.46 2D\_NoPML\_DL\_1 model Subdomain Settings, subdomain 1

$\times (\mu_r^{-1} \nabla \times E_z) - (\epsilon_r - j\sigma/\omega\epsilon_0) k_0^2$	$E_z = 0, \epsilon_r = n^2$	
Subdomains Groups	Physics PML Init	Element Color
Subdomain selection	Material properties Library material: Quantity Value/Expression Specify material properties in terms of refrac	Unit Description
Group: () Select by group	$ \begin{array}{c} \bullet & \downarrow \\ \bullet & \text{Specify material properties in terms of } \epsilon_r, \mu_r, \\ & \Gamma & \sigma \\ \sigma & \sigma \\ & \sigma \\ & \mu_r & 1 \end{array} $	Kerractive index

**FIGURE 9.47** 2D\_NoPML\_DL\_1 model Subdomain Settings, subdomain 2

Select subdomain 2 (the dielectric lens). Enter  $\varepsilon_r = 3$ ,  $\sigma = 0$ , and  $\mu_r = 1$ . Click the Apply button. See Figure 9.47. Click OK.

#### Physics Boundary Settings: In-Plane TE Waves (rfweh)

Having established the subdomain settings for the 2D\_NoPML\_DL\_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, select Physics > Boundary Settings. Select boundaries 1, 2, 3, and 5 (the outer edges of the model domain). Select "Scattering boundary condition" from the Boundary condition pull-down list. See Figure 9.48. Click OK.

#### **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 2 (the dielectric lens). Enter 0.05 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 9.49.

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 9.50.

#### Solving the 2D\_NoPML\_DL\_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" in the Solver list. Enter lambda0\_rfweh in the Parameter name edit window. Enter linspace(0.5,1.5,11) in the Parameter values edit window. (For later versions of the COMSOL Multiphysics software enter range( $0.5,\frac{1}{10},1.5$ ) in the Parameter values edit window.) See Figure 9.51.

**FIGURE 9.48** 2D\_NoPML\_DL\_1 model Boundary Settings

0.0	Free Mesh Parameters	
Global	Subdomain Boundary Point Advanced	
Subdomain selection	Subdomain mesh parameters Maximum element size: 0.05 Element growth rate: Method: Quad \$	(Car (Ap (He
Select Meshed	Remesh Mesh Selected	

**FIGURE 9.49** 2D\_NoPML\_DL\_1 model subdomain Free Mesh Parameters

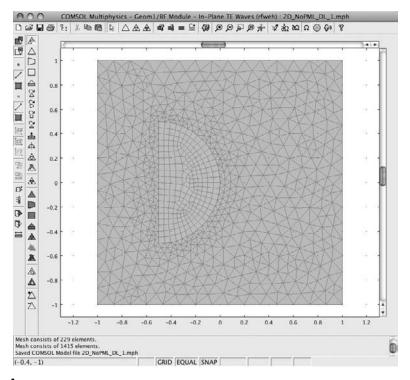


FIGURE 9.50 2D\_NoPML\_DL\_1 model mesh

900	Solver Par	rameters			
Analysis: Harmonic propagation	General Pa	rametric St	tationary	Adaptive	Advanced
Auto select solver Solver: Stationary	Parameter Parameter name: Parameter values:		lambda0_i	rfweh 0.5,1.5,11)	
Time dependent Eigenvalue Rarametric Stationary segregated Parametric segregated	Linear system solver Linear system solver: Preconditioner:	Direct (UMF	PACK)	\$	
Adaptive mesh refinement	Matrix symmetry:	Automatic			Settings
		Help			ancel) (Of

FIGURE 9.51 2D\_NoPML\_DL\_1 model Solver Parameters edit window

701

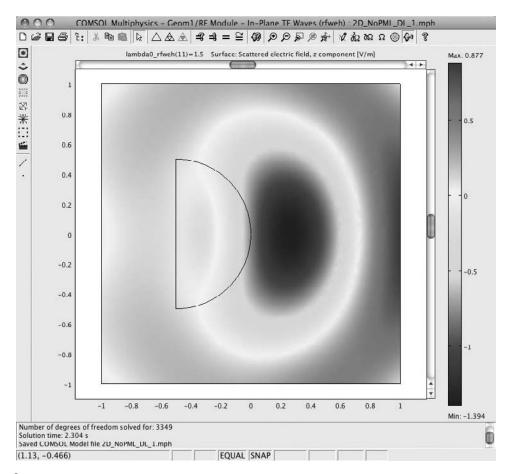


FIGURE 9.52 2D\_NoPML\_DL\_1 model solution, scattered electric field, z-component (V/m)

Click OK. Using the menu bar, select Solve > Solve Problem.

#### Postprocessing and Visualization

The default plot shows a surface plot of the scattered electric field, *z*-component (V/m). See Figure 9.52.

An alternative approach to viewing the effect of the dielectric lens on the plane wave within the modeling domain is to view the electric field, *z*-component. Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select "Electric field, z component" from the Predefined quantities pull-down list. See Figure 9.53.

0	Plot F	Parameters		_	
General Surface	Contour	Boundary	Arrow	Principal	•
Surface plot					
	Surface Data	Height Data			
Predefined quantities:	Electric field	z component	Ran		
		2 component			
Expression:	Ez		Sm	ooth	
Unit:	V/m		\$		
Coloring and fill					
Coloring: Interpo	lated	Fill style:	Filled	\$	
Colormap:	jet 🛟	Colors: 1024	Col	or scale	

FIGURE 9.53 2D\_NoPML\_DL\_1 model postprocessing Plot Parameters, Surface tab

Click OK. See Figure 9.54.

Using the menu bar, select Postprocessing > Plot Parameters > Animate. Verify that all the Solutions to use are selected. See Figure 9.55. Click the Start Animation button.

# 2D Dielectric Lens Model, with and without PMLs: Summary and Conclusions

The 2D dielectric lens models, with and without PMLs (2D\_PML\_DL\_1 and 2D\_NoPML\_DL\_1, respectively) have been built and solved. The best method of comparison between the two models is to view visualizations for the electric field,

703

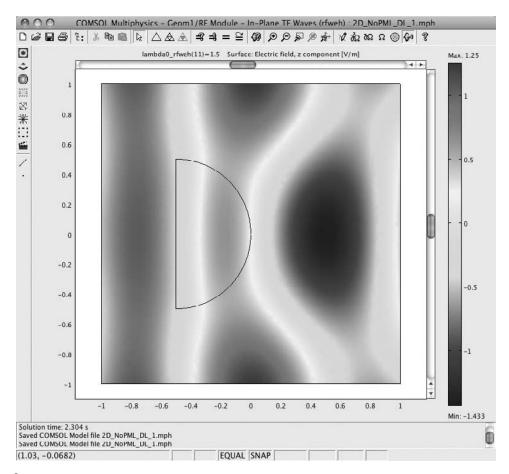


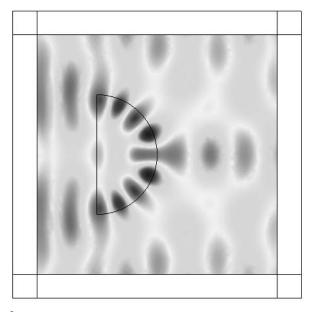
FIGURE 9.54 2D\_NoPML\_DL\_1 model electric field, z-component

*z*-component for the same wavelength from each model together. Figures 9.56 through 9.61 show visualizations for 0.5 m (Figures 9.56 and 9.57), 1.0 m (Figures 9.58 and 9.59), and 1.5 m (Figures 9.60 and 9.61).

The differences in the electric field, *z*-component visualizations between the PML and no-PML models amount to approximately 2%. Depending on the nature of the problem, such differences may or may not be significant. What these differences show the modeler is that he or she needs to understand the application environment well to build the best model. The PML model best approximates a free space environment (no reflections). For other environments, the modeler needs to determine the best boundary condition approximation using standard practices and a first principles approach. When all else fails (or even before), do a first principles analysis of the environment before building the model.

Particle Tracing Avi 640 640 10 Advanced	Max/Min Solutions to Select via 0.5 0.5 0.7 0.8 0.9 1 1.1	Deform use	Animate
640 480 10 (Advanced)	Select via 0.5 0.6 0.7 0.8 0.9 1	use	:
640 480 10 (Advanced)	0.5 0.6 0.7 0.8 0.9 1		:
480 10 ( Advanced )	0.6 0.7 0.8 0.9 1		
10 (Advanced)	0.7 0.8 0.9 1		
(Advanced)	0.9 1		
	1		
n animation	1.2		ç
Full harmonic 😫	1.4	_	۲
11	Times:		
		Start	Animation
	11 igs from main window	11 gs from main window	gs from main window

**FIGURE 9.55** 2D\_NoPML\_DL\_1 model Plot Parameters, Animate tab



I FIGURE 9.56 2D\_PML\_DL\_1 model plot electric field, z-component, 0.5 m

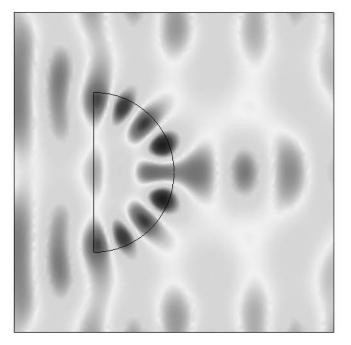
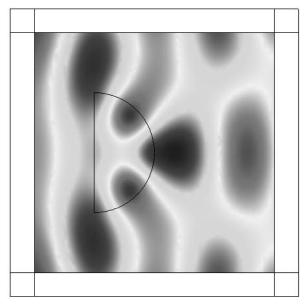


FIGURE 9.57 2D\_NoPML\_DL\_1 model plot electric field, z-component, 0.5 m



**FIGURE 9.58** 2D\_PML\_DL\_1 model plot electric field, *z*-component, 1.0 m

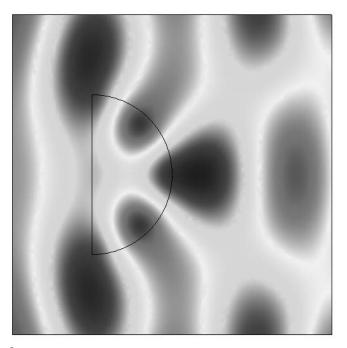
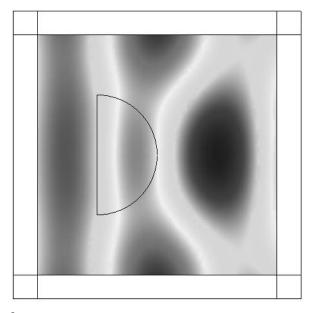


FIGURE 9.59 2D\_NoPML\_DL\_1 model plot electric field, z-component, 1.0 m



**FIGURE 9.60** 2D\_PML\_DL\_1 model plot electric field, *z*-component, 1.5 m

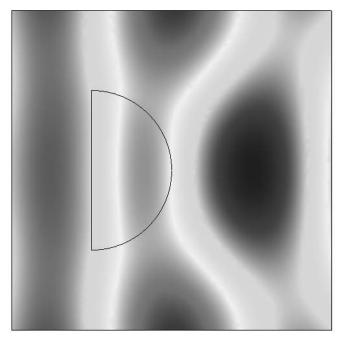


FIGURE 9.61 2D\_NoPML\_DL\_1 model plot electric field, z-component, 1.5 m

# 2D Concave Mirror Model, with PMLs

The following numerical solution model (2D\_PML\_CM\_1 model) is derived from the preceding dielectric lens model (2D\_PML\_DL\_1 model). In this case, however, the electromagnetic waves interact with a fixed, curved metallic mirror. The purpose of this model (2D\_PML\_CM\_1) and the following model (2D\_NoPML\_CM\_1) is to demonstrate empirically the difference between having or not having PMLs at the model boundaries.

To start building the 2D\_PML\_CM\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select RF Module > In-Plane Waves > TE Waves > Scattered harmonic propagation. See Figure 9.62. Click OK.

**NOTE** The Model Navigator command sequence (In-Plane Waves > TE Waves > Scattered harmonic propagation) selects a transverse electric field (*z*-direction) wave traveling in the plane (x, y-plane) of the modeling domain.

# **Geometry Modeling**

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 9.2. Click OK after filling in the parameters

Name	Width	Height	Base	Х	Y	Figure Number
R1	2.4	0.2	Corner	-1.2	1.0	9.63
R2	2.4	0.2	Corner	-1.2	-1.2	9.64
R3	0.2	2.4	Corner	-1.2	-1.2	9.65
R3	0.2	2.4	Corner	1.0	-1.2	9.66

Table 9.2	Geometry	Components
-----------	----------	------------

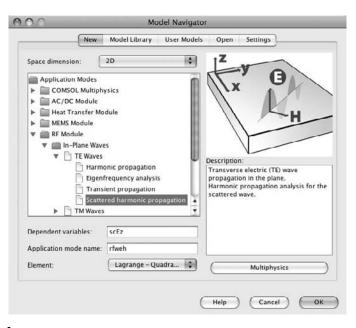


FIGURE 9.62 2D\_PML\_CM\_1 Model Navigator setup

Size		Rotat	ion angle-	
Width	2.4	α.	0	(degrees)
Height:	0.2			
Position				
Base:	Corner	\$ Style:	Solid	
х:	-1.2	Name:	R1	
y:	1.0			

FIGURE 9.63 2D\_PML\_CM\_1 model Rectangle (R1) edit window

e :		Rectangle		_
Size		Rota	tion angle	
Width:	2.4	α:	0	(degrees)
Height:	0.2			
Position				
Base:	Corner	\$ Style:	Solid	¢
x:	-1.2	Name:	R2	
y:	-1.2			

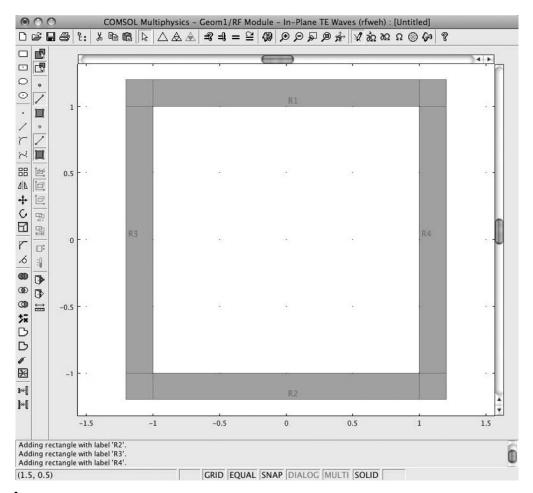
FIGURE 9.64 2D\_PML\_CM\_1 model Rectangle (R2) edit window

Size		Rotation angle
Width:	0.2	α: 0 (degrees)
Height:	2.4	
Position		
Base:	Corner	Style: Solid
x:	-1.2	Name: R3
y:	-1.2	

FIGURE 9.65 2D\_PML\_CM\_1 model Rectangle (R3) edit window

Size		Rotat	ion angle-	
Width:	0.2	α:	0	(degrees)
Height:	2.4			
Position				
Base:	Corner	Style:	Solid	\$
k:	1.0	Name:	R4	
ŗ:	-1.2			

I FIGURE 9.66 2D\_PML\_CM\_1 model Rectangle (R4) edit window



**FIGURE 9.67** 2D\_PML\_CM\_1 model PML rectangles

of each separate rectangle in the Rectangle edit window. See Figures 9.63–9.66. Click the Zoom Extents button before drawing the next rectangle. Figure 9.67 shows the PML rectangles of model 2D\_PML\_CM\_1.

Select File > Save As. Enter 2D\_PML\_CM\_1.mph in the Save As edit window. See Figure 9.68. Click the Save button.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes = 0.3, B-semiaxes = 0.5, Base = Center, x = 0, and y = 0. See Figure 9.69. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes = 0.29, B-semiaxes = 0.49, Base = Center, x = 0, and y = 0. See Figure 9.70. Click OK.

11_Models	\$
Name	Date Modified
2D_NOPML_DL_1.mph 2D_PML_DL_1.mph	Wednesday, April 29, 2009 6:19 PM Wednesday, April 29, 2009 2:32 PM
File Format: COMSC	DL Multiphysics Mo

**FIGURE 9.68** 2D\_PML\_CM\_1 model Save As edit window

Size		Rotati	on angle	
A-semiaxes:	0.3	α:	0	(degrees)
B-semiaxes:	0.5			
Position				
Base:	Center	\$ Style:	Solid	\$
x:	0	Name:	E1	
y:	0			

I FIGURE 9.69 2D\_PML\_CM\_1 model Ellipse (E1) edit window

Size		Rotation angle
A-semiaxes:	0.29	α: 0 (degree
B-semiaxes:	0.49	1
Position		
Base:	Center ‡	Style: Solid
к:	0	Name: E2
y:	0	1

I FIGURE 9.70 2D\_PML\_CM\_1 model Ellipse (E2) edit window

Size		Rotat	ion angle	
Width:	0.5	α:	0	(degrees
Height:	1.0			
Position				
Base:	Corner	\$ Style:	Solid	\$
x:	-0.5	Name:	R5	
	-0.5			

FIGURE 9.71 2D\_PML\_CM\_1 model Rectangle (R5) edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.5, Height = 1.0, Base = Corner, x = -0.5, and y = -0.5. See Figure 9.71. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter E1-E2-R5 in the Set formula edit window. See Figure 9.72.

Click OK. See Figure 9.73.

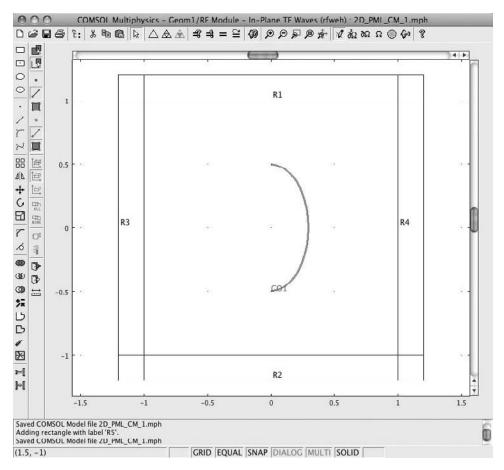
Using the menu bar, select Draw > Specify Objects > Square. Enter Width = 2.0, Base = Center, x = 0, and y = 0. See Figure 9.74. Click OK.

Using the menu bar, select Edit > Select All. See Figure 9.75.

Having established the geometry for the 2D\_PML\_CM\_1 model, the next step is to define the fundamental Physics properties.

000 0	Create Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
R1	E1-E2-R5
R2 R3 R4 E1 E2 R5	Keep interior boundaries Repair Repair 1.0E-4

**FIGURE 9.72** 2D\_PML\_CM\_1 model Create Composite Object edit window



**FIGURE 9.73** 2D\_PML\_CM\_1 model concave mirror (CO1)

Size		Rotat	ion angle	
Width:	2	α:	0	(degrees)
Positio	n			
Base:	Center	Style:	Solid	\$
c	0	Name:	SQ1	
/:	0			

FIGURE 9.74 2D\_PML\_CM\_1 model Square (SQ1) edit window

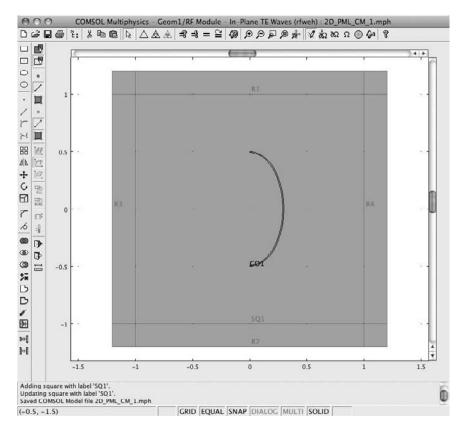


FIGURE 9.75 2D\_PML\_CM\_1 model domain plus PMLs

# Physics Application Mode Properties: In-Plane TE Waves (rfweh)

Select Physics > Properties. Select "Free space wavelength" from the Specify wave using pull-down list. See Figure 9.76. Click OK.

# Physics Application Scalar Variables: In-Plane TE Waves (rfweh)

Select Physics > Scalar Variables. Enter 0.5 in the lambda0\_rfweh edit window. See Figure 9.77. Click OK.

# Physics Subdomain Settings: In-Plane TE Waves (rfweh)

Having established the basic Physics settings for the 2D\_PML\_CM\_1 model, the next step is to define the fundamental Physics subdomain settings. Select Physics > Subdomain Settings. Click the PML tab. Select subdomains 1–4, 6, and 8–10 (the PMLs). Select "Cartesian" from the Type of PML pull-down list. Click the Apply button. See Figure 9.78.

I FIGURE 9.76 2D\_PML\_CM\_1 model Application Mode Properties edit window

Name	Expression	Unit	Description
epsilon0_rfweh mu0_rfweh	8.854187817e-12 4*pi*1e-7	F/m H/m	Permittivity of vacuum Permeability of vacuum
lambda0_rfweh	0.5	m	Free space wavelength
E0iz_rfweh	exp(-j*k0_rfweh*x)	V/m	Incident electric field, z component

FIGURE 9.77 2D\_PML\_CM\_1 model Application Scalar Variables (lambda0\_rfweh) edit window

Subdomains Groups	Physics	PML I	nit Elemen	t Color
ubdomain selection	Type of PML: Absorbing in x direction Absorbing in y direction x <sub>0</sub> , y <sub>0</sub>	Cartesia Value/Expl Sdx_gues Sdy_gues	ression s_rfweb	Description <sup>m</sup> Width in x direction <sup>m</sup> Width in y direction <sup>m</sup> Center point
Froup: 2000 Constraints and a constraint of the	Ī.			

**FIGURE 9.78** 2D\_PML\_CM\_1 model Subdomain Settings, PML type selection

Subdomains Groups	)(	Physics PML	Init Elemen	t Color
Subdomain selection	Type of PML:	Carte		
4 5 6	Absorbing in x	direction Sdx_gu	xpression uess_rfweh uess_rfweh	Description <sup>m</sup> Width in x direction <sup>m</sup> Width in y direction
7 8 9 10	× <sub>0</sub> . y <sub>0</sub>	o	0	<sup>m</sup> Center point
Group:	실제 제			
Select by group				

**FIGURE 9.79** 2D\_PML\_CM\_1 model Subdomain Settings, *x* absorption

Select subdomains 1–3, and 8–10 (the vertical PMLs). Check the Absorbing in x direction check box. Click the Apply button. See Figure 9.79.

Select subdomains 1, 3, 4, 6, 8, and 10 (the horizontal PMLs). Check the Absorbing in y direction check box. Click the Apply button. See Figure 9.80.

Subdomains Groups	Physics	PML Init Element	Color
Subdomain selection	Type of PML: ✓ Absorbing in x direction ✓ Absorbing in y direction x <sub>0</sub> , y <sub>0</sub>	Jux_guess_nwen	Description <sup>m</sup> Width in x direction <sup>m</sup> Width in y direction <sup>m</sup> Center point

**FIGURE 9.80** 2D\_PML\_CM\_1 model Subdomain Settings, *y* absorption

$\times (\mu_r^{-1} \nabla \times E_x) - (\epsilon_r - j\sigma/\omega\epsilon_0) k_0^2$	$E_z = 0, \epsilon_r = n^2$	
Subdomains Groups	Physics PML Init	Element Color
Subdomain selection	Material properties Library material: Library material: Loa Quantity Value/Expression Specify material properties in terms of refrain n i Specify material properties in terms of $\mathbf{r}_r, \boldsymbol{\mu}_r$	Unit Description ctive index 1 Refractive index
Group: 2 Select by group	ε, 1 σ 0 μ <sub>r</sub> 1	1         Relative permittivity           S/m         Electric conductivity           1         Relative permeability

FIGURE 9.81 2D\_PML\_CM\_1 model Subdomain Settings, Physics tab, subdomain 5

Click the Physics tab. Select subdomain 5 (the model domain). Enter  $\varepsilon_r = 1, \sigma = 0$ , and  $\mu_r = 1$ . Click the Apply button. See Figure 9.81.

Select subdomain 7 (the concave mirror). Click the Load button. Select Basic Material Properties > Copper. See Figure 9.82. Click OK.

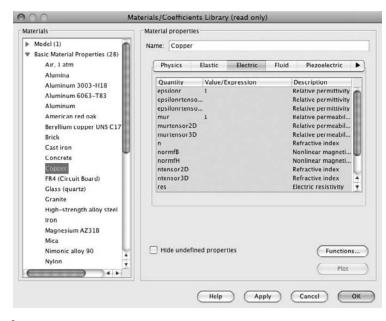


FIGURE 9.82 2D\_PML\_CM\_1 model Materials/Coefficients Library, copper

FIGURE 9.83 2D\_PML\_CM\_1 model Subdomain Settings, Physics tab, subdomain 7

See Figure 9.83. Click OK.

#### **Physics Boundary Settings: In-Plane TE Waves (rfweh)**

Having established the subdomain settings for the 2D\_PML\_CM\_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, select Physics > Boundary Settings. Select boundary 1. Check the Select by Group check box to select the outer edge of the PMLs (boundaries). Select "Scattering boundary condition" from the Boundary condition pull-down list. See Figure 9.84. Click OK.

#### **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 7 (the concave mirror). Enter 0.05 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 9.85.

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 9.86.

#### Solving the 2D\_PML\_CM\_1 Model

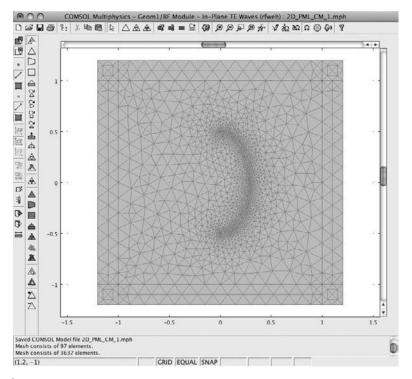
Using the menu bar, select Solve > Solver Parameters. Select "Parametric" in the Solver list. Enter lambda0\_rfweh in the Parameter name edit window. Enter linspace(0.5, 1.5, 11) in the Parameter values edit window. (For later versions of the COMSOL Multiphysics software enter range(0.5, 1/10, 1.5) in the Parameter values edit window.) See Figure 9.87.

Click OK. Using the menu bar, select Solve > Solve Problem.

I FIGURE 9.84 2D\_PML\_CM\_1 model Boundary Settings

2.02	Free Mesh Parameters	
Global	Subdomain Boundary Point Advanced	
Subdomain selection	Subdomain mesh parameters Maximum element size: 0.05 Element growth rate: Method: Quad \$	Car Ap He
Select Meshed	Remesh Mesh Selected	

**FIGURE 9.85** 2D\_PML\_CM\_1 model subdomain Free Mesh Parameters





0.0	Solver Par	ameters		
nalysis Harmonic propagation	General Par	rametric Stationar	<b>y</b> Adaptive	Advanced
Auto select solver	Parameter Parameter name: Parameter values:		da0_rfweh ace(0.5,1.5,11)	
Time dependent igenvalue Parametric Itationary segregated Parametric segregated	Linear system solver Linear system solver: Preconditioner:	Direct (UMFPACK)	¢ ;	
Adaptive mesh refinement	Matrix symmetry:	Automatic	\$	Settings
		(Help)	Apply ) ( C	ancel ) ( UK

FIGURE 9.87 2D\_PML\_CM\_1 model Solver Parameters

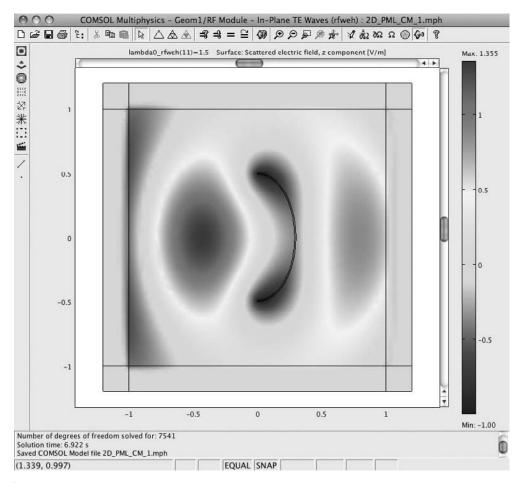


FIGURE 9.88 2D\_PML\_CM\_1 model solution, scattered electric field, z-component (V/m)

### Postprocessing and Visualization

The default plot shows a surface plot of the scattered electric field, *z*-component (V/m). See Figure 9.88.

An alternative approach to viewing the effect of the dielectric lens on the plane wave within the modeling domain is to suppress the plot within the PMLs and visualize the electric field, *z*-component. Using the menu bar, select Options > Suppress > Suppress Subdomains. Select subdomains 1–4, 6, and 8–10 (the PMLs). Click the Apply button. See Figure 9.89. Click OK.

Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select "Electric field, z component" from the Predefined quantities pull-down list. See Figure 9.90.

Subdomain sel	Suppress Subdomains
1	Select Current Suppression
	Invert Suppression
5 6	Suppress None
7 8	U.
9 10	<b>T</b>
Help	Apply Cancel OK

**FIGURE 9.89** 2D\_PML\_CM\_1 model Suppress Subdomains

General Surface	Contour Boundary Arrow Principal
Surface plot	
	Surface Data Height Data
Predefined quantities:	
Expression:	Ez Smooth
Unit:	V/m 🗘
oloring and fill	
oloring: Interpo	olated 🗘 Fill style: Filled 🛟
Uniform color:	Color,

**FIGURE 9.90** 2D\_PML\_CM\_1 model Plot Parameters, Surface tab

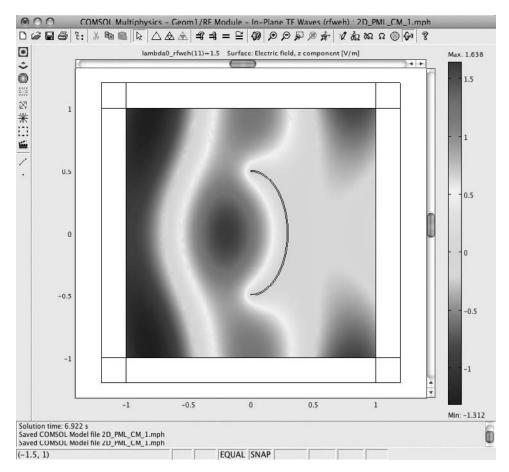


FIGURE 9.91 2D\_PML\_CM\_1 model electric field, z-component

Click OK. See Figure 9.91.

Using the menu bar, select Postprocessing > Plot Parameters > Animate. Verify that all the Solutions to use are selected. See Figure 9.92. Click the Start Animation button.

# 2D Concave Mirror Model, with PMLs: Summary and Conclusions

The 2D concave mirror model, with PMLs (2D\_PML\_CM\_1), has been built and solved. This model employs PMLs and a concave mirror to explore the geometric behavior of transverse electric field RF waves in the presence of a metallic focusing element (concave mirror). It can easily be observed by watching the animation that the position and intensity of the electric field, *z*-component varies greatly as a function of the free space wavelength.

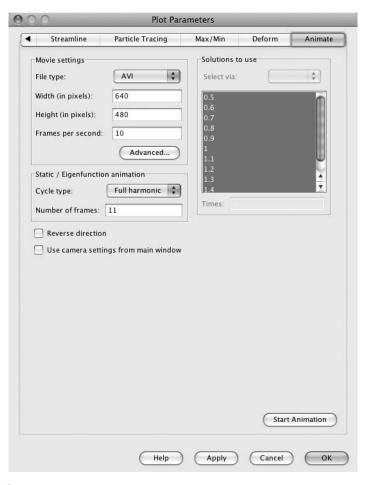


FIGURE 9.92 2D\_PML\_CM\_1 model Plot Parameters, Animate tab

# 2D Concave Mirror Model, without PMLs

The following numerical solution model (2D\_NoPML\_CM\_1 model) is derived from the preceding concave mirror model (2D\_PML\_CM\_1 model). In this case, however, the electromagnetic waves interact with a fixed, curved metallic mirror without PMLs at the boundaries of the modeling domain. The purpose of this model (2D\_NoPML\_CM\_1) is to demonstrate empirically the difference between having or not having PMLs at the model boundaries.

To start building the 2D\_NoPML\_CM\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "2D" from the Space dimension pull-down list. Select RF Module > In-Plane Waves > TE Waves > Scattered harmonic propagation. See Figure 9.93. Click OK.

00	Model Navigator	
New	Model Library User Models Open Settings	
Eigenf	dule as ponic propagation frequency analysis ient propagation red harmonic propagation ter harmonic propagation	is for the
Dependent variables: Application mode name:	scEz rfweh	
Element:	Lagrange – Quadra	

FIGURE 9.93 2D\_NoPML\_CM\_1 Model Navigator setup

Note The Model Navigator command sequence (In-Plane Waves > TE Waves > Scattered harmonic propagation) selects a transverse electric field (*z*-direction) wave traveling in the plane (x, y-plane) of the modeling domain.

Select File > Save As. Enter 2D\_NoPML\_CM\_1.mph in the Save As edit window. See Figure 9.94. Click the Save button.

# **Geometry Modeling**

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes = 0.3, B-semiaxes = 0.5, Base = Center, x = 0, and y = 0. See Figure 9.95. Click OK.

Using the menu bar, select Draw > Specify Objects > Ellipse. Enter A-semiaxes = 0.29, B-semiaxes = 0.49, Base = Center, x = 0, and y = 0. See Figure 9.96. Click OK.

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.5, Height = 1.0, Base = Corner, x = -0.5, and y = -0.5. See Figure 9.97. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter E1-E2-R1 in the Set formula edit window. See Figure 9.98.

11_Mode	els 🗘
Name 2D_NOPML_DL_1.mph 2D_PML_CM_1.mph 2D_PML_DL_1.mph	<ul> <li>Date Modified</li> <li>Wednesday, April 29, 2009 6:19 PM</li> <li>Thursday, April 30, 2009 5:13 PM</li> <li>Wednesday, April 29, 2009 2:32 PM</li> </ul>

**FIGURE 9.94** 2D\_NoPML\_CM\_1 model Save As edit window

Size		Rotat	ion angle	
A-semiaxes:	0.3	α:	0	(degrees
B-semiaxes:	0.5			
Position				
Base:	Center	Style:	Solid	10
х:	0	Name:	E1	
	0			

**FIGURE 9.95** 2D\_NoPML\_CM\_1 model Ellipse (E1) edit window

Size		Rotat	ion angle	
A-semiaxes:	0.29	α:	0	(degrees)
B-semiaxes:	0.49			
Position				
Base:	Center	Style:	Solid	\$
<b>x</b> :	0	Name:	E2	
	0	-		

FIGURE 9.96 2D\_NoPML\_CM\_1 model Ellipse (E2) edit window

Size		Rotation angle	
Width:	0.5	α: 0 (degree	es
Height:	1.0		
Position			
Base:	Corner	Style: Solid	÷
x:	-0.5	Name: R1	
y:	-0.5		

FIGURE 9.97 2D\_NoPML\_CM\_1 model Rectangle (R1) edit window

Click OK. See Figure 9.99.

Using the menu bar, select Draw > Specify Objects > Square. Enter Width = 2.0, Base = Center, x = 0, and y = 0. See Figure 9.100.

Click OK, and then click the Zoom Extents button. See Figure 9.101.

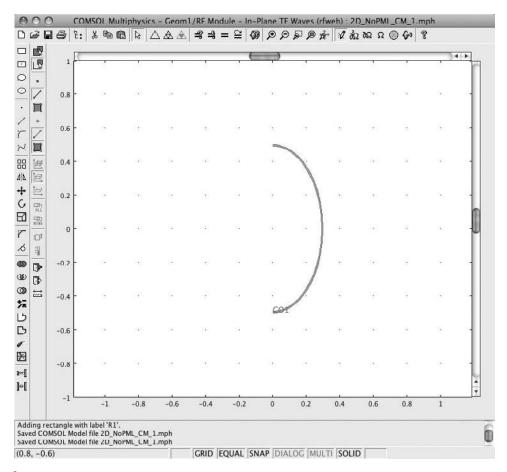
Having established the geometry for the 2D\_NoPML\_CM\_1 model, the next step is to define the fundamental Physics properties.

# Physics Application Mode Properties: In-Plane TE Waves (rfweh)

Select Physics > Properties. Select "Free space wavelength" from the Specify wave using pull-down list. See Figure 9.102. Click OK.

OOO Cre	eate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: E1 E2 R1	Set formula: E1-E2-R1 Keep interior boundaries Repair Repair tolerance: 1.0E-4

FIGURE 9.98 2D\_NoPML\_CM\_1 model Create Composite Object edit window



**FIGURE 9.99** 2D\_NoPML\_CM\_1 model concave mirror (CO1)

Size		Rotat	ion angle
Width:	2.0	α:	0 (degrees
Positio	n		
Base:	Center	Style:	Solid 🗘
x:	0	Name:	SQ1
y:	0		

**FIGURE 9.100** 2D\_NoPML\_CM\_1 model Square (SQ1) edit window

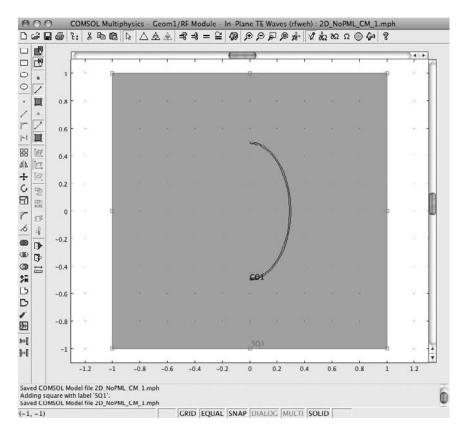


FIGURE 9.101 2D\_NoPML\_CM\_1 model domain

000	Applicatio	n Scal	lar Variables
Name	Expression	Unit	Description
epsilon0_rfweh	8.854187817e-12	F/m	Permittivity of vacuum
mu0_rfweh	4*pi*1e-7	H/m	Permeability of vacuum
lambda0_rfweh	0.5	m	Free space wavelength
E0iz_rfweh	exp(-j*k0_rfweh*x)	V/m	Incident electric field, z component
Synchronize	e equivalent variables		
	Help		pply Cancel OK

FIGURE 9.103 2D\_NoPML\_CM\_1 model Application Scalar Variables (lambda0\_rfweh) edit window

### Physics Application Scalar Variables: In-Plane TE Waves (rfweh)

Select Physics > Scalar Variables. Enter 0.5 in the lambda0\_rfweh edit window. See Figure 9.103. Click OK.

### Physics Subdomain Settings: In-Plane TE Waves (rfweh)

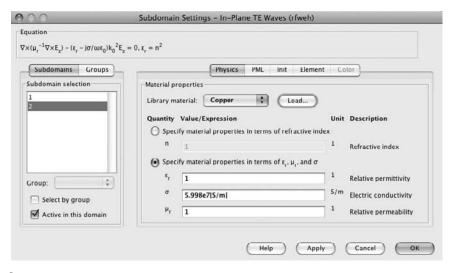
Having established the basic Physics settings for the 2D\_NoPML\_CM\_1 model, the next step is to define the fundamental Physics subdomain settings. Select Physics > Subdomain Settings. Click the Physics tab. Select subdomain 1 (the model domain). Enter  $\varepsilon_r = 1$ ,  $\sigma = 0$ , and  $\mu_r = 1$ . Click the Apply button. See Figure 9.104.

Materials	Material properties	
<ul> <li>Model (0)</li> <li>Basic Material Properties (28)</li> </ul>	Name: Copper	
Air, 1 atm	Physics Elastic Electric Fluid	Piezoelectric
Alumina		
Aluminum 3003-H18	Quantity Value/Expression epsilonrtenso	Description Relative permittivity
Aluminum 6063-T83	murtensor3D	Relative permeabil
Aluminum	normfH	Nonlinear magneti
American red oak	ntensor3D	Refractive index
Beryllium copper UNS C17	sigmatensor3D	Electric conductivity
Brick		
Cast iron		
Concrete		
Copper		
FR4 (Circuit Board)		
Glass (guartz)		
Granite		
High-strength alloy steel		
Iron		
Magnesium AZ318		
Mica		
Nimonic alloy 90	Hide undefined properties	Functions
Nylon		
		Plot

FIGURE 9.105 2D\_NoPML\_CM\_1 model Materials/Coefficients Library, copper

Select subdomain 2 (the concave mirror). Click the Load button. Select Basic Material Properties > Copper. See Figure 9.105. Click OK.

See Figure 9.106. Click OK.



**FIGURE 9.106** 2D\_NoPML\_CM\_1 model Subdomain Settings, Physics tab, subdomain 2

Boundaries Groups	Conditions Material Properties Port Far-Field Color/Style
	Boundary condition: Scattering boundary condition
5 6 7 8 9 10	Quantity Value/Expression Unit Description Wave type:
Group: 🔹 🗘	

**FIGURE 9.107** 2D\_NoPML\_CM\_1 model Boundary Settings

# Physics Boundary Settings: In-Plane TE Waves (rfweh)

Having established the subdomain settings for the 2D\_PML\_CM\_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, select Physics > Boundary Settings. Select boundaries 1, 2, 3, and 6 (the outer edges of the model domain). Select "Scattering boundary condition" from the Boundary condition pull-down list. See Figure 9.107. Click OK.

### **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 2 (the concave mirror). Enter 0.05 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 9.108.

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 9.109.

### Solving the 2D\_NoPML\_CM\_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" in the Solver list. Enter lambda0\_rfweh in the Parameter name edit window. Enter linspace(0.5, 1.5, 11) in the Parameter values edit window. (For later versions of the

	Free Mesh Para	meters	_
Global 5	Subdomain Boundary Poi	int Advanced	ОК
selection	Subdomain mesh paramet Maximum element size: Element growth rate: Method:	0.05 Quad	Canc Appl Help
by group emaining Meshed			
	selection y group emaining	Subdomain mesh paramet Maximum element size: Element growth rate. Method:	Selection Subdomain mesh parameters Maximum element size: 0.05 Element growth rate: Method: Quad

FIGURE 9.108 2D\_NoPML\_CM\_1 model subdomain Free Mesh Parameters

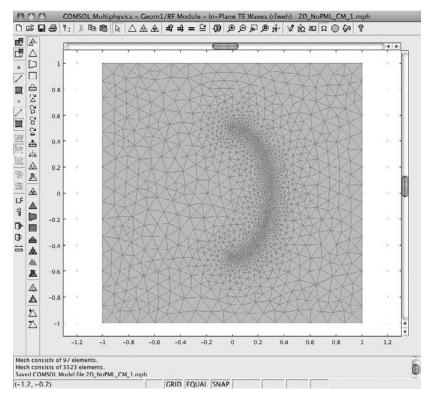


FIGURE 9.109 2D\_NoPML\_CM\_1 model mesh

Analysis:	Solver Par				
Harmonic propagation	General Par	rametric Sta	ationary A	daptive Advar	iced
Auto select solver olver: Stationary Time dependent Eigenvalue Parametric	Parameter Parameter name: Parameter values: Linear system solver		lambda0_rfv linspace(0.5	,1.5,11)	
ionary segregated ametric segregated	Linear system solver:	Direct (UMF	PACK)	÷	ettings
Adaptive mesh refinement	Matrix symmetry:	Automatic		¢	
		Help	Apply	Cancel	(

**FIGURE 9.110** 2D\_NoPML\_CM\_1 model Solver Parameters

COMSOL Multiphysics software enter  $range(0.5, \frac{1}{10}, 1.5)$  in the Parameter values edit window.) See Figure 9.110.

Click OK. Using the menu bar, select Solve > Solve Problem.

### **Postprocessing and Visualization**

The default plot shows a surface plot of the scattered electric field, z-component (V/m). See Figure 9.111.

An alternative approach to viewing the effect of the dielectric lens on the plane wave within the modeling domain is to visualize the electric field, *z*-component.

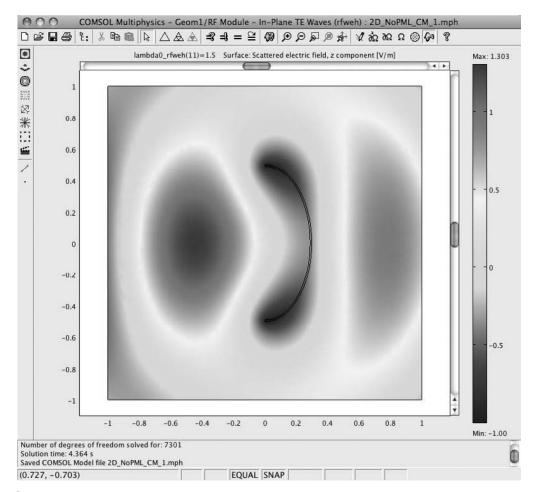


FIGURE 9.111 2D\_NoPML\_CM\_1 model solution, scattered electric field, z-component (V/m)

Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select "Electric field, z component" from the Predefined quantities pull-down list. See Figure 9.112.

Click OK. See Figure 9.113.

Using the menu bar, select Postprocessing > Plot Parameters > Animate. Verify that all the Solutions to use are selected. See Figure 9.114. Click the Start Animation button.

eneral Surface	Contour	Poundany	Arrow Brigging
Surface	Contour	Boundary	Arrow Principal
Surface plot			
	Surface Data	Height Data	
	Junace Data	Theight Data	
Predefined quantities:	Electric field, z	component	Range
Expression:	Ez		Smooth
Unit:	V/m		•
oloring and fill			
oloring: Interpo	lated 🗘	Fill style:	Filled
			1
urface color			
_	iet 🌒	Colors: 1024	M Calar crala
• Colormap:	jet 🗘	Colors: 1024	Color scale
• Colormap:	jet	Colors: 1024	Color scale
O Colormap:		Colors: 1024	Color scale
O Colormap:		Colors: 1024	Color scale
O Colormap:		Colors: 1024	Color scale
O Colormap:		Colors: 1024	Color scale
O Colormap:		Colors: 1024	Color scale
O Colormap:		Colors: 1024	Color scale
Olormap:		Colors: 1024	Color scale
O Colormap:		Colors: 1024	Color scale
• Colormap:		Colors: 1024	Color scale
O Colormap:		Colors: 1024	Color scale
• Colormap:		Colors: 1024	Color scale
• Colormap:		Colors: 1024	Color scale
• Colormap:		Colors: 1024	Color scale

**FIGURE 9.112** 2D\_NoPML\_CM\_1 model Plot Parameters, Surface tab

# 2D Concave Mirror Model, with and without PMLs: Summary and Conclusions

The 2D concave mirror models, with and without PMLs (2D\_PML\_CM\_1 and 2D\_NoPML\_CM\_1, respectively), have been built and solved. The best method of comparison between the two models is to view visualizations for the electric field, *z*-component for the same wavelength from each model together. Figures 9.115 through 9.120 show visualizations for 0.5 m (Figures 9.115 and 9.116), 1.0 m (Figures 9.117 and 9.118), and 1.5 m (Figures 9.119 and 9.120).

In comparison to the dielectric lens models presented in the first half of this chapter, it is apparent that there are also only small differences in the electric field, *z*-component visualizations between the PML and no-PML models for the concave mirror. This lack of large differences between the PML and no-PML models shows

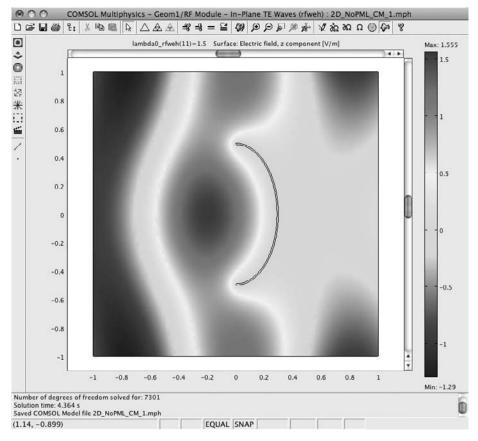


FIGURE 9.113 2D\_NoPML\_CM\_1 model electric field, z-component

the modeler that he or she needs to understand the relative importance of the modeled values needed to evaluate the application and the application environment so as to build the best model. The PML model best approximates a free space environment (no reflections). For other environments, the modeler needs to determine the best boundary condition approximation using standard practices and a first principles approach. When all else fails (or even before), do a first principles analysis of the environment before building the model.

Why do the solutions of the two dielectric lens models differ significantly and the solutions of the two concave mirror models converge to similar solutions? Consider the fact that a lossless dielectric is electromagnetically transparent and a metal mirror (e.g., copper) is electromagnetically opaque. Then a first principles analysis should answer the question.

# 738 CHAPTER 9 PERFECTLY MATCHED LAYER MODELS

Streamline	Particle Tracing	Max/Min	Deform	Animat
Movie settings		Solutions to	o use	
File type:	AVI 🛟	Select via:		A Y
Width (in pixels):	640	0.5	_	Ō
Height (in pixels):	480	0.6 0.7		
Frames per second:	10	0.8 0.9		
	Advanced	1		
Static / Eigenfunction	n animation	1.1		č
Cycle type:	Full harmonic	1.3 1.4		Ŧ
Number of frames:	11	Times:		
Reverse direction	ngs from main window			
			Start	Animation

**FIGURE 9.114** 2D\_NoPML\_CM\_1 model Plot Parameters, Animate tab

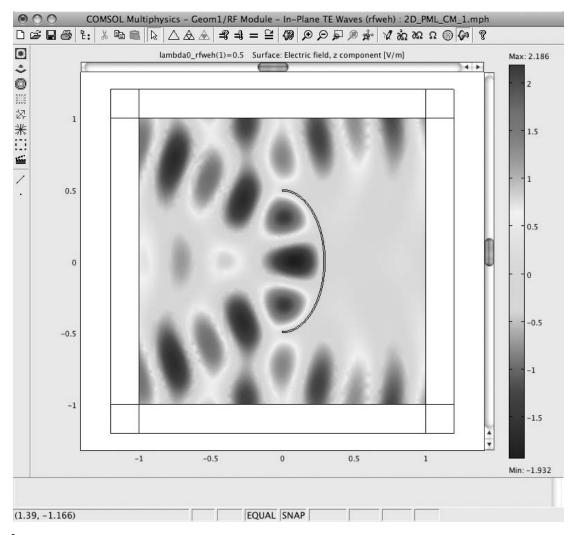


FIGURE 9.115 2D\_PML\_CM\_1 model plot electric field, z-component, 0.5 m

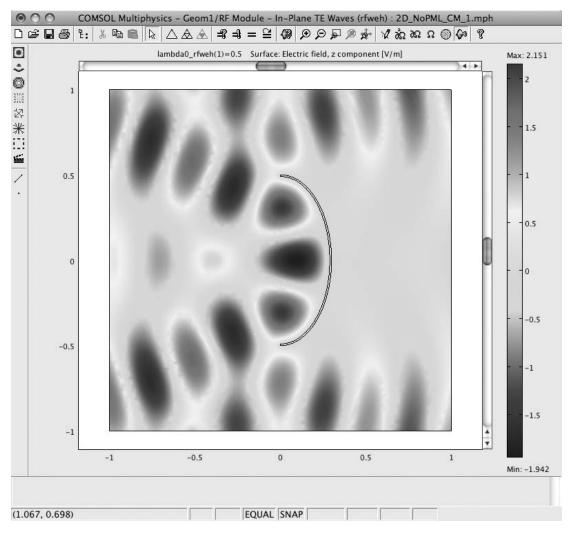


FIGURE 9.116 2D\_NoPML\_CM\_1 model plot electric field, z-component, 0.5 m

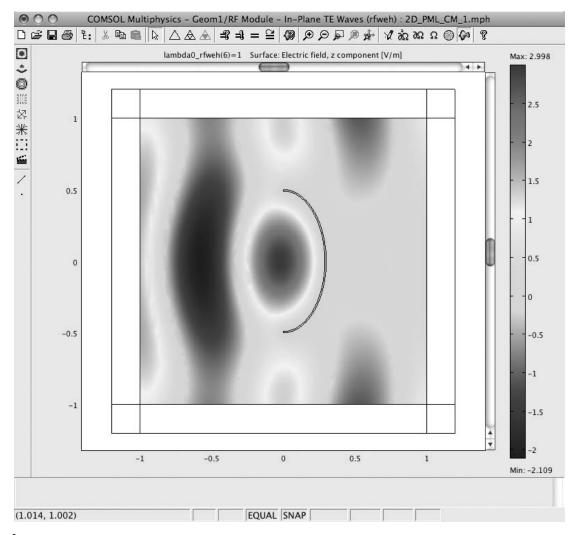


FIGURE 9.117 2D\_PML\_CM\_1 model plot electric field, z-component, 1.0 m

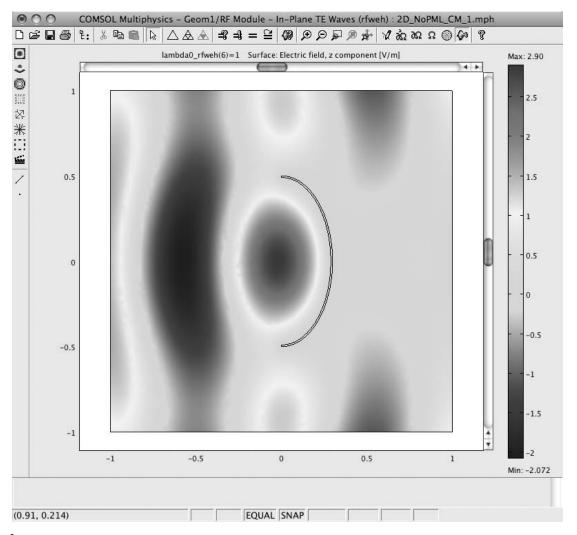


FIGURE 9.118 2D\_NoPML\_CM\_1 model plot electric field, z-component, 1.0 m

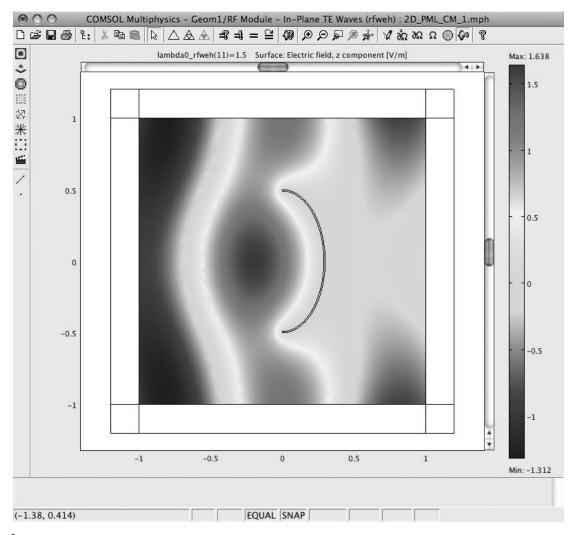


FIGURE 9.119 2D\_PML\_CM\_1 model plot electric field, z-component, 1.5 m

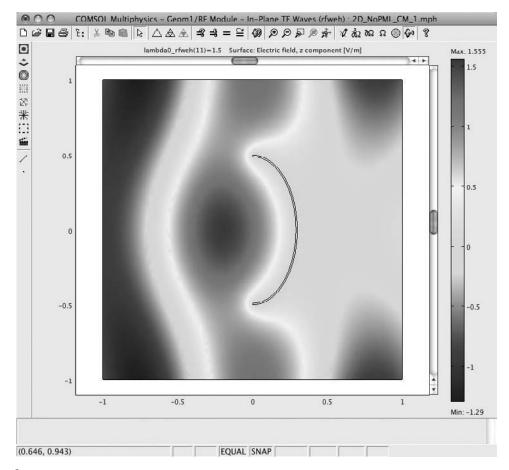


FIGURE 9.120 2D\_NoPML\_CM\_1 model plot electric field, z-component, 1.5 m

## References

- 1. http://en.wikipedia.org/wiki/Maxwell%27s\_Equations
- 2. http://en.wikipedia.org/wiki/Boundary\_conditions
- 3. J. P. Berenger, "A perfectly matched layer for the absorption of electromagnetic waves", J. Comput. Phys., Vol. 114, No. 2, 1994, pp. 185–200.
- 4. http://en.wikipedia.org/wiki/Finite-difference\_time-domain\_method
- 5. http://en.wikipedia.org/wiki/Perfectly\_matched\_layer
- 6. http://math.mit.edu/~stevenj/18.369/pml.pdf

- 7. *COMSOL RF Module User's Guide*, Version 3.4, October 2007, COMSOL AB, Stockholm, Sweden, pp. 43–48.
- 8. http://en.wikipedia.org/wiki/Concave\_mirror

#### Exercises

- 1. Build, mesh, and solve the 2D dielectric lens model, with PMLs, problem presented in this chapter.
- 2. Build, mesh, and solve the 2D dielectric lens model, without PMLs, problem presented in this chapter.
- 3. Build, mesh, and solve the 2D concave mirror model, with PMLs, problem presented in this chapter.
- 4. Build, mesh, and solve the 2D concave mirror model, without PMLs, problem presented in this chapter.
- 5. Explore other materials as applied in the 2D dielectric lens model, with PMLs.
- 6. Explore other materials as applied in the 2D dielectric lens model, without PMLs.
- 7. Explore other materials as applied in the 2D concave mirror model, with PMLs.
- 8. Explore other materials as applied in the 2D concave mirror model, without PMLs.
- 9. Explore the different geometries in the 2D dielectric lens model, with PMLs.
- 10. Explore the different geometries in the 2D concave mirror model, with PMLs.

# **10** Bioheat Models

# In This Chapter

Bioheat Modeling Guidelines and Coordinate Considerations **Bioheat Equation Theory** Tumor Laser Irradiation Theory 2D Axisymmetric Tumor Laser Irradiation Model 2D Axisymmetric Tumor Laser Irradiation Model: Summary and Conclusions Microwave Cancer Therapy Theory 2D Axisymmetric Microwave Cancer Therapy Model

2D Axisymmetric Microwave Cancer Therapy Model: Summary and Conclusions

# Bioheat Modeling Guidelines and Coordinate Considerations

# **Bioheat Equation Theory**

For the new modeler or those readers unfamiliar with this topic, bioheat modeling is the development of models for the analysis of heat transfer in materials (e.g., tissues, fluids) and systems derived from or related to currently or previously living organisms. The solution of the bioheat equation as applied to particular models is most important, obviously, when those models are developed to explore potential techniques for critical therapeutic applications (e.g., destroying cancer cells, killing tumors).

In August 1948, Harry H. Pennes published his landmark paper "Analysis of Tissue and Arterial Blood Temperatures in the Resting Human Forearm."<sup>1</sup> In that paper, he proposed that heat flow is proportional to the difference in temperature between the arterial blood and the local tissue. Pennes's work is considered fundamental in this area of study and has since been cited extensively.<sup>2</sup>

In the COMSOL<sup>®</sup> Multiphysics<sup>®</sup> software, the bioheat equation (Pennes equation) takes the form of an application mode within the Heat Transfer Module. In the Bioheat Equation Application Mode, the bioheat equation is formulated as follows:

$$\delta_{\rm ts}\rho C \frac{\partial T}{\partial t} + \nabla \cdot \left(-\overrightarrow{k}\nabla T\right) = \rho_{\rm b}C_{\rm b}\omega_{\rm b}(T_{\rm b} - T) + Q_{\rm met} + Q_{\rm ext}$$
(10.1)

where 
$$\delta_{ts} = time-scaling coefficient (default value = 1; dimensionless)$$
  
 $\rho = tissue density (kg/m^3)$   
 $C = tissue heat capacity [J/(kg \cdot K)]$   
 $T = temperature (K)$   
 $\vec{k} = tissue thermal conductivity tensor [W/(m \cdot K)]$   
 $\rho_b = blood density (kg/m^3)$   
 $C_b = blood heat capacity [J/(kg \cdot K)]$   
 $\omega_b = blood perfusion rate [m^3/(m^3 \cdot s)]$   
 $T_b = temperature, arterial blood (K)$   
 $Q_{met} = metabolic heat source (W/m^3)$   
 $Q_{ext} = external environmental heat source (W/m^3)$ 

**NOTE** The perfusion<sup>3</sup> rate is the rate at which a fluid (e.g., blood) flows through a type of tissue (e.g., muscle, heart, liver). It is, of course, very important to know the correct perfusion value for the tissue/fluid type in question.

Even though equation 10.1 is shown as formulated for blood flow, it can be equally well employed for other fluids or fluid compositions under the appropriate circumstances (e.g., artificial blood, different animal-life fluids). When employing variations of the formulation of the bioheat equation, modelers need to carefully verify the underlying assumptions employed in their particular model.

The bioheat equation is similar to the conduction heat equation. In the case of steady-state heat flow, the first term on the left vanishes:

$$\delta_{\rm ts}\rho C \frac{\partial T}{\partial t} = 0 \tag{10.2}$$

In the bioheat equation, what would normally be the single heat source term on the left side of the heat conduction equation (Q) is now separated into three terms.

The perfusion term:	$ ho_{ m b}C_{ m b}\omega_{ m b}(T_{ m b}-T)$	(10.3)
The metabolic term:	$Q_{ m met}$	(10.4)
The external source term:	$Q_{\rm ext}$	(10.5)

The division of the normally single heat source term in the bioheat equation into three terms is done to facilitate a conceptual linkage and to ease the formulation of the PDE when creating models for this type of problem (biological).

The bioheat equation, as constructed by Pennes, constitutes a good first-order approximation to the physical processes (thermal conduction) involved in the solution

of the heat transfer problem for biological specimens. This formulation is typically adequate for the modeling of most biological problems. More terms can, of course, be added if perceived as necessary, albeit at the risk of increased complexity, associated model size, and computational time.

However, because the bioheat equation already serves the needed level of accuracy for a typical decision point, little additional beneficial knowledge will be gained from the addition of second-order effects to the equation, considering the intrinsic fundamental limits of most biological system model problems.

## Tumor Laser Irradiation Theory

The optical coefficient of absorption for laser photons (irradiation) of tumors does not generally differ significantly from the optical coefficient of absorption for the surrounding tissue. To develop this laser irradiation therapeutic methodology, it is necessary to raise the local absorption coefficient by artificial means. The change in absorption coefficient is accomplished by injection into the tumor of a designed high-absorption material.<sup>4</sup> This type of procedure is usually designated as a minimally invasive procedure.

The laser beam energy contributes a heat source to the bioheat equation as follows:

$$Q_{\rm laser} = I_0 a e^{az - \frac{r^2}{2\sigma^2}}$$
(10.6)

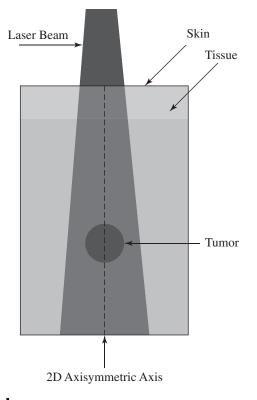
where

 $I_0$  = irradiation intensity (W/m<sup>2</sup>) a = absorptivity (1/m)  $\sigma$  = irradiated region width parameter (m)

### 2D Axisymmetric Tumor Laser Irradiation Model

The following numerical solution model (2D\_Bio\_TLI\_1 model) is derived from a model that was originally developed by COMSOL as a Heat Transfer Module tutorial model for the demonstration of the solution of a bioheat equation model. That model was developed for distribution with the Heat Transfer Module software as part of the COMSOL Heat Transfer Module Model Library.

**NOTE** The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C  $(315.15 \text{ K})^5$  to 60 °C (333.15 K).<sup>6</sup> If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.



**FIGURE 10.1** 2D\_Bio\_TLI\_1 model modeling domain overview

This first model takes advantage of the transparency of human tissue in certain infrared (IR) wavelengths.<sup>7</sup> Figure 10.1 shows the structure of the modeling domain. Because the model is created as a 2D axisymmetric model, only the right half of the structure will be used in the calculations.

To start building the 2D\_Bio\_TLI\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > Bioheat Equation > Transient analysis. See Figure 10.2. Click OK.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 10.1; see also Figure 10.3. Click OK.

#### **Geometry Modeling**

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 10.2. Click OK after filling in the parameters of each separate rectangle in the Rectangle edit window. See Figures 10.4 and 10.5.

0.0	Model Navigator	
Space dimension:	Model Library User Models	Open Settings
Transient	tule Transfer on te analysis analysis essible Navier-Stokes e Model ce Model lal Interaction Interaction	Description: Hear transfer described by the Biohear equation, specifying heat sources from blood perfusion, metabolism, and spatial heating. Transfert analysis in 2D axial symmetry.
Dependent variables: Application mode name:	T htbh	aluuro l.
Element	Lagrange - Quadra	( Multiphysics )

**FIGURE 10.2** 2D\_Bio\_TLI\_1 Model Navigator setup

Table 10.1 (	Constants Edit	Window
--------------	----------------	--------

Name	Expression	Description
rho_blood	1000[kg/m^3]	Density blood
C_blood	4200[J/(kg*K)]	Heat capacity blood
T_blood	37[degC]	Temperature blood
k_skin	0.2[W/(m*K)]	Thermal conductivity skin
rho_skin	1200[kg/m^3]	Density skin
C_skin	3600[J/(kg*K)]	Heat capacity skin
wb_skin	3e-3[1/s]	Blood perfusion rate skin
k_tissue	0.5[W/(m*K)]	Thermal conductivity tissue
rho_tissue	1050[kg/m^3]	Density tissue
C_tissue	3600[J/(kg*K)]	Heat capacity tissue
wb_tissue	6e-3[1/s]	Blood perfusion rate tissue
k_tumor	0.5[W/(m*K)]	Thermal conductivity tumor
rho_tumor	1050[kg/m^3]	Density tumor
C_tumor	3600[J/(kg*K)]	Heat capacity tumor
wb_tumor	6e-3[1/s]	Blood perfusion rate tumor
Q_met	400[W/m^3]	Metabolic heat generation
TO	37[degC]	Temperature reference blood
h_conv	10[W/(m^2*K)]	Heat transfer coefficient skin
T_inf	10[degC]	Temperature domain boundary
10	1.4[W/mm^2]	Laser irradiation power
sigma	5[mm]	Laser beam width coefficient

C_blood         42           T_blood         37           k_skin         0.2           rho_skin         12           C_skin         36           wb_skin         3e	00[kg/m^3] 00[J/(kg*K)] [degC] 2[W/(m*K)] 00[kg/m^3]	1000[kg/m <sup>3</sup> ] 4200[J/(kg·K)] 310.15[K] 0.2[W/(m·K)]	Density blood Heat capacity blood Temperature blood
T_blood 37 k_skin 0.2 rho_skin 12 C_skin 36 wb_skin 3e	[degC] 2[W/(m*K)] 00[kg/m^3]	310.15[K]	Temperature blood
k_skin 0.2 rho_skin 12 C_skin 36 wb_skin 3e	2[W/(m*K)] 00[kg/m^3]		
rho_skin 12 C_skin 36 wb_skin 3e	00[kg/m^3]	0.2[W/(m·K)]	
C_skin 36 wb_skin 3e			Thermal conductivity skin
wb_skin 3e		$1200[kg/m^3]$	Density skin
	00[J/(kg*K)]	3600[J/(kg·K)]	Heat capacity skin
k tissue 01	-3[1/s]	0.003[1/s]	Blood perfusion rate skin
K_USSUE U.	5[W/(m*K)]	0.5[W/(m·K)]	Thermal conductivity tissue
rho_tissue 10	50[kg/m^3]	1050[kg/m <sup>3</sup> ]	Density tissue
C_tissue 36	00[J/(kg*K)]	3600[J/(kg·K)]	Heat capacity tissue
wb_tissue 6e	-3[1/s]	0.006[1/s]	Blood perfusion rate tissue
k_tumor 0.5	5[W/(m*K)]	0.5[W/(m·K)]	Thermal conductivity tumor
rho_tumor 10	50[kg/m^3]	$1050[kg/m^3]$	Density tumor
C_tumor 36	00[J/(kg*K)]	3600[J/(kg·K)]	Heat capacity tumor
wb_tumor 6e	-3[1/s]	0.006[1/s]	Blood perfusion rate tumor
Q_met 40	0[W/m^3]	400[W/m <sup>3</sup> ]	Metabolic heat generation
T0 37	[degC]	310.15[K]	Temperature reference blood
h_conv 10	[W/(m^2*K)]	$10[W/(m^2 \cdot K)]$	Heat transfer coefficient skin
T_inf 10	[degC]	283.15[K]	Temperature domain boundary
10 1.4	4[W/mm^2]	1.4e6[W/m <sup>2</sup> ]	Laser irradiation power
sigma 5[I	mm]	0.005[m]	Laser beam width coefficient
(			) 4 >

**FIGURE 10.3** 2D\_Bio\_TLI\_1 model Constants (R1) edit window

Table 10.2	<b>Geometry Components</b>	

Name	Width	Height	Base	r	z	Figure Number
R1	0.1	0.09	Corner	-0.05	-0.1	10.4
R2	0.1	0.01	Corner	-0.05	-0.01	10.5

Size		Rotat	ion angle-	
Width:	0.1	α:	0	(degrees)
Height:	0.09			
Position				
Base:	Corner	Style:	Solid	
r:	-0.05	Name:	R1	
	-0.1			

FIGURE 10.4 2D\_Bio\_TLI\_1 model Rectangle (R1) edit window

Size		Rotat	tion angle	
Width	0.1	α:	0	(degrees)
Height:	0.01			
Position				
Base:	Corner	\$ Style:	Solid	
r:	-0.05	Name:	R2	
z:	-0.01			

FIGURE 10.5 2D\_Bio\_TLI\_1 model Rectangle (R2) edit window

Click the Zoom Extents button. See Figure 10.6.

Select File > Save As. Enter 2D\_Bio\_TLI\_1.mph in the Save As edit window. See Figure 10.7. Click the Save button.

Using the menu bar, select Draw > Specify Objects > Circle. Enter Radius = 0.005, Base = Center, r = 0, and z = -0.05. See Figure 10.8. Click OK.

12_Model	s 🗘		
Name	Date Modified		
LI_BioHeat_1.mph	Monday, May 4, 2009 1:50 PM		
LL_BioHeat_1a.mph LL_BioHeat_Constants_1.txt	Monday, May 4, 2009 1:50 PM		
LL_BioHeat_Constants_1.txt LL_BioHeat_SV_1.txt	Monday, May 4, 2009 10:58 AM Monday, May 4, 2009 11:26 AM		
MCT_BioHeat_la.mph	Monday, May 4, 2009 5:44 PM		
MCT_BioHeat_1a.txt	Monday, May 4, 2009 4:51 PM		
File Format: CO	MSOL Multiphysics Mo		

FIGURE 10.7 2D\_Bio\_TLI\_1 model Save As edit window

Using the menu bar, select Draw > Specify Objects > Rectangle. Enter Width = 0.05, Height = 0.1, Base = Corner, r = -0.05, and z = -0.1. See Figure 10.9. Click OK.

Using the menu bar, select Draw > Create Composite Object. Enter R1+R2+C1-R3 in the Set formula edit window. Check the Keep interior boundaries check box. See Figure 10.10.

Size		Rotat	ion angle	
Radius:	0.005	α:	0	(degrees
Position				
Base:	Center	\$ Style:	Solid	\$
r:	0	Name:	C1	
z:	-0.05			

FIGURE 10.8 2D\_Bio\_TLI\_1 model Circle (C1) edit window

Size		Rota	ation angle-	
Width:	0.05	α:	0	(degrees)
Height:	0.1			
Position				
Base:	Corner	\$ Style:	Solid	\$
r	-0.05	Name	: R3	
	-0.1			

FIGURE 10.9 2D\_Bio\_TLI\_1 model Rectangle (R3) edit window

Click OK. See Figure 10.11.

Having established the geometry for the 2D\_Bio\_TLI\_1 model, the next step is to define the fundamental Physics properties.

#### **Physics Settings: Scalar Expressions**

Select Options > Expressions > Scalar Expressions. Enter Name = Q\_laser. Enter Expression =  $I0*a*exp(a*z-r^2/(2*sigma^2))$ . See Figure 10.12. Click OK.

#### **Physics Settings: Subdomain Expressions**

Select Options > Expressions > Subdomain Expressions.

**NOTE** In the entries in the Subdomain Expressions window, the variable Name a needs to be entered only once, as indicated by the following instructions.

Object type	Shortcuts
Solids	Иліол ОК
-	Cancel
O Curves	Intersection
O Points	Select All Apply
bject selection:	Set formula:
R1	R1+R2+C1-R3
R2 C1	Keep interior boundaries
R3	

FIGURE 10.10 2D\_Bio\_TLI\_1 model Create Composite Object edit window

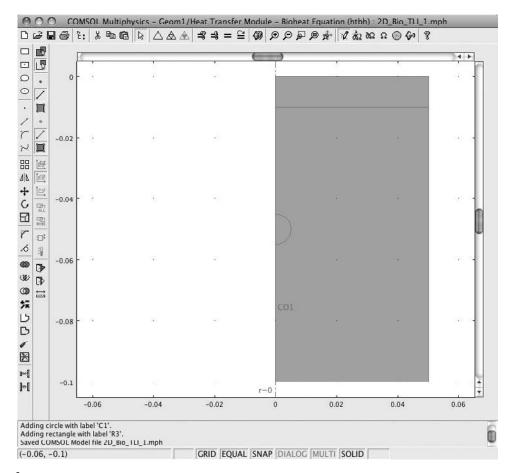


FIGURE 10.11 2D\_Bio\_TLI\_1 model domain: skin, tissue, and tumor (CO1)

Name	Expression	Unit	Description
Q_laser	10*a*exp(a*z-r^2/(2*sigma^2))	0	Laser energy distribution
c	Help	Apply	Cancel OK

**FIGURE 10.12** 2D\_Bio\_TLI\_1 model Scalar Expressions edit window

a 0.1[1/m] 1/m		Name	Expression	Unit
		a	0.1[1/m]	1/m
Select by group	Select by group			

FIGURE 10.13 2D\_Bio\_TLI\_1 model Subdomain Expressions (1) edit window

For subdomain 1, enter Name = a, Expression = 0.1[1/m]. See Figure 10.13. For subdomain 2, enter Expression = 4[1/m]. See Figure 10.14. For subdomain 3, enter Expression = 0.1[1/m]. See Figure 10.15. Click OK.

000		Subdomain Expressions	
Subdomain selection	Name a	Expression 4[1/m]	Unit 1/m
3			
Select by group			
		Help Apply	Cancel OK

FIGURE 10.14 2D\_Bio\_TLI\_1 model Subdomain Expressions (2) edit window

main selection	Name	Expression	Unit
	a	0.1[1/m]	1/m
elect by group			
nect by group			

FIGURE 10.15 2D\_Bio\_TLI\_1 model Subdomain Expressions (3) edit window

Name	Subdomain 1	Subdomain 2	Subdomain 3
k (isotropic)	k_tissue	k_tumor	k_skin
ρ	rho_tissue	rho_tumor	rho_skin
С	C_tissue	C_tumor	C_skin
$ ho_{b}$	rho_blood	rho_blood	rho_blood
C <sub>b</sub>	C_blood	C_blood	C_blood
$\omega_{\text{b}}$	wb_tissue	wb_tumor	wb_skin
T <sub>b</sub>	T_blood	T_blood	T_blood
Q <sub>met</sub>	Q_met	Q_met	Q_met
O <sub>ext</sub>	Q_laser	Q_laser	Q_laser

#### Table 10.3Subdomain Settings

### Physics Subdomain Settings: Bioheat Equation (htbh)

Having established the basic Physics settings for the 2D\_Bio\_TLI\_1 model, the next step is to define the fundamental Physics subdomain setting. Using the menu bar, select Physics > Subdomain Settings. In the Subdomain Settings edit window, enter the information shown in Table 10.3 and Figures 10.16, 10.17, and 10.18. Click the Apply button after filling in the parameters of each separate subdomain in the subdomain edit window.

FIGURE 10.17 2D\_Bio\_TLI\_1 model Subdomain Settings (2) edit window

00	Subdomain Settings - Bioheat Equation (htbh)
quation <sub>ts</sub> pC∂T/∂t + ∇·(-k⊽T) = p <sub>b</sub> C <sub>b</sub> u	$u_b(T_b - T) + Q_{met} + Q_{ext}, T = temperature$
Subdomains Groups Subdomain selection	Physics Init Element Color Initial value T(t <sub>0</sub> ) TO K Temperature
Group: 🔅	

FIGURE 10.19 2D\_Bio\_TLI\_1 model Subdomain Settings, Initial value edit window

Select subdomains 1, 2, and 3. Click the Init tab. Enter T0 in the Initial value edit window. Click the Apply button. See Figure 10.19. Click OK.

## **Physics Boundary Settings: Bioheat Equation (htbh)**

Having established the subdomain settings for the 2D\_Bio\_TL1\_1 model, the next step is to define the fundamental Physics boundary settings. Using the menu bar, Select Physics > Boundary Settings. In the Boundary Settings edit window, enter the information shown in Table 10.4. Click the Apply button after choosing or entering the

Boundary	Boundary Condition	Parameter	Value	Figure Number
1, 3–5	Axial symmetry	—	_	10.20
2, 8, 9	Thermal insulation	—	_	10.21
7	Heat flux	h	h_conv	10.22
		T <sub>inf</sub>	T_inf	

#### Table 10.4 Boundary Settings

Boundaries Groups		Coefficients	Color/Style	
Boundary selection	Boundary sources an	d constraints		
2	Boundary condition.	Axial symmetry	•	
3	Quantity	Value/Expression	Unit	Description
ę U	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
6	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficient
7 A 8 Y	Tinf	273.15	— к	External temperature
Group:	то	310.15	— к	Temperature
Select by group				
Interior boundaries				

FIGURE 10.20 2D\_Bio\_TLI\_1 model Boundary Settings (1, 3–5) edit window

parameters of each Boundary Settings group in the Boundary Settings edit window. Click OK. See Figures 10.20, 10.21, and 10.22.

#### **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Subdomain tab. Select subdomain 2 (the tumor). Enter 0.005 in the Maximum element size edit window. Select "Quad" from the Method pull-down list. See Figure 10.23.

Boundaries Groups		Coefficients	Color/Style	
Boundary selection	Boundary sources an	d constraints		
4 5	Boundary condition.	Thermal insulation	\$	
° n	Quantity	Value/Expression	Unit	Description
δ	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
9	h	0	W/(m <sup>2</sup> ·K)	Heat transfer coefficien
10 A	T <sub>inf</sub>	273.15	— к	External temperature
Group: (	τ <sub>0</sub>	310.15	к	Temperature
Select by group				
Interior boundaries				

FIGURE 10.21 2D\_Bio\_TLI\_1 model Boundary Settings (2, 8, 9) edit window

Boundaries Groups		Coefficients	Color/Style	
Boundary selection	Boundary sources an	d constraints		
4	Boundary condition:	Heat flux	÷	
6	Quantity	Value/Expression	Unit	Description
7	q <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
9	h	h_conv	W/(m <sup>2</sup> ·K)	Heat transfer coefficien
10 <b>A</b> 11 <b>Y</b>	T <sub>inf</sub>	T_inf	к	External temperature
Group: 📫	то	310.15	к	Temperature
Select by group				
Interior boundaries				

**I FIGURE 10.22** 2D\_Bio\_TLI\_1 model Boundary Settings (7) edit window

	Free Mesh Parameters	
Global Su	bdomain Boundary Point Advanced	_ Сок
Subdomain selection	Subdomain mesh parameters Maximum element size: 0.005 Element growth rate: Method: Quad \$	Cano App Hel
Select by group Select Remaining Select Meshed		
	Remesh Mesh Selected	

**FIGURE 10.23** 2D\_Bio\_TLI\_1 model Free Mesh Parameters, subdomain 2 edit window

**FIGURE 10.24** 2D\_Bio\_TLI\_1 model mesh

Click the Mesh Selected button. Click the Select Remaining button. Click the Mesh Selected button. Click OK. See Figure 10.24.

#### Solving the 2D\_Bio\_TLI\_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Time dependent" in the Solver list. Enter 0:10:600 in the Times edit window. See Figure 10.25.

Click OK. Using the menu bar, select Solve > Solve Problem.

#### Postprocessing and Visualization

The default plot shows a surface plot of the temperature (K). See Figure 10.26.

Typically, such analytical plots are viewed in degrees Centigrade. The plot can be easily converted through the following steps. Using the menu bar, Select

Analysis	6	eneral Time S	tepping Advanced	٦.
Transient	Time stepping	Time s	epping Advanced	
Auto select solver	Times:		0:10:600	-
Stationary	Relative tolerance:		0.01	
Time dependent Eigenvalue Parametric Stationary segregated	Absolute tolerance:	umbers	0.0010	
Parametric segregated	-Linear system solver Linear system solver: Preconditioner:	Direct (UMFPA	лск) 🛟	(Settings)
	Matrix symmetry:	Automatic		
		Help	(Apply) (C	uncel OK

FIGURE 10.25 2D\_Bio\_TLI\_1 model Solver Parameters edit window

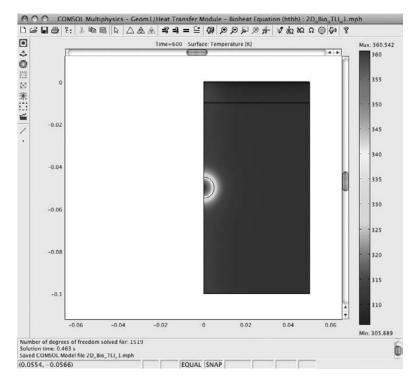


FIGURE 10.26 2D\_Bio\_TLI\_1 model solution, temperature (K)

General	Surface	Cantaur	Baundama	A	Deinsigal	-
	Surface	Contour	Boundary	Arrow	Principal	
Surface	plot					
0						
		Surface Data	Height Data			
Duadafinas	d quantities:	Temperature	\$	Range		
Frederined	i quantities.	remperature				
Expressio	n:	Т		Smoot	h	
Unit:		°c	\$	1		
		53		50 		
Coloring an	d fill					
Coloring:	Interpo	ated 🛟	Fill style:	Filled	\$	
coloring.		acco v	r in style.	Thicu		
Surface colo	or					
Colorn	nap:	jet 🛟	Colors: 1024	Color :	cale	
	m color:	Color				
O Unifor	m color:	Color				
O Unifor	m color:	Color				
O Unifor	m color: (	Color)	<u> </u>			
O Unifor	m color:	Color				
O Unifor	m color: (	Color	<u> </u>		,	
O Unifor	m color: (	Color	<u> </u>			
O Unifor	m color: (	Color				
O Unifor	m color: (	Color	<u> </u>			
O Unifor	m color: (	Color	<u> </u>			
O Unifor	m color: (	Color	<u> </u>			
O Unifor	m color: (	Color	<u> </u>			
OUnifor	m color:	Color				

FIGURE 10.27 2D\_Bio\_TLI\_1 model Plot Parameters, Surface tab (°C) edit window

Postprocessing > Plot Parameters > Surface. Select "degC (°C)" from the Unit pulldown list. See Figure 10.27.

Click OK. See Figure 10.28.

**NOTE** The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C  $(315.15 \text{ K})^5$  to 60 °C (333.15 K).<sup>6</sup> If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

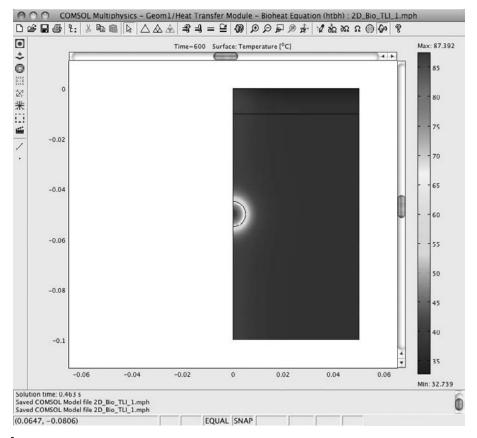


FIGURE 10.28 2D\_Bio\_TLI\_1 model, surface temperature (°C)

Now that the 2D\_Bio\_TLI\_1 model has been successfully calculated, the modeler can determine the time to reach the desired temperature for a preliminary estimate of an effective treatment. Final determination of the veracity of the calculation will, of course, need to be made experimentally. The results (estimated time values) from the model calculations will significantly reduce the effort needed to determine an accurate experimental value.

To determine the time to the desired temperature of 60 °C at the boundary of the tumor, proceed as follows. Using the menu bar, select Postprocessing > Domain Plot Parameters. Click the Point tab. Select point 6 in the Point selection window. Select "Temperature" from the Predefined quantities pull-down list. Select "degC (°C)" from the Unit pull-down list. See Figure 10.29.

Click OK. Figure 10.30 shows that the time to 60 (C at the boundary of the tumor is approximately 220 seconds under the specified conditions of this model).

Expression: T Unit: CC	
	•

**FIGURE 10.29** 2D\_Bio\_TLI\_1 model Domain Plot Parameters, Point tab



FIGURE 10.31 2D\_Bio\_TLI\_1 model Plot Parameters, Animate tab

The development of the entire model can be viewed by following these steps. Using the menu bar, select Postprocessing > Plot Parameters > Animate. Select all solutions in the Solutions to use window. See Figure 10.31.

Click the Start Animation button. See Figure 10.32.

# 2D Axisymmetric Tumor Laser Irradiation Model: Summary and Conclusions

The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. Now that the 2D\_Bio\_TLI\_1 model has been successfully calculated, the modeler can determine the time to reach the desired temperature for a preliminary estimate of an effective treatment. Final determination of the veracity of

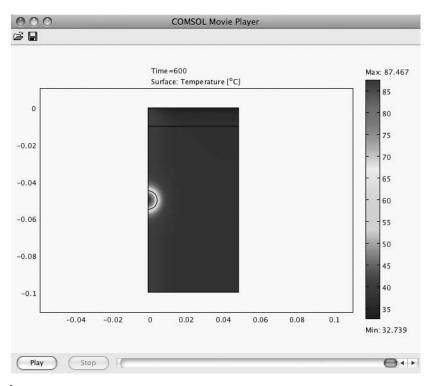


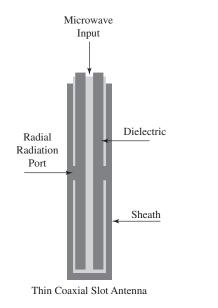
FIGURE 10.32 2D\_Bio\_TLI\_1 model Plot Parameters, animation, final frame

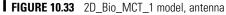
the calculation will, of course, need to be made experimentally. The literature cites temperatures that range from 42 °C (315.15 K) to 60 °C (333.15 K).<sup>6</sup>

If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful. The results (estimated time values) from the model calculations will significantly reduce the effort needed to determine an accurate experimental value. The guiding principle needs to be that tumor cells die at elevated temperatures.

# Microwave Cancer Therapy Theory

Hyperthermic (high-temperature) oncology (cancer, tumor)<sup>8</sup> involves the use of elevated temperatures to kill cancer and other tumor cells. As discussed in the previous model, it is necessary to locally raise the temperature of the cancer/tumor cells, without doing significant damage to the normal (healthy) cells surrounding the tumor. In the previous model, the energy was supplied as photothermal energy using laser irradiation. In this model, the externally applied energy is supplied through the use of a





specialized microwave antenna and the application of Ohm's and Joule's Laws.<sup>9,10</sup> This type of procedure is typically designated as a minimally invasive procedure.<sup>11</sup>

Figure 10.33 shows the microwave antenna in cross section.

# 2D Axisymmetric Microwave Cancer Therapy Model

The following numerical solution model (2D\_Bio\_MCT\_1 model) is derived from a model that was originally developed by COMSOL as a Heat Transfer Module tutorial model for the demonstration of the solution of a bioheat equation model. That model was developed for distribution with the Heat Transfer Module software as part of the COMSOL Heat Transfer Module Model Library.

**NOTE** The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C  $(315.15 \text{ K})^5$  to 60 °C (333.15 K).<sup>6</sup> If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

This model takes advantage of the conductivity of human tissue. Figure 10.34 shows the microwave antenna in cross section, embedded in the modeling domain (tissue) and radiating power. Because the model is created as a 2D axisymmetric model, only the right half of the structure is used in the calculations.

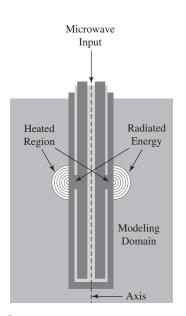


FIGURE 10.34 2D\_Bio\_MCT\_1 model, antenna plus tissue

To start building the 2D\_Bio\_MCT\_1 model, activate the COMSOL Multiphysics software. In the Model Navigator, select "Axial symmetry (2D)" from the Space dimension pull-down list. Select Heat Transfer Module > Bioheat Equation > Steady-state analysis. Click the Multiphysics button, and then click the Add button. See Figure 10.35.



FIGURE 10.35 2D\_Bio\_MCT\_1 Model Navigator, Heat Transfer Module

Iodel Library User Models	Open Settings
ial symmetry (2D)	Multiphysics
le Waves	Add Remove Geom1 (2D) Bioheat Equation (htbh) TM Waves (rfwh)
c propagation quency analysis t propagation e Waves Analysis	Dependent variables: Hphidr Application Mode Propertie
Interaction	Add Geometry
phidr2	Add Frame
fwh2	Ruling application mode: Bioheat Equation (htbh)
Lagrange - Quadra 🗘	Multiphysics
	Waves c propagation quency analysis t propagation e Waves Analysis Interaction s Module

FIGURE 10.36 2D\_Bio\_MCT\_1 Model Navigator, RF Module

Select RF Module > Electromagnetic Waves > TM Waves > Harmonic propagation. Select "Lagrange-Quartic" from the Element pull-down list. Click the Add button. See Figure 10.36. Click OK.

NOTE To verify the Lagrange-Quartic choice, the modeler can at any time go to the menu bar, select "Model Navigator," select the Application mode of choice, click on the Application Mode Properties button, and then verify the setting of choice.

#### Constants

Using the menu bar, select Options > Constants. In the Constants edit window, enter the information shown in Table 10.5; see also Figure 10.37. Click OK.

Using the menu bar, select File > Save As. Enter 2D\_Bio\_MCT\_1.mph in the Save As edit window. See Figure 10.38. Click the Save button.

#### **Geometry Modeling**

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 10.6. Click OK after filling in the parameters of each separate rectangle in the Rectangle edit window. See Figures 10.39 and 10.40.

Name	Expression	Description
k_liver	0.56[W/(m*K)]	Thermal conductivity liver
rho_blood	1e3[kg/m^3]	Density blood
C_blood	3639[J/(kg*K)]	Heat capacity blood
omega_blood	3.6e-3[1/s]	Blood perfusion rate
T_blood	37[degC]	Temperature blood
P_in	10[VV]	Microwave power input
nu	2.45[GHz]	Microwave frequency
eps_diel	2.03	Dielectric relative permittivity
eps_cat	2.6	Catheter relative permittivity
eps_liver	43.03	Liver relative permittivity
sig_liver	1.69[S/m]	Conductivity liver

#### Table 10.5Constants Edit Window

k_liver         0.56[W/(m*K)]         0.56[W/(m K)]         Thermal conc           rho_blood         1e3[kg/m^3]         1000[kg/m <sup>3</sup> ]         Density blooc           C_blood         369U/(kg*K)         3669U/(kg)/(kg)         Heat capacity           omega_blood         3.6e-3[1/s]         0.0036[1/s]         Blood perfusi           Jobod         37[degc]         310.15[K]         Temperature	blood
C_blood 3639[J/(kg*K)] 3639[J/(kg*K)] Heat capacity omega_blood 3.6e-3[1/s] 0.0036[1/s] Blood perfusi	blood
omega_blood 3.6e-3[1/s] 0.0036[1/s] Blood perfusi	
T blood 37[deoC] 310.15[K] Temperature	on rate
	blood
P_in 10[W] 10[W] Microwave po	wer input
nu 2.45[GHz] 2.45e9[1/s] Microwave fre	quency
eps_diel 2.03 2.03 Dielectric rela	tive permittivity
eps_cat 2.6 2.6 Catheter relat	tive permittivity
eps_liver 43.03 43.03 Liver relative	permittivity
sig_liver 1.69[5/m] 1.69[5/m] Conductivity	liver

FIGURE 10.37 2D\_Bio\_MCT\_1 model Constants edit window

Name	Width	Height	Base	r	z	Figure Number
R1	0.595e-3	0.01	Corner	0	0	10.39
R2	29.405e-3	0.08	Corner	0.595e-3	1.25e-30	10.40

Table 10.6	Geometry (	Components
------------	------------	------------

Click the Zoom Extents button. See Figure 10.41.

Using the menu bar, select Draw > Create Composite Object. Enter R1+R2 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 10.42.

Click OK. See Figure 10.43.

**NOTE** The composite object (CO1) created through these steps forms the modeling domain that constitutes the liver tissue.

Size		Rota	tion angle	5
Width:	0.595e-3	α:	0	(degrees)
Height:	0.01			
Positior	i			
Base:	Corner	Style:	Solid	\$
Base: r:	Corner 0	Style:		•

FIGURE 10.39 2D\_Bio\_MCT\_1 model Rectangle (R1) edit window

Size		Rotat	ion angle	
Width:	29.405e-3	α:	0	(degrees
Height:	0.08			
Position				
Base:	Corner	\$ Style:	Solid	\$
r:	0.595e-3	Name:	R2	
Z:	0			

FIGURE 10.40 2D\_Bio\_MCT\_1 model Rectangle (R2) edit window

31		ዩ:	光 陶 (	8 6 2		-\$° -\$ = \$	≧ (Ø)	( a Q Q	© ≱-   ½	20 26 26	0 60 1	3	
e		10					(						•
ď			25		2				1	1			ſ
•	0.08	-				5		0					- 1
1						1							- 1
重						1							1
9	0.07	1	*10										- 1
1													- 1
	0.06	-	42								1		-
極													- 1
E													- 1
10	0.05	-	45										-
맒													
						i.							
0	0.04	-	**	•		a I			P	•			1
1						i							- 1
•	0.03												- 4
D.	0.01					1							- 1
=						1							- 1
	0.02	-	7										-
													- 1
													- 1
	0.01	F	<u>13</u>			Ī				•		1	1
						R.1							- 1
	0		+1					RZ					
						r=0							2
			-0.03	-0.02	-0.01	0	0.01	0.02	0.03	0.04	0.05	0.06	
ing r ing r	ectang	e with e with	label 'R1' label 'R2' file 2D_Bi										

**FIGURE 10.41** 2D\_Bio\_MCT\_1 model rectangles (R1, R2)

000 c	reate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection: R1 R2	Set formula: R1+R2 Keep interior boundaries Repair Repair 1.0E-4

**FIGURE 10.42** 2D\_Bio\_MCT\_1 model Create Composite Object edit window

FIGURE 10.43 2D\_Bio\_MCT\_1 model composite object (C01)

Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 10.7. Click OK after filling in the parameters of each separate rectangle in the Rectangle edit window. See Figures 10.44 and 10.45.

Using the menu bar, select Draw > Create Composite Object. Enter R1+R2 in the Set formula edit window. Uncheck the Keep interior boundaries check box. See Figure 10.46. Click OK.

Name	Width	Height	Base	r	z	Figure Number
R1	0.125e-3	1.0e-3	Corner	0.47e-3	0.0155	10.44
R2	3.35e-4	0.0699	Corner	0.135e-3	0.0101	10.45

Table 10.7 Geometry Components

Size		Rotat	ion angle	
Width:	0.125e-3	α:	0	(degrees
Height:	1.0e-3			
Position				
Base:	Corner	\$ Style:	Solid	•
r:	0.47e-3	Name:	R1	
z:	0.0155			

FIGURE 10.44 2D\_Bio\_MCT\_1 model Rectangle (R1) edit window

Size		Rotation ang	le
Width:	3.35e-4	α: 0	(degrees)
Height:	0.0699		
Positior	1		
Base:	Corner	Style: Soli	d 🗘
r:	0.135e-3	Name: R2	
z:	0.0101		

FIGURE 10.45 2D\_Bio\_MCT\_1 model Rectangle (R2) edit window

000 c	reate Composite Object
Object type Solids Curves Points	Shortcuts Union Intersection Select All Help
Object selection:	Set formula:
R1 R2	Keep interior boundaries         Repair         Repair tolerance:

FIGURE 10.46 2D\_Bio\_MCT\_1 model Create Composite Object edit window

Coord	inates	ОК
r: [	0 8.95e-4 8.95e-4	UK
z: [	9.5e-3 0.01 0.08	Cancel
Style:	Polyline 🗘	Apply
Name:	B1	Help

FIGURE 10.47 2D\_Bio\_MCT\_1 model Line (B1) edit window

	<b>Table 10.8</b>	Geometry Components	;
--	-------------------	---------------------	---

Name	Width	Height	Base	r	z	Figure Number
R3	1.25e-4	1.0e-3	Corner	4.7e-4	0.0155	10.48

**NOTE** The composite object created through these steps (CO2) forms the modeling domain that constitutes the antenna dielectric.

Next, add a line to the geometry. Using the menu bar, select Draw > Specify Objects > Line. In the r edit window, enter 0 8.95e-4 8.95e-4. In the z edit window, enter 9.5e-3 0.01 0.08. See Figure 10.47. Click OK.

#### NOTE The line created through these steps forms the boundary of the antenna sheath.

Add the last rectangle to the geometry. Using the menu bar, select Draw > Specify Objects > Rectangle. In the Rectangle edit window, enter the information shown in Table 10.8. Click OK after filling in the parameters of the rectangle in the Rectangle edit window. See Figure 10.48.

Size		Rotat	tion angle	
Width:	1.25e-4	α:	0	(degrees)
Height:	1.0e-3			
Position				
Base:	Corner	\$ Style:	Solid	\$
r:	4.7e-4	Name:	R1	
	0.0155			

FIGURE 10.48 2D\_Bio\_MCT\_1 model Rectangle (R1) edit window

FIGURE 10.49 2D\_Bio\_MCT\_1 model, antenna and model domain

**NOTE** The rectangle created through these steps forms the modeling domain that constitutes the slot in the coaxial antenna that allows energy to be radiated into the liver tissue.

Click OK. See Figure 10.49.

Having established the geometry for the 2D\_Bio\_MCT\_1 model, the next step is to define the fundamental Physics properties.

#### **Physics Settings: Bioheat Equation (htbh)**

Using the menu bar, select Multiphysics > 1 Bioheat Equation (htbh). Select Physics > Subdomain Settings. Select subdomains 2, 3, and 4. Uncheck the Active in this domain check box. See Figure 10.50. Click the Apply button.

Subdomains Groups		Physics Init	Element Color
Subdomain selection	Thermal properties a	nd heat sources/sinks	
1	Library material:	\$	Load
3	Quantity	Value/Expression	Unit Description
*	🕑 k (isotropic)	0.5	W/(m·K) Thermal conductivity of tissue
	k (anisotropic)	0.5 0 0 0.5	W/(m·K) Thermal conductivity of tissue
	ρ <sub>b</sub>	0	kg/m <sup>3</sup> Density of blood
	C <sub>b</sub>	0	J/(kg·K) Specific heat of blood
	ω <sub>b</sub>	0	1/s Blood perfusion rate
Group:	ть	310.15	K Arterial blood temperature
Select by group	Q <sub>met</sub>	0	W/m <sup>3</sup> Metabolic heat source
Active in this domain	Q <sub>ext</sub>	0	W/m <sup>3</sup> Spatial heat source

FIGURE 10.50 2D\_Bio\_MCT\_1 model Subdomain Settings (2, 3, 4) edit window

Select subdomain 1. In the subdomain 1 edit window, enter the information as shown in Table 10.9. Click OK after filling in the parameters of the rectangle in the Rectangle edit window.

**NOTE** The metabolic energy  $(Q_{met})$  is sufficiently small, relative to the microwave energy, that it can be ignored in this model. Thus it is set to zero.

Click the Apply button. See Figure 10.51. Click OK.

Table 10.9	Subdomain	1	Settings
------------	-----------	---	----------

Name	Setting
k (isotropic)	k_liver
$ ho_b$	rho_blood
C <sub>b</sub>	C_blood
$\omega_{b}$	omega_blood
Т <sub>b</sub>	T_blood
Q <sub>met</sub>	0
Q <sub>ext</sub>	Qav_rfwh

FIGURE 10.51 2D\_Bio\_MCT\_1 model Subdomain Settings (1) edit window

#### **Physics Boundary Settings: Bioheat Equation (htbh)**

Having established the bioheat equation subdomain settings for the 2D\_Bio\_MCT\_1 model, the next step is to define the bioheat equation physics boundary settings.

Using the menu bar, select Physics > Boundary Settings. Select boundary 1. Check the Select by group check box.

Select "Thermal insulation" from the Boundary condition pull-down list. See Figure 10.52. Click OK.

Boundaries Groups	_	Coefficients	Color/Style	
Boundary selection	Boundary sources an	d constraints		
2	Boundary condition:	Thermal insulation	4	
3	Quantity	Value/Expression	Unit	Description
4	u <sub>0</sub>	0	W/m <sup>2</sup>	Inward heat flux
6	h	0	W/(m <sup>2</sup> K)	Heat transfer coefficien
78	T <sub>int</sub>	273.15	к	External temperature
Group:	το	310.15	к	Temperature
Select by group				
Interior boundaries				

FIGURE 10.52 2D\_Bio\_MCT\_1 model Boundary Settings (1) edit window

Name	Expression	Unit	Description
epsilon0_rfwh	8.854187817e-12	F/m	Permittivity of vacuum
mu0_rfwh	4*pi*1e-7	H/m	Permeability of vacuum
nu_rfwh	nu	Hz	Frequency
Synchronize	equivalent variables		
Synchronize	equivalent variables		

FIGURE 10.53 2D\_Bio\_MCT\_1 model Application Scalar Variables

Note The thermal insulation boundary condition can be employed because most of the heat energy is removed by perfusion, rather than by conduction through the boundaries.

### Physics Settings: 2 TM Waves (rfwh), Scalar Variables

Using the menu bar, select Multiphysics > 2 TM Waves (rfwh). Using the menu bar, select Physics > Scalar Variables. Enter nu in the nu\_rfwh (Frequency) edit window. See Figure 10.53. Click OK.

#### Physics Settings: 2 TM Waves (rfwh), Subdomain Settings

Using the menu bar, select Physics > Subdomain Settings. In the Subdomain Settings edit window, enter the information shown in Table 10.10 and Figures 10.54 through 10.57. Click the Apply button after filling in the parameters of each separate subdomain in the Subdomain settings edit window. Click OK.

## Physics Settings: 2 TM Waves (rfwh), Boundary Settings

Using the menu bar, select Physics > Boundary Settings. In the Boundary Settings edit window, enter the information as shown in Table 10.11. Click the Apply button after choosing or entering the parameters of each Boundary Settings group in the Boundary Settings edit windows. Click OK. See Figures 10.58–10.62.

Name	Subdomain 1	Subdomain 2	Subdomain 3	Subdomain 4
$\epsilon_r$ (isotropic)	eps_liver	eps_cat	eps_diel	1
$\sigma$ (isotropic)	sig_liver	0	0	0
$\mu_r$	1	1	1	1

#### Table 10.10 Subdomain Settings

Boundary	Boundary Condition	Wave Type	Value	Figure Number
1, 3	Axial symmetry	_	_	10.58
2, 14, 18, 20, 21	Scattering boundary condition	Spherical	_	10.59
5–7, 9, 11–13, 15, 17	Perfect electric conductor	_	_	10.60
8	Port	Wave excitation selected	P_in	10.61
8	Port tab	_	Coaxial	10.62

## Table 10.11 Boundary Settings

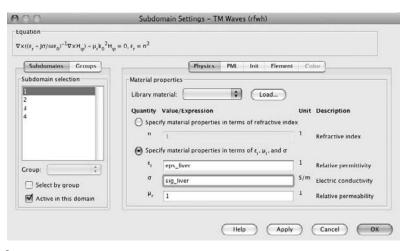


FIGURE 10.54 2D\_Bio\_MCT\_1 model, Subdomain Settings (1) edit window

Subdomains Groups	Physics PML Init	Element Color
ubdomain selection	Material properties Library material: Library material: Loc Quantity Value/Expression O Specify material properties in terms of refra n 1 Ο Specify material properties in terms of ε <sub>r</sub> , μ <sub>r</sub>	1 Refractive index
Group:	ε <sub>r</sub> eps_diel σ 0	1         Relative permittivity           S/m         Electric conductivity
Active in this domain	μ <sub>r</sub> 1	1 Relative permeability

FIGURE 10.56 2D\_Bio\_MCT\_1 model, Subdomain Settings (3) edit window

## **Mesh Generation**

Using the menu bar, select Mesh > Free Mesh Parameters. Click the Custom mesh size radio button. Enter 3e-3 in the Maximum element size edit window. Click the Apply button. See Figure 10.63.

Subdomains Groups	Physics PML Init	Element Color
iubdomain selection 1 2 3 4	Material properties	Unit Description ctive index
Group: *	$\begin{bmatrix} \varepsilon_r & 1 \\ \sigma & 0 \\ \mu_r & 1 \end{bmatrix}$	1     Relative permittivity       S/m     Electric conductivity       1     Relative permeability

FIGURE 10.57 2D\_Bio\_MCT\_1 model, Subdomain Settings (4) edit window

Boundaries Groups	Conditions Material Properties Port Far-Field Color/Style	D.
oundary selection	Boundary sources and constraints	
	Boundary condition: Axial symmetry	
2		
· U		
0		
-		
12 13		
roup:		
Select by group		
Interior boundaries		

I FIGURE 10.58 2D\_Bio\_MCT\_1 model Boundary Settings (1, 3) edit window

00	Boundary Settings – TM Waves (rfwh)
Equation	
n × E = 0	
Boundaries Groups	Conditions Material Properties Port Far-Field Color/Style
Boundary selection	Boundary sources and constraints
7 8	Boundary condition: Perfect electric conductor
9	
10	
12	
15 14	
15	
16	
18	
19	
Group:	
Select by group	POWEREN.IR
Interior boundaries	
	Help Apply Cancel OK
	(help) (Apply) (cancel) (OK

**FIGURE 10.60** 2D\_Bio\_MCT\_1 model Boundary Settings (5–7, 9, 11–13, 15, 17) edit window

Boundaries Groups	Conditions Material Properties Port Far-Field Color/Style
Boundary selection	Port definition
7	-
8	Mode specification: Coaxial
9	
10	
11	
12	
13	
14	
15	
16	
17	
18 19	
19	
Group:	
Select by group	
Interior boundaries	

**FIGURE 10.62** 2D\_Bio\_MCT\_1 model Boundary Settings (8), Port tab edit window

Predefined mesh sizes: Custom mesh size Maximum element size:	Adomain Boundary Point Advanced	Can
• Custom mesh size		
Maximum element size:	34-3	Apr
	30-3	
	Je-J	
Maximum element size scali	ing factor: 1	Не
Element growth rate:	1.3	
Mesh curvature factor:	0.3	
Mesh curvature cutoff:	0.001	
Resolution of narrow regior	ns: 1	
Optimize quality of the second	Jlar 🗘	

**FIGURE 10.63** 2D\_Bio\_MCT\_1 model Free Mesh Parameters, Global tab

000		Free Mesh Parameters	
	Global St	abdomain Boundary Point Advanced	ОК
1 2 3 4	ain selection	Subdomain mesh parameters Maximum element size: 1.5e~4 Element growth rate: Method: Triangle ♦	Cancel Apply Help
Sele	ect Meshed	Remesh Mesh Selected	

FIGURE 10.64 2D\_Bio\_MCT\_1 model Free Mesh Parameters, subdomain 3

Click the Subdomain tab. Select subdomain 3. Enter 1.5e-4 in the Maximum element size edit window. See Figure 10.64.

Click the Remesh button, and then click OK. See Figure 10.65.

#### Solving the 2D\_Bio\_MCT\_1 Model

Using the menu bar, select Solve > Solver Parameters. Select "Parametric" from the solver list. Enter P\_in in the Parameter name edit window. Enter 2:0.5:10 in the Parameter values edit window. Click the Apply button. See Figure 10.66.

Click OK. Using the menu bar, select Solve > Solve Problem.

#### Postprocessing and Visualization

The default plot shows a surface plot of the temperature (K). See Figure 10.67.

Typically, such analytical plots are viewed in degrees Centigrade. The plot can be easily converted through the following steps. Using the menu bar, select Postprocessing > Plot Parameters > Surface. Select "degC (°C)" from the Unit pull-down list. See Figure 10.68.

Click OK. See Figure 10.69.

Note: The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. The guiding principle needs to be that tumor cells die at elevated temperatures. The literature cites temperatures that range from 42 °C (315.15 K) to 60 °C (333.15 K).<sup>6</sup> If the postulated method raises the local temperature of

## **FIGURE 10.65** 2D\_Bio\_MCT\_1 model mesh

000	Solver Par	rameters		
Analysis	General Pa	rametric Stationary	Adaptive	Advanced
Stationary 🗘	Parameter			
Auto select solver		-		
Solver:	Parameter name:	P_in		
Stationary	Parameter values:	2:0.5:10		
Time dependent				
Eigenvalue	Linear system solver			
Parametric Stationary segregated	Linear system solver:	Direct (UMFPACK)	4	
Parametric segregated	Preconditioner:		\$	
	Preconditioner.		( ¥ )	
Adaptive mesh refinement	Matrix symmetry:	Automatic	•	Settings
		Help App	ity) (Ca	ncel (

**FIGURE 10.66** 2D\_Bio\_MCT\_1 model Solver Parameters

FIGURE 10.67 2D\_Bio\_MCT\_1 model solution, temperature (K)

the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful.

Now that the 2D\_Bio\_MCT\_1 model has been successfully calculated, the modeler can determine the temperature for a preliminary estimate of the input power. Final determination of the veracity of the calculation will, of course, need to be made experimentally. The results (solutions at different input powers) from the model calculations will significantly reduce the effort needed to determine an accurate initial experimental value.

It can readily be seen in the solution of the 2D\_Bio\_MCT\_1 model at 10 W that the peak temperature in the region immediately adjacent to the antenna may be higher (approximately 100 °C) than desired. The range of solutions for powers from 2 W to 10 W is easily viewed for selection. Using the menu bar, select Postprocessing >

General Surf	ace Contour	Boundary	Arrow	Principal
Surface plot				
	Surface Data	Height Data		
Predefined quanti	ities: Temperature		Ran	ge
Expression:	Т		Sm	ooth
Unit:	°c		_	
Unit:	<u> </u>		9	
oloring and fill				
	terpolated 🛟	Fill style:	Filled	\$
oloring:		Fill Style.	rineu	•
urface color				
Colormap:	jet 🗘	Colors: 1024	Col	or scale
		Colors: 1024	Col	or scale
2 H		Colors: 1024	Col	or scale
20 N		Colors: 1024	Col	or scale
2 10		Colors: 1024	Col	or scale
2000		Colors: 1024	Col	lor scale
2 10		Colors: 1024	Col	ior scale
2000		Colors: 1024	Col	or scale
20 N		Colors: 1024	Col	or scale
2000		Colors: 1024	Col	or scale
2000		Colors: 1024	Col	or scale
20 N		Colors: 1024	Col	or scale
<ul> <li>Colormap:</li> <li>Uniform color</li> </ul>		Colors: 1024	Col	or scale
20 N		Colors: 1024	Col	or scale
20 N		Colors: 1024	Col	or scale

FIGURE 10.68 2D\_Bio\_MCT\_1 model Plot Parameters, Surface tab (°C)

Cross-Section Plot Parameters > General. Select all of the solutions in the Solutions to use selection window. See Figure 10.70.

Click the Line/Extrusion tab. Select "degC (°C)" from the Unit pull-down list. Select "r" from the x-axis data pull-down list. Enter r0 = 0, and r1 = 0.03. Enter z0 = 0.02, and z1 = 0.02. See Figure 10.71.

Click the Apply button, and then click OK. See Figure 10.72.

Using the cross-section graph, an appropriate power/temperature/distance profile can be chosen for the desired therapy.

**NOTE** The plot lines of temperature on the cross-section graph are arranged in order of ascending power vertically. As more power is supplied to the tissue, more heat is

**FIGURE 10.69** 2D\_Bio\_MCT\_1 model, surface temperature (°C)

FIGURE 10.71 2D\_Bio\_MCT\_1 model Cross-Section Plot Parameters, Line/Extrusion tab

**FIGURE 10.72** 2D\_Bio\_MCT\_1 model, temperature (*T*) as a function of applied power vs. radius (*r*)

dissipated in the tissue; accordingly, the temperature rises. The graphical plots start at 2 W and ascend to 10 W in 0.5-W increments.

## 2D Axisymmetric Microwave Cancer Therapy Model: Summary and Conclusions

The bioheat equation is a valuable approach for calculating the efficacy of potential treatment methodologies. Now that the 2D\_Bio\_MCT\_1 model has been successfully calculated, the modeler can determine the power needed to reach the desired temperature for a preliminary estimate of an effective treatment. Final determination of the veracity of the calculation will, of course, need to be made experimentally. The literature cites temperatures that range from 42 °C (315.15 K)<sup>5</sup> to 60 °C (333.15 K).<sup>6</sup>

If the postulated method raises the local temperature of the tumor cells, without excessively raising the temperature of the normal cells, then the proposed method will probably be successful. The results (estimated power values) from the model calculations will significantly reduce the effort needed to determine an accurate experimental value. The guiding principle needs to be that tumor cells die at elevated temperatures.

#### References

- 1. H. H. Pennes, J. Appl. Physiology, Vol. 1, No. 2, August 1948, pp. 93–122.
- 2. http://en.wikipedia.org/wiki/Bioheat\_transfer
- 3. http://medical-dictionary.thefreedictionary.com/perfusion
- L. R. Hirsch et al., "Targeted Photothermal Tumor Therapy using Metal Nanoshells", Proceedings of the Second Joint EMBS/BMES Conference, Houston, TX, USA, October 23–26, 2002, pp. 530–531.
- L. R. Hirsch et al., Proceedings of the 25th Annual International Conference of the IEEE EMBS, Cancun, Mexico. September 17–21, 2003.
- K. Saito et al., "Localized Heating by the Coaxial-Dipole Antenna for Microwave Coagulation Therapy", Antennas, Propagation, and EM Theory, 2000. *Proceedings of ISAPE*, 2000, Beijing, China, 5<sup>th</sup> International Symposium.
- D. P. O'Neal et al., "Photo-thermal tumor ablation in mice using near infraredabsorbing nanoparticles", Cancer Letters 209 (2004), pp. 171–176.
- 8. http://www.cancer.gov/cancertopics/factsheet/Therapy/hyperthermia
- 9. http://en.wikipedia.org/wiki/Ohm%27s\_law
- 10. http://en.wikipedia.org/wiki/Joule%27s\_Law
- 11. http://en.wikipedia.org/wiki/Minimally\_invasive

## Exercises

- 1. Build, mesh, and solve the 2D axisymmetric tumor laser irradiation model problem presented in this chapter.
- 2. Build, mesh, and solve the 2D axisymmetric microwave cancer therapy model problem presented in this chapter.
- 3. Explore other receptor materials as applied in the 2D axisymmetric tumor laser irradiation model.
- 4. Explore other materials as applied in the 2D axisymmetric microwave cancer therapy model.
- 5. Explore the different geometries in the 2D axisymmetric tumor laser irradiation model.
- 6. Explore the different geometries in the 2D axisymmetric microwave cancer therapy model.
- 7. Explore the different tissues in the 2D axisymmetric tumor laser irradiation model.
- 8. Explore the different tissues in the 2D axisymmetric microwave cancer therapy model.

# Index

Italicized page locators indicate a figure; tables are noted with a t.

## Α

Absolute tolerance edit window, 559 2D AC Generator model: transient, 523 AC, intrinsic nature of, 495 AC/DC Module, 221, 572 Hall effect models and, 171 mixed-mode models and employment of. 460 AC/DC Module Model Library, 464 AC/DC Module Motors and Drives Library Model, 498, 531 AC/DC Module Small In-Plane Currents Application Mode, 463 AC electrical power generation and distribution systems, development and commercialization of, 494 AC induction, 453 AC power transmission system transformed. 497 untransformed, 497 AC realm expanding modeling calculations from. 462 skin depth and, 463 AC theory. See Alternate current (AC) theory AC voltage, resistive/reactive vector phase diagram, 462 ALE. See Arbitrary Lagrangian-Eulerian Algebraic multigrid, 618, 619, 630 Alternate current (AC) theory, electrical impedance in, 461–462 Alumina, 2D Resistive Heating 2 and introduction of. 366

"Analysis of Tissue and Arterial Blood Temperatures in the Resting Human Forearm" (Pennes), 747 Angular transform, rotational sense of, 4 Animation. See also Postprocessing animation solution presented as, 75 Animation Plot Parameters window 2D Hall Effect 1 model, 187 2D Hall Effect 2 model, 204 2D Hall Effect 3 model, 221 Anisotropic conductivity, 192, 210, 221 Antenna and model domain, 2D axisymmetric microwave cancer therapy model, 779 Application Mode, defined, 5 Application Mode Name window two mode names in, 395, 413 2D axisymmetric Inductive Heating 2 model, second variation on, 435 Application Mode Properties: Perpendicular Induction Currents, Vector Potential (emqa), 2D AC Generator Sector model, static, 542, 543 Application Mode Properties button, 172,772 Application Mode Properties dialog box, 2D AC Generator model, static, 514 Application Mode Properties edit window 2D concave mirror model, without PMLs, 729 2D concave mirror model, with PMLs, 715 2D dielectric lens model, without PMLs. 696 2D dielectric lens model, with PMLs, 682

Application Mode Properties window 2D Hall Effect model, 172 2D Hall Effect model, first variation on, 188 2D Hall Effect model, second variation on, 205 Application Scalar Variables edit window 2D axisymmetric Inductive Heating 1 model. 399 2D axisymmetric Inductive\_Heating\_2 model. 420 2D axisymmetric Inductive Heating 3 model, second variation on, 441 2D concave mirror model, without PMLs. 730 2D concave mirror model, with PMLs, 715 2D dielectric lens model, without PMLs, 697 2D dielectric lens model, with PMLs, 683 2D electric impedance sensor model: advanced, 484 2D electric impedance sensor model: basic. 469 Application Scalar Variables window, 2D axisymmetric microwave cancer therapy model, 782 Approximation models, first, most meaningful, 37 - 38Arbitrary Lagrangian-Eulerian, 116, 117 Asperity(ies), 145 second variation of 2D electrochemical polishing model, 147 change in position of electrode surface and removal of material from, 164 2D. on electrode. 116 2D electropolishing 1 model electrode with, 122.135 2D electropolishing\_3 model electrode with. 154 avi extension, for saved animation, 75 Axial position space coordinates, 227 Axial Symmetry coordinate system (1D and 2D), 5 Axis edit window, 2D Axisymmetric Inductive Heating 1 model, 395.395*t* Axisymmetric geometry (cylindrical) modeling, 229

Azimuthal Induction Currents, Vector Potential (emqa) physics boundary settings 2D Axisymmetric Inductive\_Heating\_2 model, 423 2D Inductive\_Heating\_3 model, second variation on, 442, 445 physics subdomain settings 2D Axisymmetric Inductive\_Heating\_2 model, 422 2D Inductive\_Heating\_3 model, second variation on, 441–442

## В

Behavior of model, accurate anticipation of, 64 Berenger, Jean-Pierre, 671 **Bioheat** equation conduction heat equation and, 748 guiding principle behind, 749, 765, 770, 788.794 laser beam energy and, 749 physics boundary settings, 2D axisymmetric tumor laser irradiation model, 760-761 physics boundary settings, 2D axisymmetric microwave cancer therapy model, 781 physics settings, 2D axisymmetric microwave cancer therapy model, 779-780 physics subdomain settings, 2D axisymmetric tumor laser irradiation model, 758, 760 value of, 768-769, 794 Bioheat Equation Application Mode, bioheat equation formulation in, 747–748 Bioheat equation theory, 747–749 Bioheat modeling, defining, 747 Bioheat models, 747–794 2D axisymmetric microwave cancer therapy model, 770–794 2D axisymmetric tumor laser irradiation model, 749–769 Biomagnetic studies, magnetostatic modeling applied to, 649 Bismuth, as 2D axisymmetric Inductive Heating 2 model, second variation, material of choice, 434

Bismuth subdomain 3, 2D axisymmetric Inductive Heating 3 model, second variation on. 440 BLK1 edit window, 3D thin layer resistance modeling: thin layer subdomain, 594 BLK2 edit window 3D thin layer resistance model: thin layer approximation, 578, 579 3D thin layer resistance model: thin layer subdomain. 595 BLK3 edit window, 3D thin layer resistance model: thin layer subdomain, 595 Block edit window, 593 Boltzmann, Ludwig, 228 Boundary check box, 606, 619 Boundary conditions COMSOL 1D telegraph equation model, 93 first variation on COMSOL 1D telegraph equation model. 99 second variation on COMSOL 1D telegraph equation model, 106 settings Kdv equation model, 70-71 specifications of, 71 Boundary Data Expression edit window, 647 Boundary Integration edit window 2D axisymmetric Thermos Container 1 model. 284 2D axisymmetric Thermos\_Container\_2 model, 300 2D axisymmetric Thermos Container 3 model. 315 Boundary Pairs Value edit window, 3D thin layer resistance model: thin layer approximation, 588 Boundary plot, 3D thin layer resistance model: thin layer approximation, 591 **Boundary Settings** Conditions page, 2D AC Generator Sector model, static, 548 Conductive Media DC (dc) edit window, 2D Resistive Heating 1 model, 337t General Heat Transfer Edit window, 2D axisymmetric cylinder conduction model, first variation on, 246t

general heat transfer edit window, 2D axisymmetric heat conduction model, 235t In Plane Electric Currents (emqvw) edit window, 2D electric impedance sensor model: basic, 471 1D single-pane heat flow model and, 10–11 1D triple-pane, 32t physics settings: 2 TM waves (rfwh), 2D axisymmetric microwave cancer therapy model, 782 3D electrostatic potential between two cylinders, 616t 3D electrostatic potential five cylinders model, 627*t* 2D axisymmetric microwave cancer therapy model, 781, 783t, 785, 786, 787 2D axisymmetric tumor laser irradiation model. 760t 2D concave mirror model, without PMLs, 732 2D concave mirror model, with PMLs, 719 2D dielectric lens model, without PMLs, 699 2D dielectric lens model, with PMLs, 686 2D Hall effect model, 176–180 first variation on. 194–197 second variation on, 211–214, 211t Weak Constr. page, 2D AC Generator Sector model, static, 548 Boundary Settings, Conductive Media DC (dc) window 2D electrochemical polishing model first variation on, 137t second variation on, 156t Boundary Settings, Moving Mesh (ALE) window 2D electrochemical polishing model, 125t first variation on, 140t second variation on, 159, 159t Boundary Settings Boundary selection window, 1D single-pane heat flow model, 11 Boundary Settings-Conductive Media DC (dc) edit window 3D thin layer resistance model: thin layer approximation, 586t 3D thin layer resistance model: thin layer subdomain, 601t 2D Resistive Heating 2 model, 359

- Boundary Settings—Conductive Media DC (dc) edit window, pairs, 3D thin layer resistance model: thin layer approximation, 586t
- Boundary Settings—Conductive Media DC (emdc) edit window, 2D Resistive\_ Heating\_3 model, second variation on, 379
- Boundary Settings edit window 3D electrostatic potential between five cylinders model, 628, 629
  - 3D electrostatic potential between two cylinders model, *617*, *618*
  - 3D magnetic field of a Helmholtz coil model, *642*, *643*
  - 3D magnetic field of a Helmholtz coil with a magnetic test object model, 659
  - 3D thin layer resistance model: thin layer approximation, 586, 587
  - 3D thin layer resistance model: thin layer subdomain, 603, 604
  - 2D Axisymmetric Cylinder\_Conduction\_1 model, 237
  - 2D Axisymmetric Inductive\_Heating\_1 model, 401*t*, 402*t*, 404*t*
  - 2D Axisymmetric Inductive\_Heating\_2 model, 423*t*, 426*t*, 427*t*
  - 2D Electric Impedance Sensor model: advanced, 486, 487
  - 2D Electric Impedance Sensor model: basic, 471, 472
  - 2D Inductive\_Heating\_3 model, second variation on, 444*t*, 445, 445*t*, 448*t*, 449, 450
- Boundary Settings—General Heat Transfer edit window
  - 2D Axisymmetric Cylinder Conduction model, second variation on, including vacuum cavity, 259t
  - 2D Axisymmetric Thermos\_Container\_1 model, 277*t*
  - 2D Axisymmetric Thermos\_Container\_2 model, 293*t*, 294*t* second variation on, 309*t*, 310*t*

Boundary Settings—Heat Transfer by Conduction (ht) edit window, 334t 2D Resistive\_Heating\_2 model, 354t 2D Resistive Heating 3 model, second variation on, 375t Boundary Settings—In-Plane Electric Currents (emqvw) edit window, 2D electric impedance sensor model: advanced, 485t Boundary Settings window blank, 1D triple-pane, 31 for boundary 1, filled-in 1D dual-pane, 21 for boundary 1, filled-in 1D triple-pane, 31 for boundary 4, filled-in 1D dual-pane, 22 COMSOL 1D Telegraph Equation model, 93 first variation on, 99t. 100 second variation on. 106, 106t KdV equation model, 71t 12 Dual-pane, 21 2D axisymmetric Tumor Laser Irradiation model, 761, 762 Brain function, 2D electric impedance tomography research on, 460, 493 Breast cancer, 2D electric impedance tomography research and detection of, 460,493

## С

CAD file, imported, 2D AC Generator Sector model. static. 534 Calculated temperature 1D dual-pane analysis and, 26 1D single-pane analysis and, 13 Carnot, Nicolas Leonard Sadi, 228 Carrier types, density of, 168 Cartesian coordinate systems, 5 GUI example of, 3 Centigrade degrees analytical plots viewed in, 788 2D axisymmetric Inductive Heating 2 model and temperature distribution in, 431, 435 2D axisymmetric Inductive\_heating\_3 model and temperature distribution in, 452 2D axisymmetric Tumor Laser Irradiation model and analytical plots viewed in, 763

2D Inductive Heating 1 model and temperature distribution in, 408, 410 2D Resistive Heating 1 model and temperature distribution in, 339, 341, 342 2D Resistive\_Heating\_2 model and temperature distribution in, 361, 364 2D Resistive\_Heating\_3 model and temperature distribution in, 383-384, 386 Circle creation, 2D AC Generator model. static. 501 Circle edit window, 151 3D thin layer resistance model: thin layer approximation, 581 3D thin layer resistance model: thin layer subdomain. 598 2D Axisymmetric Inductive\_Heating\_1 model, 397 2D Axisymmetric Inductive\_Heating\_2 model, 419t 2D Axisymmetric Tumor Laser Irradiation model, 754 2D Dielectric Lens model, without PMLs, 693 2D Dielectric Lens model, with PMLs, 680 2D Resistive\_Heating\_1 model, 330 Circles pasted, 2D AC Generator model, static, 507 rotor created, 2D AC Generator model, static. 508 two newly created, 2D AC Generator model, static, 505 Circles and rectangles created, 2D AC Generator model, static, 502 Coil 2D Axisymmetric Inductive\_Heating\_2 model, 427 2D Inductive Heating 3 model, 446 CO1 (intersection of circle and square) 3D thin layer resistance model: thin layer approximation, 582 3D thin layer resistance model: thin layer subdomain. 599 Color button, 526, 562 Color difference, determining voltage difference between upper/lower surfaces and, 181

Color expression, 1D dual-pane window solution set to use, 26 Color Expression button, 13 1D dual-pane window, 23 Color Range edit window 2D electric impedance sensor model: advanced, 491 2D electric sensor model: basic and selection of. 476 Color select button, 181 Combined Heat Transfer by Conduction (ht) boundary settings, 2D Resistive Heating 2 model, 356 Complex AC theory, 493 Complex impedance, 493 Composite Object edit window, 2D dielectric lens model, without PMLs, 694 Composite objects, 2D AC Generator model, static. 511 Computers, 573 COMSOL AC/DC Module Model Library, 636,652 COMSOL 2D electrochemical polishing model. See 2D electrochemical polishing model COMSOL Heat Transfer Module Model Library, 749, 770 COMSOL KdV equation model. See KdV equation model COMSOL Material Library: searchable materials library Materials/Coefficient Library search and/or selection window, 41, 42 Materials/Coefficient Library search results, 42 Model Navigator, 41 UNS C10100 properties, 43 COMSOL Material Library Module: searchable materials library, 40-43 COMSOL Multiphysics, exported file from PKS-MPD directly imported into, 57, 59-60 COMSOL Multiphysics 3D Scalar Expressions window, with spherical coordinate transform equations entered, 4 COMSOL Multiphysics Electro-Thermal Application Mode, 325

**COMSOL** Multiphysics General Heat Transfer Application Mode Model, 229 COMSOL Multiphysics modelers guidelines for, 1-6 coordinate systems, 2-5 hardware considerations, 1–2 implicit assumptions, 5-6 new, 1D guidelines for, 63-64 coordinate system, 64 1D modeling considerations, 63-64 COMSOL Multiphysics Model Library, 577,613 COMSOL Multiphysics software basic materials libraries in, 40 default interior boundary conditions set in. 36 installing, 2 2D modeling modes in, 226 COMSOL Multiphysics telegraph equation model, 90-92 COMSOL RF Module model library, 676 Concave mirror 2D concave mirror model, without PMLs, 728 2D concave mirror model, with PMLs, 713 Conduction heat equation, bioheat equation and, 748 Conductive Media DC, 165, 221 physics boundary settings 3D thin layer resistance model: thin layer approximation, 586 3D thin layer resistance model: thin layer subdomain. 601 2D Resistive Heating 1 model, 335, 338 physics subdomain settings 3D thin layer resistance model: thin layer approximation, 583-584 3D thin layer resistance model: thin layer subdomain. 600 2D Resistive Heating 1 model, 334 Conductivity Matrix edit window, closing, 175 Conductivity matrix elements, 2D Hall Effect 2 model. 194 Conductivity relation pull-down list, 173 Conductivity type pull-down list, 175 Conservation of energy, 495

Constants 1D telegraph equation model, 93 3D magnetic field of a Helmholtz coil with a magnetic test object model, 652-653 2D AC Generator model, static, 499 2D AC Generator Sector model, static, 532 2D axisymmetric Cylinder Conduction model, second variation on, including a vacuum cavity, 253 2D axisymmetric Cylinder Conduction 1 model. 231 2D axisymmetric Inductive\_Heating\_1 model, 396, 398 2D axisymmetric Inductive Heating 3 model, 437, 439-440 2D axisymmetric microwave cancer therapy model. 772 2D axisymmetric Thermos Container 1 model. 266 2D axisymmetric Thermos Container 2 model, 302 2D axisymmetric Tumor Laser Irradiation model, 750 2D Electric Impedance Sensor model: advanced, 477, 479, 481-482 2D Electric Impedance Sensor model: basic, 464, 466 2D Hall effect model, second variation on, 206 2D Resistive\_Heating\_1 model, 328 2D Resistive\_Heating\_2 model, 347 2D Resistive Heating 3 model, 368 Constants edit window 3D magnetic field of a Helmholtz coil model, 637, 637t with a magnetic test object, 652t, 653 2D AC Generator model Generator Geometry, 500 static, 500, 500t Stator Geometry Circles Creation, 500t 2D AC Generator Sector model, static, 533. 533t 2D axisymmetric Cylinder Conduction model first variation on, 242, 242t second variation on, including a vacuum cavity, 253, 253t

2D axisymmetric Cylinder Conduction 1 model, 231, 231t 2D axisymmetric Inductive\_Heating\_1 model, 397, 397t 2D axisymmetric Inductive Heating 2 model, 413t, 414 2D axisymmetric Inductive Heating 3 model, 438, 438t 2D axisymmetric Microwave Cancer Therapy model, 773, 773t 2D axisymmetric Thermos Container 1 model, 267, 267t 2D axisymmetric Thermos Container 2 model, 287, 288t, 303, 303t 2D axisymmetric Tumor Laser Irradiation model, 751t, 752 2D Electric Impedance Sensor model: advanced, 478t, 479 2D Electric Impedance Sensor model: basic, 465t, 466 2D Electrochemical Polishing model, 118t first variation on. 131t second variation on, 148t 2D Hall effect model. 172 first variation on, 189t second variation on, 206t 2D Hall Effect 3 model, 206 2D Resistive\_Heating\_1 model, 328, 328t 2D Resistive\_Heating\_2 model, 348t, 349 2D Resistive Heating 3 model, 369, 369t Constants specification window, 1D dual pane, 17 Constants window 1D telegraph equation model, 93t first variation on, 99, 99, 105t second variation on, 105–111 Constitutive relationships for the medium, 634 Constraint name column, Periodoc Boundary Conditions window, 67, 67 Contact rectangles 2D Hall effect model, 191 second variation on, 208 Contour check box, 561

Contour lines determining voltage difference between upper/lower surfaces and incremental position of, 181 2D Hall effect model, 181 first variation on, 200 2D Hall\_Effect\_3 model surface distribution plot with, 219 2D Hall Effect 1 model surface voltage distribution plot with, 184 2D Hall Effect 2 model surface voltage distribution with, 201 Contour tab, 181 Coordinate systems, 2–5 2D axisymmetric, 227, 227–228 in 1D models, 64 in 2D models, 114-115 Copper PKS-MPD Composition materials selection page for, 52 PKS-MPD Composition percentage selection added page for, 51 PKS-MPD Composition percentage selection page for, 50 PKS-MPD Composition selection added page for. 50 PKS-MPD Composition selection page for, 49 PKS-MPD Materials selected verification page for, 56 **PKS-MPD** Materials selection Print Preview page 1 for, 54 **PKS-MPD** Materials selection Print Preview page 2 for, 55 **PKS-MPD** Materials selection Properties display page for, 53, 55, 56 **PKS-MPD** Materials selection properties for, 58 **PKS-MPD** Materials selection Tensile Strength display page for, 57 **PKS-MPD** Print Materials selection page for, 52 Copper connecting bars, in Resistive\_Heating\_2 model, 345, 367 Copper file, selection of, as library to be added in PKS-MPD, 59

Copy Mesh button, 2D AC Generator Sector model, static, 553 Corrosion films, 572 Coulomb, Charles-Augustin de, 115, 611 Coulomb gauge, 636 Coupling variables, 2D AC Generator Sector model, static, 542 Create Composite Object intersection (CO1, CO3), 2D AC Generator Sector model, static, 536 intersection (CO2, CO1), 2D AC Generator Sector model, static, 536 Create Composite Object edit window 3D electrostatic potential between five cylinders model, 626 3D electrostatic potential between two cylinders model, 615 2D AC Generator model, static, 503, 504.510 2D axisymmetric Cylinder Conduction model, second variation on, including a vacuum cavity, 256 2D axisymmetric Inductive\_Heating\_2 model, 415, 416, 417, 418, 421 2D axisymmetric Microwave Cancer Therapy model, 775, 777 2D axisymmetric Thermos Container 1 model. 272 2D axisymmetric Tumor Laser Irradiation model. 755 2D Concave Mirror model, without PMLs, 727 2D Concave Mirror model, with PMLs, 712 2D Dielectric Lens model, with PMLs, 680 2D Electric Impedance Sensor model: advanced, 481 2D Resistive Heating 1 model, 331 2D Resistive\_Heating\_2 model, 350 2D Resistive Heating 3 model, 371 Create Composite object result, 2D Electric Impedance Sensor model: advanced. 482 Create Composite Object window 3D thin layer resistance model: thin layer approximation, 581

3D thin layer resistance model: thin layer subdomain. 598 2D Hall Effect 2 model, 191 2D Hall Effect 3 model, 208 Create Pairs button, on Draw toolbar, 536, 537 Create Pairs edit window, 3D thin layer resistance model: thin layer approximation, 584 Cross-section field analysis Slice tab, 665–666 3D magnetic field of a Helmholtz coil model. 648–649 Cross-Section Line Data edit window 2D Hall effect model, first variation on, 202t 2D Hall effect model, second variation on, 217t Cross-Section Line Data parameters, 609t Cross-Section Line Data Parameters, 3D thin layer resistance model: thin layer approximation. 592t **Cross-Section Parameters** General page 2D Hall Effect 1 model, 184 2D Hall Effect 3 model, 219 Line/Extrusion page 2D Hall effect model, 185 Cross-Section Plot, 649, 665 3D magnetic field of a Helmholtz coil model. 651 Cross-section plot line solution plot with, 3D magnetic field of a Helmholtz coil with a magnetic test object model, 669 3D magnetic field of a Helmholtz coil model. 651 **Cross-Section Plot Parameters** General edit window 2D Resistive Heating 1 model, 343 2D Resistive\_Heating\_2 model, 364 2D Resistive Heating 3 model, 386 General tab 2D axisymmetric Microwave Cancer Therapy model, 792 Line/Extrusion tab 2D axisymmetric Microwave Cancer Therapy model, 793

Point edit window 2D Resistive\_Heating\_1 model, 344 2D Resistive Heating 2 model, 365 2D Resistive Heating 3 model, 387 Cross-Section Plot Parameters edit window, 609 3D magnetic field of a Helmholtz coil with a magnetic test object model, 668 3D thin layer resistance model: thin layer approximation, 592 Cross-Section Plot Parameters page, exact voltage difference at any point in model and, 183 Crucible coil and, 2D axisymmetric Inductive\_Heating\_2 model, 420 coil with rectangle and, 2D axisymmetric Inductive Heating 3 model, 439 heated, inductively produced heating applied to, Inductive\_Heating\_2 model, 454-455 2D axisymmetric Inductive Heating 2 model. 418 Cylinder conduction, 229 Cylinder conduction model, 228 Cylinder CYL1 edit window, 614 3D electrostatic potential between five cylinders model, 624 Cylinder CYL2 edit window, 614 3D electrostatic potential between five cylinders model, 624 Cylinder CYL3 edit window, 3D electrostatic potential between five cylinders model. 625 Cylinder CYL4 edit window, 3D electrostatic potential between five cylinders model, 625 Cylinder CYL5 edit window, 3D electrostatic potential between five cylinders model. 625 Cylinder rectangle 2D axisymmetric Cylinder\_Conduction\_1 model. 232 2D axisymmetric Cylinder\_Conduction\_2 model, 243 Cylindrical coordinates, 316

## D

Darcy's law, 577, 592 DC, intrinsic nature of, 494, 495 DC electrical power generation and distribution systems, Edison and development/commercialization of. 494 DC power transmission system, 496 DC realm, expanding modeling calculations from. 462 Default values, 1D dual-pane window solution plotted with, 24 Deformed mesh. 165 inverted mesh element warning and, 144, 163 Density, heat conduction calculation and, 234 Dependent variables case changes, 2D AC Generator Sector model, static, 532 in 3D transient solution model, 573 transient (or time-dependent) models, 460 changes in, 390 de Vries, Gustav, 65 Dewar, Sir James, 265 Dielectric lens, optics principles and, 676 Dielectric lens (CO1), 2D Dielectric Lens model, without PMLs, 695 Difference command, 2D AC Generator model, static, 502, 509 Dirichlet boundary conditions, 71 Domain Plot Parameters, Point tab, 2D axisymmetric Tumor Laser irradiation model, 767 Domain Plot Parameters edit window. 2D AC Generator model. transient. 528 Domain plus PMLs, 2D concave mirror model, with PMLs, 714 Dual-carrier systems, 170 Dual-pane windows comparison of single- and triple-pane windows with, 35t modeling, 16–17 Dust precipitators, 3D electrostatic potential models and, 612, 622, 633

## Ε

Eddy currents discovery of, 392 inductive heating model and, 322 Edison, Thomas Alva, 494 Electric (AC/DC) Materials Properties Library, 513 Electrical impedance theory, 461–464 Electrical impedance tomography, 460 Electrical resistance divider, touch screen and. 574 Electrical resistance theory, 573–575 Electrical resistance tomography, defined, 460 Electrical resistivity tomography, 460 Electric currents and solids, history behind our understanding of, 324-325, 391-393 Electric field, z-component 2D Concave Mirror model, without PMLs, 737 2D Concave Mirror model, with PMLs, 723 2D Dielectric Lens model, without PMLs, 703 Electric field plot, z-component, 0.5 m 2D Concave Mirror model, without PMLs, 740 2D Dielectric Lens model, without PMLs, 704, 705 Electric field plot, z-component, 1.0 m 2D Concave Mirror model, without PMLs, 741.742 2D Dielectric Lens model, without PMLs, 705, 706 Electric field plot, z-component, 1.5 m 2D Concave Mirror model, without PMLs, 743.744 2D Dielectric Lens model, without PMLs, 706.707 Electric field plot, z-component, 2D Concave Mirror model, without PMLs, 739 Electricity, history behind science of, 115 Electric potential plot cross-section 3D thin layer resistance model: thin layer approximation, 593, 610 3D thin layer resistance model: thin layer subdomain. 611 Electrochemical polishing (electropolishing) technique, 115, 116, 117, 154 Electromagnetic induction, 493

Electromagnetic wave equation calculations, fundamental difficulties related to, 671 Electromotive force, transformed AC power and, 497 Electronic device structures, fabricating, 203-204 Electronic lock pads, 573 Electrons n-type, in semiconductors, 203 in solids. 168 Electrostatic field mapping, 572 Electrostatic generator, invention of, 611 Electrostatic modeling basics, 611-612 Electrostatic scalar potential (V), relationship to electric field vector  $(\mathbf{E})$ , 611–612 Electrostatics (es) physics boundary settings 3D electrostatic potential between five cylinders model, 627 3D electrostatic potential between two cylinders model, 616 physics subdomain settings 3D electrostatic potential between five cylinders model, 627 3D electrostatic potential between two cylinders model, 616 Electrostatics problems, complexity and difficulty of, 618 Electro-Thermal Application Mode. See COMSOL Multiphysics Electro-Thermal Application Mode Electro-thermal coupling, 455 Ellipse, points added to in 2D Electric Impedance Sensor model: advanced, 482, 483 Ellipse edit window 2D axisymmetric Thermos\_Container\_1 model. 268t 2D Concave Mirror model, without PMLs, 726 2D Concave Mirror model, with PMLs, 711 2D Electric Impedance Sensor model: advanced. 480 Ellipsoid edit window, 3D magnetic field of a Helmholtz coil with a magnetic test object model, 656

EMB1 and BLK1 selected 3D thin layer resistance model: thin layer approximation, 584

3D thin layer resistance model: thin layer subdomain, 601

- Embed edit window
  - 3D thin layer resistance model: thin layer approximation, 582
  - 3D thin layer resistance model: thin layer subdomain, 599
- EMB1 on top of block BLK2, 3D thin layer resistance model: thin layer approximation, 583
- EMB1 on top of block BLK3, 3D thin layer resistance model: thin layer subdomain, 600
- EMF. See Electromotive force
- Equation of continuity, 634
- Errors, common sources of, 37–38
- Expression radio button, 590, 607

Exterior temperature

1D dual-pane analysis, 26, 27

1D single-pane analysis, 13, 15 1D triple-pane window analysis, 34

Extrinsic conduction mode, 204

## F

Faraday, Michael, 115, 493 Faraday's law of induction, 530 FDTD electromagnetic modeling calculations, 671 FEM. See Finite Element Method Finite-difference time-domain electromagnetic modeling calculations. See FDTD electromagnetic modeling calculations Finite Element Method, COMSOL Multiphysics software based on, 8 First approximation result models, 323 First-cut solution, 6 First estimate review of conditions of use, 572 First principles analysis, applying to model definition. 35–37 Fixed-volume impedance difference, in basic 2D Electric Impedance Sensor model, 492

Floating contacts, 171, 196, 208, 211, 221 Fluctuating difference volume, in advanced 2D Electric Impedance Sensor model, 492 Fluctuating parameters, validity of model and. 635 Foucault, Leon, 391, 392 Foucault currents, 392 Fourier, Jean Baptiste Joseph, 229, 326 Fourier's analysis, 462 Fourier's law, 229, 326 Four-wire touch screen technology, 574 Free Mesh Parameters Boundary page, 2D AC Generator Sector model, static, 552 Global tab, 2D axisymmetric Microwave Cancer Therapy model, 787 subdomain, 2D Dielectric Lens model, with PMLs. 687 subdomain 3, 2D axisymmetric Microwave Cancer Therapy model, 788 2D axisymmetric Tumor Laser Irradiation model. 762 2D Concave Mirror model, without PMLs, 733 2D Concave Mirror model, with PMLs, 719 2D Dielectric Lens model, without PMLs, 699 Free mesh parameters, 165, 317, 388 Free Mesh Parameters edit window General tab 3D magnetic field of a Helmholtz coil model, 643 3D magnetic field of a Helmholtz coil with a magnetic test object model, 660 Subdomain (2, 3) tab, 3D magnetic field of a Helmholtz coil with a magnetic test object model, 661 Subdomain (4) tab 3D magnetic field of a Helmholtz coil with a magnetic test object model, 661 Subdomain tab, 3D magnetic field of a Helmholtz coil model, 644 3D electrostatic potential between five cylinders model, 629 3D electrostatic potential between two cylinders model, 618

Free Mesh Parameters edit window (continued) 3D thin layer resistance model: thin layer subdomain. 604 2D AC Generator model, static, 519 2D AC Generator Sector model, static, 552 2D axisymmetric Thermos\_Container\_1 model, 280 2D axisymmetric Thermos\_Container\_2 model. 296 2D axisymmetric Thermos Container 3 model. 312 2D Electric Impedance Sensor model: advanced, 488 2D Electric Impedance Sensor model: basic. 473 Free Mesh Parameters window. 80 2D electrochemical polishing model, 142 KdV equation model, 72 second variation on, 86 2D Resistive Heating 3 model, 382 Free mesh (quad), 2D Electrochemical Polishing model. 143 second variation on, 162

# G

Gauge fixing, 636 Gauge transformation, choosing, 636 Gauss, Johann Carl Friedrich, 611 Gauss's law, 612 General Plot Parameters window, 1D single-pane heat flow model, 12, 14 Generator and power distribution basics, 493–498 Generator sector, created one-eighth (CO3, CO4), 2D AC Generator Sector model, static, 537 Generator Sector Geometry, 2D AC Generator Sector model, static, 532 Geometric assembly (pair creation across a boundary), 529, 568 Geometry 2D Hall effect model, 189 second variation on, 206-207 Geometry Circles Copy, Rotate, and Paste, 2D AC Generator model, static, 506t Creation, 2D AC Generator model, static, 505t Geometry components 3D electrostatic potential between five cylinders model, 623t 3D electrostatic potential between two cylinders model, 613t 3D magnetic field of a Helmholtz coil model. 638t 3D magnetic field of a Helmholtz coil with a magnetic test object model, 654t 3D thin layer resistance model: thin layer approximation, 578t 3D thin layer resistance model : thin layer subdomain, 594t 2D axisymmetric Microwave Cancer Therapy model, 774*t*, 776*t*, 778*t* 2D axisymmetric Tumor Laser Irradiation model. 752t 2D Concave Mirror model, with PMLs, 708t 2D Dielectric Lens model, with PMLs, 677t Geometry edges check box, 662 Geometry modeling 3D electrostatic potential between five cylinders model, 623 3D electrostatic potential between two cylinders model, 613, 615 3D magnetic field of a Helmholtz coil model, 638-639 3D magnetic field of a Helmholtz coil with a magnetic test object model, 653–655 3D thin layer resistance model: thin layer approximation, 578 3D thin layer resistance model: thin layer subdomain, 593, 595-596, 598-600 2D axisymmetric Microwave Cancer Therapy model, 772, 774, 776, 778, 779 2D axisymmetric Tumor Laser Irradiation model, 750, 753–755 2D Concave Mirror model, without PMLs, 725.727 2D Concave Mirror model, with PMLs, 707, 710,712 2D Dielectric lens model, without PMLs, 693-694 2D Dielectric Lens model, with PMLs, 677-678,681

Geometry Rectangles Creation, 2D AC Generator model, static, 502t Geom2 work-plane 3D thin layer resistance model: thin layer approximation, 580 3D thin layer resistance model: thin layer subdomain, 597 Geophysics, electrical resistance tomography and, 460 Germanium, semiconductor carrier types and, 203 Gilbert, William, 115 Global Equations, physics settings, 2D AC Generator Sector model, static, 551 Global Equations edit window, 2D AC Generator Sector model, static, 551, 551t Global Expressions, 2D AC Generator Sector model, static, 538 Global Expressions edit window, 2D AC Generator Sector model, static, 538, 538t, 539 Global variables plot, 2D AC Generator Sector model, transient, 566 Global Variables Plot edit window, 2D AC Generator Sector model, transient, 565 GMRES (iterative solver), 618, 619, 630 Good first approximation, 323, 388, 454, 455, 748 Graphical user interface, 2 Grid edit window, 2D Axisymmetric Inductive\_Heating\_1 model, 395t, 396 Group index designation, floating contact couples and, 176 GUI. See Graphical user interface

## Н

Hall, Edwin, 169
Hall coefficient (R<sub>H</sub>), 170
Hall effect, 115

discovery of, 169

Hall effect model. See also 2D Hall effect model

implicit assumptions in, 170

Hall effect sensor geometry, electron flow, 170
Hall effect sensors

applications of, 171
magnetostatic modeling applied to, 649

Hall effect voltage, 181, 200

Hall voltage  $(V_{\rm H})$ , 169, 170 Hard nonlinear magnetic materials, 529, 568 Hardware, selecting, for successful modeling, 1–2 Heat capacity, heat conduction calculation and, 234 Heat conduction theory, 228-229, 316, 325-326 Heater bar assembly 2D Resistive\_Heating\_2 model, 351 2D Resistive Heating 3 model, 372 Heat flow indicator, 1D triple-pane window with. 36 Heat flux demonstrating, 2D Resistive Heating 1 model, 341 2D Resistive Heating 3 model, 388 Heat transfer, 434. See also Bioheat models general, physics boundary settings 2D axisymmetric Cylinder Conduction model, first variation on, 245 2D axisymmetric Cylinder Conduction model, second variation on, including vacuum cavity, 259 2D Axisymmetric Inductive Heating 2 model, 427 2D axisymmetric Thermos\_Container\_1 model, 277, 279 2D axisymmetric Thermos Container 2 model, 293–294, 296 2D Resistive\_Heating\_3 model, 375 general, physics subdomain settings 2D axisymmetric Cylinder Conduction model, first variation on, 245 2D axisymmetric Cylinder Conduction model, second variation on, including a vacuum cavity, 256 2D axisymmetric Cylinder Conduction 1 model, 233-234 2D Axisymmetric Inductive\_Heating\_2 model, 426 2D axisymmetric Thermos Container 1 model, 274 2D axisymmetric Thermos Container 2 model, 289-290, 305-306 2D Inductive\_Heating\_3 model, 446 2D Resistive Heating 3 model, 371–372

Heat transfer (*continued*) history behind, 228 as important design consideration, 227 modeling, 229 Heat transfer by conduction physics boundary settings 2D Resistive Heating 1 model, 334 2D Resistive Heating 2 model, 353-354 physics subdomain settings 2D Resistive Heating 1 model, 332 2D Resistive Heating 2 model, 352-353 Heat Transfer by Conduction Application Mode, 326, 371 Heat transfer coefficients, 317 Heat transfer coefficient window, 1D single-pane heat flow model. 11 Heat Transfer Module, 228, 316 bioheat equation and, 747 2D axisymmetric Cylinder Conduction model, first variation on, 240 Heat Transfer Module Model Library, 229,265 Heaviside, Oliver, 90, 462 Helmholtz coil, 635, 666 3D magnetic field of a Helmholtz coil model and creation of, 638 Helmholtz coil model, 572 Helmholtz coil pair 3D magnetic field of a Helmholtz coil model. 640 3D magnetic field of a Helmholtz coil model and uniformity of, 649 3D magnetic field of a Helmholtz coil with a magnetic test object model. 656 Helmholtz coil pair, sphere, and magnetic ellipsoid, 3D magnetic field of a Helmholtz coil with a magnetic test object model, 657 Helmholtz coil pair and sphere, 3D magnetic field of a Helmholtz coil model, 641 High-frequency currents, 2D electric impedance sensor models and, 460, 492

Hole, block with, 2D Resistive\_Heating\_1 model, 331 Holes p-type, in semiconductors, 203 in solids, 168 Hyperthermic oncology, 769

## I

Identity Boundary Pairs window, 2D AC Generator Sector model, static, 538 Identity pair, 3D thin layer resistance model: thin layer approximation, 585 Identity-pair-contact resistance approximation, in 3D thin layer resistance model: thin layer approximation, 586–587 Imbalance-offset geometry, 221 Impedance resistance mapped into, 462 single frequency analysis of, 462 Import button, 534 Import CAD Data From File select window, 2D AC Generator Sector model, static. 533 Induced Voltage vs. Time view, 2D AC Generator model, transient, 527, 529 Induction heating equations, 392–394 Inductive heating model, 322 Industrial process imaging, electrical resistance tomography and, 460 Information transmission, measurement of a difference and, 77 Init button, in 1D single-pane heat flow model.9 Initial Conditions window KdV equation model, 71*t* first variation on. 79t second variation on, 85 1D telegraph equation model, 94t first variation on. 101t second variation on, 107t Initialized mesh 1D dual-pane window with air gaps, 23 1D triple-pane window with air gaps, 32 Initialize Mesh button, 125, 235, 338, 360,450

In-Plane Electric Currents (emqvw) physics boundary settings 2D Electric Impedance Sensor model: advanced, 484 2D Electric Impedance Sensor model: basic, 470, 472 physics subdomain settings 2D Electric Impedance Sensor model: advanced, 484 2D Electric Impedance Sensor model: basic, 470 In-Plane TE waves (rfweh) physics application mode properties 2D Concave Mirror model, without PMLs, 727 2D Concave Mirror model. with PMLs, 714 2D Dielectric Lens model. without PMLs, 696 2D Dielectric Lens model, with PMLs, 682 physics application scalar variables 2D Concave Mirror model. without PMLs, 730 2D Concave Mirror model, with PMLs, 714 2D Dielectric Lens model. without PMLs, 697 2D Dielectric Lens model, with PMLs, 683 physics boundary settings 2D Concave Mirror model. without PMLs, 732 2D Concave Mirror model, with PMLs, 718 2D Dielectric Lens model. without PMLs, 698 2D Dielectric Lens model, with PMLs, 686 physics subdomain settings 2D Concave Mirror model, without PMLs, 730 2D Concave Mirror model. with PMLs, 714 2D Dielectric Lens model, without PMLs, 697 2D Dielectric Lens model. with PMLs, 683-685 Interchange source and destination button, 538, 538

Interior temperature 1D dual-pane analysis and, 26, 27 1D single-pane analysis and, 13, 15 1D triple-pane analysis and, 34 Intersection command, 2D AC Generator model, static, 502, 509 Intrinsic conduction mode, 203 Inverted mesh element warning, 144, 163 Ions, in solids, 168

### J

Joule, James Prescott, 228, 324, 391 Joule heating, 322, 388, 434, 454 device design considerations and, 324, 391 modern, importance of, 326, 346, 367, 394, 411, 434 Joule Heating in Conductive Media DC Application Mode, 326, 371 Joule's law, 391, 392, 455, 494, 495, 770

### К

KdV equation, 64-65 in standard notation and formula in COMSOL documentation. 65 KdV equation model boundary conditions settings, 70-71 defining fundamental physics conditions, 67–69 first variation on. 77-83 changing subdomain settings, 77-79 Free Mesh Parameters window, 80 Initial Conditions window, 79 mesh generation, 79 model solution, 82 model solution, Animate page, 83 Open Model window, 78 postprocessing animation, 82–83 Remeshed model. 80 scalar expressions, 77 Scalar Expressions window, 78 Solve Parameters window, 81 Solver Parameters window, Time Stepping page, 81 solving, 79-82 Subdomain Settings window, PDE coefficients, 79t

KdV equation model (continued) Free Mesh Parameters window, 72 initial condition for. 65 Initial Conditions window, 71t KdV model solution, Animate page, 76 KdV model solution Plot Parameters Window, Line page, 76 Meshed model, 72 mesh generation, 72 1D Axis/Grid Settings window (x), 66, 88 1D geometry for, 66, 66-67 periodic boundary condition settings, 67 Periodic Boundary Conditions window, 67 Destination page, 69 Destination page, boundary 2,68 Destination Vertices page, vertex 2, 69 Source page, 67 Source page, boundary 1, variable u2, 69 Source Vertices page, 70 Source Vertices page, vertex 1,68 postprocessing, 75 second variation on, 83-90 Animate page, 89 changing subdomain settings, 85-86 Free Mesh Parameters window, 86 Initial Conditions window, 85t mesh generation, 86 model solution. 89 Open Model window, 84 postprocessing animation, 88 remeshed model, 87 scalar expressions, 84 Scalar Expressions window, 85 Solver Parameters window, 87 Solver Parameters window, Time Stepping page, 88 solving, 86, 88 Subdomain Settings window, PDE coefficients, 85t Solver Parameters window, 73 Solver Parameters window, Time Stepping page, 74 solving, 72, 75 negated KdV model solution, 74 start building, 65–70

subdomain settings, 71 Subdomain Settings window, PDE coefficients, 71*t* summary and conclusions about, 90 viewing solution to, as an animation, 75, 76 Keep interior boundaries check box, 754, 774, 776 2D AC Generator model, 502, 503 static, 509, 510 Kennelley, Arthur E., 462 Kittel, Charles, 168 Korteweg, Diederik, 65

#### L

Lagrange-quartic choice, verifying, 772 Laplace, Pierre-Simon de, 612 Laplacian operator, uses for, 612 Large dimensional fields, 572 Laser beam energy, bioheat equation and, 749 Length semiaxes edit windows, 655 Line, closed polyline (solid) edit window, 535 Line Color Expression window, 13, 15 Line Data edit window, 2D Hall effect model, 185t Line edit window, 2D axisymmetric Microwave Cancer Therapy model, 778 Line/Extrusion plot radio button, 590, 607, 649,665 Line specification window, 1D dual pane, for left pane, 17 Line window, 1D dual-pane window, 25 linspace command, 2D Electric Impedance Sensor model: advanced and solver, 489 Load button, 717, 731 Subdomain Settings page, COMSOL Material Library, 40 Lorentz force, 169, 170 Low dimensionality geometric choices, making, 5 Lung function, 2D electric impedance tomography research on, 460, 493

#### Μ

Magnetic ellipsoid, sphere, and Helmholtz coil pair, 3D magnetic field of a Helmholtz coil with a magnetic test object model, 657 Magnetic field, obtaining graphical plot of, in 3D magnetic field of a Helmholtz coil model, 648-649 Magnetic flux density and magnetic potential, 2D AC Generator model, transient, 528 Magnetic flux density and magnetic potential, z component, 2D AC Generator Sector model, transient, 565 Magnetic vector potential (A) with zcomponent, in static and transient sector-based models, 530 Magnetism, 115 Magnetometers, magnetostatic modeling applied to, 649 Magnetostatic field mapping, 572 Magnetostatic models applications for, 649, 666 basics, 634-635 relationships between potentials and fields, 635 Magnetostatics (emqa) physics boundary settings 3D magnetic field of a Helmholtz coil model. 642 3D magnetic field of a Helmholtz coil with a magnetic test object model, 659 physics subdomain settings 3D magnetic field of a Helmholtz coil model. 641 3D magnetic field of a Helmholtz coil with a magnetic test object model, 658 Mass action law, electron and hole carrier densities and, 204 Materials and databases, 39-61 COMSOL Material Library module: searchable materials library, 40-43 guidelines and considerations, 39-40 MatWeb: searchable materials properties website, 43-47 PKS-MPD: searchable materials properties database, 48-61 Materials/Coefficients Library copper 2D Concave Mirror model. without PMLs. 731 2D Concave Mirror model, with PMLs, 717

edit page, PKS-MPD, 58, 59 load window 2D AC Generator model, static, 515 2D AC Generator Sector model, static, 544 UNS C10100 defined properties, first half, 44 UNS C10100 defined properties, second half. 44 Materials/Coefficients Library window, 2D AC Generator model, static, 514 Materials selected page, PKS-MPD, 60 Materials selected Properties page, PKS-MPD, 61 Materials selection and definition, importance of. 39 Matrix Elements edit window, 175t 2D Hall effect model, first variation on, 192t 2D Hall effect model, second variation on, 210t MatWeb: searchable materials properties website. 43–48 access classes, 43, 45, 45 MatWeb membership level features comparison page, 45 MatWeb metal alloy UNS C10100 selection, 46 MatWeb Metal Alloy UNS Number search selection page, 46 MatWeb properties of UNS C10100, oxygenfree electronic-grade copper, 48 MatWeb search results for UNS C10100, oxygen-free electronic-grade copper, 47 MatWeb selection of UNS C10100, oxygen-free electronic-grade copper, 47 MatWeb selection search types, login home page, 45 Maximum element size, 165, 317, 388 Maximum Element Size edit window, 603, 660, 661, 732, 761, 784 Maxwell, James Clerk, 228, 634 Maxwell's equations, 634 fundamental difficulties related to, 671 Mechanical polishing technique, 116 surface normal vector **n** and current vector **J**, 116.117 Medical imaging, electrical resistance tomography and, 460 Medical studies, magnetostatic modeling applied to, 649

MEMS Module, Hall effect models and, 171 Mesh

- 3D electrostatic potential between five cylinders model, *630*
- 3D electrostatic potential between two cylinders model, *619*
- 3D magnetic field of a Helmholtz coil model, 644
- 3D magnetic field of a Helmholtz coil with a magnetic test object model, *662*
- 3D thin layer resistance model: thin layer approximation, 589
- 3D thin layer resistance model: thin layer subdomain, 605
- 2D AC Generator model, static, 519
- 2D axisymmetric Microwave Cancer Therapy model, 789
- 2D axisymmetric Tumor Laser Irradiation model, 763
- 2D Concave Mirror model, without PMLs, 733
- 2D Concave Mirror model, with PMLs, 720
- 2D Dielectric Lens model, without PMLs, 700
- 2D Dielectric Lens model, with PMLs, 687
- 2D Electric Impedance Sensor model: advanced, 489
- 2D Electric Sensor model: basic, 473
- Mesh Application Mode (ALE), rotation modeling and, in static and transient sector-based models, 531
- Meshed boundaries, copied, 2D AC Generator Sector model, static, 553
- Meshed model, KdV equation model, 72 Mesh generation
  - KdV equation model, 72
    - first variation on, 79 second variation on, 86
    - 1D telegraph equation model, 94 first variation on, 101 second variation on, 108
  - 3D electrostatic potential between five cylinders model, 628
  - 3D electrostatic potential between two cylinders model, 617
  - 3D magnetic field of a Helmholtz coil model, 643

- 3D magnetic field of a Helmholtz coil with a magnetic test object model, 660
- 3D thin layer resistance model: thin layer approximation, 588
- 3D thin layer resistance model: thin layer subdomain, 603–604
- 2D AC Generator model, static, 518
- 2D AC Generator Sector model, static, 552–554
- 2D axisymmetric Cylinder Conduction model, first variation on, 246
- 2D axisymmetric Cylinder Conduction model, second variation on, including vacuum cavity, 261
- 2D axisymmetric heat conduction model, 235
- 2D Axisymmetric Inductive\_Heating\_1 model, 406
- 2D Axisymmetric Inductive\_Heating\_2 model, 430
- 2D axisymmetric Microwave Cancer Therapy model, 784
- 2D axisymmetric Thermos\_Container\_1 model, 279
- 2D axisymmetric Thermos\_Container\_2 model, 296
- 2D axisymmetric Tumor Laser Irradiation model, 761, 763
- 2D Concave Mirror model, without PMLs, 732
- 2D Concave Mirror model, with PMLs, 718
- 2D Dielectric Lens model, without PMLs, 698
- 2D Dielectric Lens model, with PMLs, 686
- 2D Electric Impedance Sensor model: advanced, 488
- 2D Electric Impedance Sensor model: basic, 472
- 2D Electrochemical Polishing model, 125 first variation on, 140 second variation on, 159
- 2D Hall effect model, 180 first variation on, 198, *198* second variation on, 214
- 2D Inductive\_Heating\_3 model, 450
- 2D Resistive\_Heating\_1 model, 338
- 2D Resistive\_Heating\_2 model, 360
- 2D Resistive\_Heating\_3 model, 380

Mesh generator, thin layer approximation and, 577, 592–593 Mesh mapping, 568 Mesh model, 2D Resistive Heating 3 model. 382 Mesh refinement 1D dual-pane heat flow model, 22 1D dual-pane window with air gap, 23 1D single-pane heat flow model, 11, 13 1D triple-pane heat flow model, 32 1D triple-pane window with air gaps, 33 Mesh Remaining (Free) button, on Mesh toolbar, 554 Mesh Selected button, 553, 603, 604, 718, 732, 763 Mesh window 2D axisymmetric Cylinder Conduction model, first variation on, 248 2D axisymmetric Cylinder\_Conduction\_1 model, 237 2D axisymmetric Inductive Heating 1 model, 407 2D axisymmetric Inductive\_Heating\_2 model. 431 2D Inductive Heating 3 model, 451 2D Resistive Heating 1 model, 339 2D Resistive Heating 2 model, 361 Microwave AC, joule heating and, 324 Microwave antenna cross-section, 770 plus tissue, 771 Microwave cancer therapy, 769–770 Mixed boundary conditions, 71 Mixed-materials modeling, 388, 453 Mixed-mode modeling, 321–323, 388, 454. See also 2D complex mixed-mode modeling Model building preparation, materials selection, definition and, 39 Model definition, first principles analysis applied to, 35-37 Modeling errors, common sources of, 37 - 38

Model mesh 2D axisymmetric Thermos Container 1 model, 281 2D axisymmetric Thermos Container 2 model, 297 2D axisymmetric Thermos Container 3 model, 313 Model Navigator Heat Transfer Module, 2D axisymmetric Microwave Cancer Therapy model, 771 initial solution. 2D AC Generator Sector model, transient, 558 initial solution selection, 2D AC Generator Sector model, transient, 558 1D telegraph equation model, 92 RF Module, 2D axisymmetric Microwave Cancer therapy model, 772 Model Navigator command sequence, 677, 707 Model Navigator initial solution, 2D AC Generator model, transient, 523 Model Navigator setup 3D electrostatic potential between five cylinders model, 623 3D magnetic field of a Helmholtz coil model. 636 3D thin layer resistance model: thin layer approximation, 577 3D thin layer resistance model: thin layer subdomain. 594 2D AC Generator model, static, 499 2D AC Generator Sector model, static, 532 2D axisymmetric Cylinder Conduction model first variation on, 242 second variation on, including a vacuum cavity, 252 2D axisymmetric Cylinder Conduction 1 model. 230 2D axisymmetric Inductive Heating 1 model, 394 2D axisymmetric Inductive Heating 2 model, 412 2D axisymmetric Inductive Heating 3 model, 437 2D axisymmetric Thermos Container 1 model. 266

Model Navigator setup (*continued*) 2D axisymmetric Thermos Container 2 model, 287, 303 2D axisymmetric Tumor Laser Irradiation model, 751 2D Concave Mirror model. without PMLs. 725 2D Concave Mirror model, with PMLs, 708 2D Dielectric Lens model, without PMLs, 692 2D Dielectric Lens model, with PMLs, 676 2D Electric Impedance Sensor model: advanced, 478 2D Electric Impedance Sensor model: basic, 465 2D Resistive\_Heating\_1 model, 327 2D Resistive Heating 2 model, 348 Morse code, 77 Move edit window 3D thin layer resistance model: thin layer approximation, 583 3D thin layer resistance model: thin layer subdomain. 600 .mov extension, for saved animation in Power Mac. 75 Moving mesh (ALE), 529, 568 Moving Mesh Application Mode, 2D Electrochemical Polishing model and. 117-118 Multi-coil heating models, 454 Multiphysics General Industrial Applications demonstration model, 464 Multiphysics Model Library, 326, 393 Multiphysics Model Navigator window, 171.188 2D Hall effect model, second variation on, 205

# Ν

NAFEMS collection, 229 Neumann boundary conditions, 71 Newton, Isaac, 228, 325 Newton's law of cooling, 228, 325–326 Nichrome heating bars 2D Resistive\_Heating\_2 model and, 345 2D Resistive\_Heating\_3 model and, 367 Niobium in 2D axisymmetric Cylinder Conduction model, 229 in 2D axisymmetric Cylinder Conduction model, first variation on, 240 in 2D axisymmetric Cylinder Conduction model, second variation on, including a vacuum cavity, 250 Nitrogen, crucible surrounded by, in 2D axisymmetric Inductive\_Heating\_2 model, 434 Nonlinear partial differential equations, role of, 64

# 0

ODE. See Ordinary differential equation Ohm, Georg, 115, 167, 324, 391, 461, 493, 573 Ohm's law, 167, 168, 322, 324-325, 388, 391, 392, 454, 455, 461, 493, 494 microwave cancer therapy theory and, 770 touch screen principle and, 573-575 1D axes/grid settings window, 66 1D Axisymmetric advanced-level modeling, 63 1D beginning-level-moderate-level modeling, 63 1D coordinate system, 5 1D dual-pane heat flow model, 16-27 Boundary Settings window, 21 Constants specification window, 17 dual-pane analysis and conclusions, 26-27 filled-in 1D dual-pane Boundary Settings window for boundary 1, 21 filled in-1D dual-pane Boundary Settings window for boundary 4, 22 Line specification window for air gap, 18 Line specification window for left pane, 17 1D dual-pane window general Plot Parameters window, 24 1D dual-pane window Line window, 25 1D dual pane window solution plotted using default values, 24 1D dual-pane window solution plotted using °F and Color Bar, 27 1D dual-pane window solution set to use color expression, 26

1D dual-pane window solution set to use °F, 25

1D dual-pane window with air gap with initialized mesh, 23 1D dual-pane window with air gap with refined mesh, 23 1D dual-pane workspace after clicking Zoom Extents icon, 19 1D dual-pane workspace showing both panes and air gap, 18 opening workspace, 16 subdomain settings, initial conditions, 20t Subdomain Settings Physics window settings air gap, 20 left pane, 19 right pane, 20 1D KdV equation, solitons, optical fibers and, 65 1D modeling considerations, 63–64 1D modes, with COMSOL Multiphysics software, 63 1D single-pane heat flow model, 6–16 analysis and conclusions, 13, 15 boundary settings entered, 10-11 Boundary Settings window, 10 Filled-in Boundary Settings window for boundary 1, 11 Filled-in Boundary Settings window for boundary 2, 12 1D Constants specification window, 7 1D line specification window, 7 1D single-pane window General Plot Parameters window, 12, 14 1D single-pane window line Plot Parameters window, 12, 15 1D single-pane window solution plotted using default values, 11, 14 1D single-pane window solution plotted using °F and color bar, 13, 16 1D single-pane window with initialized mesh. 13 1D single-pane window with refined mesh, 13 1D Subdomain Settings Physics window settings, 9 1D Subdomain Settings window, 8 opening workspace, 6-7 physics of heat flow through single-pane window, 13, 15

plot presentation changes, 12–13 Subdomain Settings Init window settings, 10 0.005 m line shown in workspace, 8 1D telegraph equation model, 90–111 boundary conditions, 93 Boundary Settings window, 93t constants, 93 Constants window, 93t first variation on, 98–105 boundary conditions, 99 Boundary Settings window, 99t, 100 Constants window, 99, 99t, 105t Initial Conditions window, 101t mesh. 102 mesh generation, 101 model reset, 100 PDE, Coefficient window, Init page, 102 PDE, Subdomain Settings window, PDE coefficients. 101 Plot Parameters window, Animate page, 104 postprocessing, 103 postprocessing animation, 103 pulse amplitude plot, low-loss line, 104 Solver Parameters window, Time Stepping page, 103 solving, 101 subdomain settings, 99–100 Subdomain Settings window, PDE, 100 Initial Conditions window, 94t mesh generation, 94 Model Navigator, 92 1D geometry, 92 PDE, Coefficient window, 94 PDE, Coefficient window, Init window, 95 PDE, telegraph equation model mesh, 95 postprocessing, 96 postprocessing animation, 96, 98 pulse amplitude plot, 97 second variation on. 105–111 boundary conditions, 106 Boundary Settings window, 106, 106t Constants window, 105 Initial Conditions window, 107t mesh generation, 108 model mesh, 108

1D telegraph equation model (continued) model reset. 108 PDE, Coefficient window, Init page, 108 Plot Parameters window, Animate page, 111 postprocessing, 109-110 postprocessing animation, 110 pulse amplitude plot, high-loss line, 111 second variation on, 107 solving, 109 subdomain settings, 106 Subdomain Settings window, PDE coefficients, 107t solution to, 97 Solver Parameters window, Time Stepping page, 96 solving, 94, 96 subdomain settings, 93 Subdomain Settings window, PDE coefficients. 93t summary and conclusions about, 110 telegraph equation geometry window, 92 Telegraph Equation Plot Parameters window, Animate page, 98 1D triple pane heat flow model, 27-35 blank 1D triple-pane Boundary Settings window, 31 boundary settings, 32t Constants specification window, 28 filled-in 1D triple-pane Boundary Settings window for boundary 1, 31 1D triple-pane window Solution plotted using default values, 33 1D triple-pane window solution plotted using °F and Color Bar, 34 1D triple-pane window with air gaps with initialized mesh, 32 1D triple-pane window with air gaps with refined mesh, 33 opening workspace, 28 postprocessing parameters, 34 subdmain settings, initial conditions, 30t Subdomain Settings window, Init settings, 30 Subdomain Settings window, Physics setting, 29 triple-pane analysis and conclusions, 34–35

triple-pane window subdomain settings, 30t triple-pane window workspace lines, 28t workspace after clicking Zoom Extents icon. 29 1D triple-pane window, with heat flow indicator. 36 1D window panes heat flow models, 6-35 1D dual-pane heat flow model, 16–27 1D single-pane heat flow model, 6-16 1D triple-pane heat flow model, 27-35 Opaque thermally conductive materials, 306, 316 Open Model window, 78, 84 Optical coefficient of absorption for laser photons of tumors, 749 Options menu, COMSOL Material Library, 40 Ordinary differential equation, 460, 568

#### Ρ

Paint sprayers, 3D electrostatic potential models and, 612, 622, 633 Paired stator (CO1) and rotor (CO2), 2D AC Generator Sector model, static, 537 Parametric Solver (UMFPACK) solving 2D axisymmetric Cylinder Conduction model, first variation on, 246,249 2D axisymmetric Cylinder Conduction model, second variation on, including vacuum cavity and, 262 2D axisymmetric Cylinder Conduction 1 model and, 236, 238 2D axisymmetric Cylinder Conduction 1p model and, 239 2D axisymmetric Thermos\_Container\_1 model and, 282 2D axisymmetric Thermos Container 2 model and, 298 2D axisymmetric Thermos Container 3 model and. 314 2D Hall effect model and, 180 first variation on, 198 second variation on. 216 Particle accelerators, 3D electrostatic potential models and, 612, 622, 633 PDAs. See Personal digital assistants

PDE

- Coefficient window, Init page, second variation on COMSOL 1D telegraph equation model, *108*
- Subdomain Settings window, PDE coefficients, second variation on COMSOL 1D telegraph equation model, *107*
- Pennes, Harry H., 747
- Pennes's equation, 747
- Perfectly matched layer models, 671–744 function of methodology, 672 theory behind, 671
- 3D Cartesian domain with, 672, 673 3D cylindrical domain with, 672, 674
- 3D spherical domain with, 672, 673
- 2D Cartesian domain with, 672, 672
- 2D Concave Mirror model with, 707–724
- 2D Concave Mirror model without, 724–744
- 2D Dielectric Lens model with, 676–691 2D Dielectric Lens model without, 691–707
- Perfusion rate, defined, 748
- Periodic boundary condition settings, for KdV equation model, 67
- Periodic Boundary Conditions window Destination page, 69 Destination page, boundary 2, 68
- Destination Page, boundary 2, 00 Destination Vertices page, vertex 2, 69
- KdV equation model, 67
- Source page, boundary 1, variable u2, 69
- Source page, KdV equation model and, 67
- Source Vertices page, 70
- Source Vertices page, vertex 1,68
- Periodic Point Conditions Destination page, 2D AC Generator Sector model, static, 550
  - Destination Vertices page, 2D AC Generator Sector model, static, 551
  - Source page, 2D AC Generator Sector model, static, 549
  - Source Vertices page, 2D AC Generator Sector model, static, 550
- Permeability for free space in SI units, numerical value of, 636, 641, 658

Permittivity, AC realm modeling, skin depth and, 463 Permittivity for free space in SI units, numerical value of, 616, 627 Perpendicular Induction Currents, Vector Potential, physics subdomain settings, 2D AC Generator model, static, 513-515 Personal digital assistants, 573 Physical properties, typical coupling of, in developed model, 37 Physics boundary settings Azimuthal Induction Currents, Vector Potential (emga), 2D axisymmetric Inductive\_Heating\_1 model, 401–402 Conductive Media DC (dc) 2D Electrochemical Polishing model, 123 2D Electrochemical Polishing model, first variation on. 137 2D Electrochemical Polishing model, second variation on, 156 2D Resistive Heating 2 model, 358, 360 2D Resistive Heating 3 model, 379 2D AC Generator model, static, 516 general heat transfer 2D axisymmetric Inductive\_Heating\_1 model, 404 2D axisymmetric Thermos Container 1 model, 277 Moving Mesh (ALE) 2D Electrochemical Polishing model, 125t 2D Electrochemical Polishing model, first variation on, 138, 140 Perpendicular Induction Currents, Vector Potential (emqa), 2D AC Generator Sector model, static, 547 Physics settings Periodic Point Conditions, 2D AC Generator Sector model, static, 549–551 2D axisymmetric Inductive Heating 1 model, 398 2D axisymmetric Inductive Heating 2 model. 419 2D axisymmetric Inductive\_Heating\_3 model, 440

Physics subdomain settings Azimuthal Induction Currents, Vector Potential (emga), 2D Axisymmetric Inductive Heating 1 model, 400 Conductive Media DC 2D Electrochemical Polishing model, 123 2D Electrochemical Polishing model, first variation on, 137 2D Electrochemical Polishing model, second variation on, 156 2D Resistive Heating 2 model, 354, 356 2D Resistive\_Heating\_3 model, 377 general heat transfer, 2D axisymmetric Inductive Heating 1 model, 403-404 Moving Mesh (ALE) (ale) 2D AC Generator model, static, 515 2D AC Generator Sector model, static, 542 Perpendicular Induction Currents, Vector Potential (emga), 2D AC Generator Sector model, static, 542–544 PKS-MPD. 48-61 Composition materials selection page for copper, 52 Composition percentage selection added page for copper, 51 Composition percentage selection page for copper, 50 Composition selection added page for copper, 50 Composition selection page for copper, 49 Copper file selected as library to be added, 59 defined properties in, 48 exporting file as text file, 57 main selection page in, 48, 49 Materials/Coefficients Library edit page, 58, 59 Materials selected page, 60 Materials selected Properties page, 61 Materials selected verification page for copper, 56 Materials selection Print Preview page 1 for copper, 54 Materials selection Print Preview page 2 for copper, 55 Materials selection Properties display page for copper, 53, 55, 56

Materials selection properties for copper (UNS C10100) exported, 58 Materials selection Tensile Strength display page for copper, 57 Print Materials selection page for copper, 52 Selection Criteria window in, 48 Planck, Max, 228 Plot Cross-Section Parameters General page, 2D Hall Effect 2 model, 201 Line/Extrusion page, 2D Hall\_Effect\_2 model. 202 Line/Extrusion page, 2D Hall\_Effect\_3 model. 220 Plot Parameters Animate tab 2D AC Generator Sector model, transient. 567 2D AC Generator model, transient, 530 2D axisymmetric Tumor Laser Irradiation model, 768 2D Concave Mirror model, without PMLs, 738 2D Concave Mirror model, with PMLs. 724 2D Dielectric Lens model, without PMLs. 704 2D Dielectric Lens model, with PMLs, 692 animation, final frame, 2D axisymmetric Tumor Laser Irradiation model, 769 Arrow edit window, 2D Resistive\_Heating\_1 model, 342 Contour Data page 2D Hall Effect 3 model, 218 2D Hall effect model, first variation on, 200 Contour tab 2D AC Generator model, transient, 527 2D AC Generator Sector model. transient. 564 General tab, 2D AC Generator model, transient, 525 postprocessing, Surface tab (°C) 2D axisymmetric Microwave Cancer Therapy model, 791 2D axisymmetric Tumor Laser Irradiation model, 765

Surface tab postprocessing, 2D Dielectric Lens model, without PMLs. 702 2D AC Generator model, transient, 526 2D AC Generator Sector model. transient. 563 2D Concave Mirror model, with PMLs, 722 2D Dielectric Lens model, with PMLs, 690 2D axisymmetric Inductive Heating model, first variation on. 436 Plot Parameters edit window Boundary tab 3D thin layer resistance model: thin layer approximation, 590 3D thin layer resistance model: thin layer subdomain. 607 General tab, 3D thin layer resistance model: thin layer subdomain, 606 3D magnetic field of a Helmholtz coil model, 650 2D axisymmetric Inductive\_Heating\_2 model. 434 2D Electric Impedance Sensor model: advanced. 491 2D Electric Sensor model: basic and selection of. 476 2D Inductive Heating 1 model, 409, 410 2D Resistive Heating 1 model, 341 2D Resistive\_Heating\_2 model, 363 2D Resistive Heating 3 model, 385 Plot Parameters selection window Arrow tab, 3D magnetic field of a Helmholtz coil with a magnetic test object model, 667 Boundary tab 3D magnetic field of a Helmholtz coil model. 648 3D magnetic field of a Helmholtz coil with a magnetic test object model. 666 General tab 3D magnetic field of a Helmholtz coil model. 646 3D magnetic field of a Helmholtz coil with a magnetic test object model, 664

Slice tab 3D magnetic field of a Helmholtz coil model. 647 3D magnetic field of a Helmholtz coil with a magnetic test object model, 665 3D electrostatic potential between five cylinders model, 632 3D electrostatic potential between two cylinders model, 621 Plot Parameters window Animate page KdV equation model, first variation on, 83 KdV equation model, second variation on, 89 KdV model solution, 76 1D telegraph equation model, 98 1D telegraph equation model, first variation on, 104 1D telegraph equation model, second variation on. 111 Contour Data page, 2D Hall Effect 1 model, 183 general, 1D dual-pane window, 24 Line page, KdV model solution, 76 1D single-pane heat flow model, 12, 15 2D axisymmetric Cylinder Conduction model, second variation on, including vacuum cavity and, 263 2D axisymmetric Cylinder Conduction 1p model, 240 2D axisymmetric Inductive\_Heating\_3 model, 456 2D axisymmetric Thermos Container 1 model, 283, 285 2D axisymmetric Thermos\_Container\_2 model, 299, 301 2D axisymmetric Thermos Container 3 model. 314 2D Electric Impedance Sensor model: advanced, 493 2D Electrochemical Polishing model, 144 2D Electrochemical Polishing model, second variation on, 14 2D Resistive\_Heating\_1 model, 346 2D Resistive\_Heating\_2 model, 366 2D Resistive Heating 3 model, 389

PML domain, wave equation solution example inside. 675 PML rectangles 2D Concave Mirror model, with PMLs, 710 2D Dielectric lens model, with PMLs, 679 Point constraints, 171 Point edit window 2D Hall effect model, 175t 2D axisymmetric Cylinder Conduction model. 232t. 233 first variation on, 244, 244t second variation on, including a vacuum cavity, 255*t* 2D Electric Impedance Sensor model: advanced. 482 2D Electric Impedance Sensor model: basic, 468 Points, adding to boundary of rectangle, 2D Hall effect model, 173 Polishing, reduction of asperities and, 116 Polyline, closed, 2D generator geometry and, 535 Postprocessing KdV equation model, 75 1D telegraph equation model, 96 first variation on, 103 second variation on, 109-110 Postprocessing animation KdV equation model first variation on. 82 second variation on, 88 1D telegraph equation model, 96, 98 first variation on, 103, 105 second variation on. 110 Postprocessing parameters 1D dual-pane heat flow model, 23 1D single-pane heat flow model, 12 1D triple-pane heat flow model, 34 Postprocessing Plot Parameters, Surface tab, 2D Concave Mirror model, without PMLs, 736 Postprocessing presentation(s), 1D single-pane heat flow model, 12 Preconditioner pull-down list, 619 Predefined quantities pull-down list, 664 Premium Member access, MatWeb, 43, 45, 45, 46

Priestley, Joseph, 115
Primary input power, 495
Proportionality constant, for 2D electropolishing technique, 117
Pryor Knowledge Systems-Materials Properties Database. See PKS-MPD
Pulse amplitude plot
1D telegraph equation model, 97
1D telegraph equation model, high-loss line, second variation on, 111
1D telegraph equation model, low-loss line, first variation on, 104

#### Q

Quadrilateral mesh (quad), 165–166, 317, 388 Quasi-static methodology, 6, 571, 572 Quasi-static models, transient models *vs.*, 321 QuickTime player, free, animations viewed with, 75

## R

Radial position in space, 227 Radius edit window, 639, 655 Rectangle circle and, 2D Resistive Heating 1 model, 330 created, 2D Resistive Heating 3 model, 370 points and physics settings, 2D Electric Impedance Sensor model: basic, 468 2D axisymmetric Cylinder Conduction model, first variation on, 244 rotor created, 2D AC Generator model. static, 509 2D axisymmetric Microwave Cancer Therapy model, 775 2D Electric Impedance Sensor model: advanced. 480 2D Electric Impedance Sensor model: basic. 467 2D Resistive\_Heating\_1 model, 329 Rectangle edit window 2D axisymmetric Cylinder Conduction model. 232 first variation on, 243 second variation on, including a vacuum cavity, 253t

2D axisymmetric Inductive Heating 1 model. 398 2D axisymmetric Microwave Cancer Therapy model, 774, 777, 778 2D axisymmetric Thermos Container 1 model, 268t, 269t 2D axisymmetric Thermos Container 2 model, 289t, 305t 2D axisymmetric Tumor Laser Irradiation model, 750, 752, 753, 755 2D Concave Mirror model, without PMLs, 727 2D Concave Mirror model, with PMLs, 708, 709.712 2D Dielectric Lens model, without PMLs, 694 2D Dielectric Lens model, with PMLs, 677, 677, 678, 680 2D Electric Impedance Sensor model: advanced. 481 2D Electric Impedance Sensor model: basic, 467 2D Hall Effect 2 model, 190, 191t 2D Hall Effect 3 model, 207 2D Resistive Heating 1 model, 329 2D Resistive\_Heating\_2 model, 349t 2D Resistive\_Heating\_3 model, 370t Rectangle geometry 2D Hall Effect 2 model, 190 2D Hall\_Effect\_3 model, 207 Rectangles (R1, R2), 2D axisymmetric Tumor Laser Irradiation model, 753 Refine Mesh button, 125, 214, 235, 238, 338, 360 Registered Member access, MatWeb, 43, 45 Relative Permittivity edit window, 616 Relativistic effects, 6 Remesh button, 141, 160, 261, 472, 518, 644, 661 Remeshed model KdV equation model, first variation on, 80 KdV equation model, second variation on, 87 Reset Model command, 79, 86 Resistance. 168 of homogeneous, isotropic solid material, 168 mapping of, into impedance, 462 Resistance changes, temperature changes and, 391

Resistive heating model, 322 first variation on, 325 second variation on, 325 solving, 325 Resistivity, of homogeneous, isotropic solid material. 169 Revolve edit window 3D magnetic field of a Helmholtz coil model, 639 3D magnetic field of a Helmholtz coil with a magnetic test object model, 655 RF Module, PML technique explicitly available in, 671, 672 Rotary motion, 2D generator model and, 460 Rotor assembly, 2D AC Generator Sector model, static, 536, 538 Rotor created circles, 2D AC Generator model, static. 508 Rotor created rectangles, 2D AC Generator model, static, 509 Rotor Geometry Circles Creation, 2D AC Generator model, static, 508t Rotor Geometry Rectangles Creation, 2D AC Generator model, static, 509t "Run out of memory" problem, 577, 593 Russell, John Scott, 65

# S

Samarium cobalt (radial, inward), addition of, to 2D AC Generator model, static, 515 Samarium cobalt (radial, outward), addition of, to 2D AC Generator model, static, 515 Save As edit window 3D electrostatic potential between five cylinders model, 626 3D electrostatic potential between two cylinders model, 615 3D magnetic field of a Helmholtz coil model. 637 3D magnetic field of a Helmholtz coil with a magnetic test object model, 653 3D thin layer resistance model: thin layer subdomain, 596 2D AC Generator model, Stator Geometry Circles Creation, 501

Save As edit window (continued) 2D AC Generator Sector model, static, 556 2D axisymmetric Tumor Laser Irradiation model. 754 2D Concave Mirror model, without PMLs, 726 2D Concave Mirror model, with PMLs, 711 2D Dielectric Lens model, without PMLs, 693 2D Dielectric Lens model, with PMLs, 679 2D Electric Impedance Sensor model: advanced. 479 2D Electric Impedance Sensor model: basic. 466 Save As window, 2D axisymmetric Microwave Cancer Therapy model, 773 Saving, model building and, 15 Scalar expressions KdV equation model, first variation on, 77 KdV equation model, second variation on, 84 physics settings 2D axisymmetric Tumor Laser Irradiation model, 755 2D Electric Impedance Sensor model: advanced, 483 2D Electric Impedance Sensor model: basic. 468 Scalar Expressions edit window, 399t, 400 2D axisymmetric Inductive Heating 2 model, 421, 421t 2D axisymmetric Inductive\_Heating\_3 model, 441, 441t 2D axisymmetric Tumor Laser Irradiation model, 756, 757 2D Electric Impedance Sensor model: advanced, 483t, 484 2D Electric Impedance Sensor model: basic. 469 Scalar Expressions file, 231, 253 Scalar Expressions window, 78, 85 Scalar variables, physics settings 2D Electric Impedance Sensor model: advanced, 484 2D Electric Impedance Sensor model: basic. 468 2 TM waves (rfwh), 2D axisymmetric Microwave Cancer Therapy model, 782

Scale button, 136, 155 Scale edit window, 2D electropolishing 3 model. 155 Scale factor Auto check box, 647, 664 Scattered electric field, z-component (V/m), 735 solution 2D dielectric lens model, with PMLs, 689 2D dielectric lens model, without PMLs, 701 Scattered electric field, z component (V/m), 2D Concave Mirror model, with PMLs, 721 Scattering boundary condition from Boundary condition pull-down list, 686 outer PML boundary and, 674 Secondary output power, 495 Select by Group check box, 686, 718 Selection Criteria window, PKS-MPD, 41 Select Remaining button, 604, 718, 732 Semiconductors, carrier types in, 168, 203-204 Semiconductor sensors, 170 Set formula edit window, 615 sigma\_Bi\_T, scalar expression for, 441 sigma Cu T, scalar expression for, 441 Signal amplitude, adequate, receiving correct message and, 83 Signal-to-noise ratio, minimum detectable signal and, 77 Silica glass, thermal conductivity of, 17 Silicon, semiconductor carrier types and, 203 Silicon sample, 2D Hall effect model, first variation on, aniosotropic coupling of magnetic field and current flowing in, 192 Silicon wafer. 2D Hall effect model, first variation on, closer approach to construction of specimen as constructed from, 186 Single-coil heating models, 454 Single-pane windows, dual- and triple-pane windows compared with, 35t 60 Hz wavelength calculation, 635 Skin, tissue, and tumor (CO1), 2D axisymmetric Tumor Laser Irradiation model, 756 Skin depth, 493 AC realm modeling and, 463 parameters in 2D Electric Impedance Sensor model: basic, 464

Slice check box, 606 Slice plot, default 3D thin layer resistance model: thin layer approximation, 589 3D thin layer resistance model: thin layer subdomain. 605 Smooth check box, 588 Soft iron (without losses), addition of, to 2D AC Generator model, static, 515 Soft nonlinear magnetic materials, 529, 568 Solid materials, three potential mobile carriers of charge in, 168 Soliton propagation problems, 65 Soliton pulses, splitting of, reduction of argument for initial conditions in KdV model solution and, 82–83 Soliton wave propagation, in diverse media, KdV equations and, 90 Solver and Solver Parameters 1D dual-pane heat flow model, 22 1D single-pane heat flow model, 11 1D triple-pane window, using default values, 33, 33 Solver Manager, Solve For page, 2D AC Generator Sector model, transient, 560 Solver Parameters Advanced edit window 2D axisymmetric Inductive Heating 1 model. 408 2D axisymmetric Inductive\_Heating\_3 model, 452 2D axisymmetric Microwave Cancer Therapy model. 789 2D axisymmetric Tumor Laser Irradiation model, 764 2D Concave Mirror model, without PMLs. 734 2D Concave Mirror model, with PMLs, 720 2D Dielectric Lens model, without PMLs, 700 2D Dielectric Lens model, with PMLs, 688 Solver Parameters edit window Advanced edit window, 2D Axisymmetric Inductive Heating 2 model, 432 3D electrostatic potential between five cylinders model, 631

3D electrostatic potential between two cylinders model, 620 2D AC Generator model static. 520 transient, 524 2D AC Generator Sector model static. 555 transient, 559 2D axisymmetric Cylinder Conduction model, first variation on, 248 2D axisymmetric Cylinder Conduction model, second variation on, including vacuum cavity, 262 2D axisymmetric Cylinder Conduction 1p model and. 239 2D axisymmetric Inductive\_Heating\_1 model, 407 2D axisymmetric Inductive Heating 2 model. 432 2D axisymmetric Thermos Container 1 model, 281 2D axisymmetric Thermos Container 2 model, 297 2D axisymmetric Thermos Container 3 model. 313 2D Electric Impedance Sensor model: advanced, 490 2D Electric Sensor model: basic, 474 2D Inductive Heating 3 model, 451 2D Resistive\_Heating\_1 model, 340 2D Resistive Heating 2 model, 362 2D Resistive Heating 3 model, 383 Solver Parameters window KdV equation model, 73 first variation on, 81 second variation on, 87 Time Stepping page, 74 KdV equation model, first variation on, 81 KdV equation model, second variation on, 88 1D telegraph equation model, first variation on, 103 1D telegraph equation model, second variation on. 109 2D Electrochemical Polishing model, 143 second variation on, 162

Solver Parameters window (continued) 2D Hall Effect 1 model, 182 2D Hall Effect 2 model, 199 2D Hall Effect 3 model, 216 SOR gauge, 636 Space coordinates in 1D model, 64 in 3D transient solution model, 573 in transient solution model, 461 in 2D model, 114, 115 Space dimensions, 6 Specify Object commands, 396 Sphere, Helmholtz coil pair, and magnetic ellipsoid, 3D magnetic field of a Helmholtz coil with a magnetic test object model, 657 Sphere and Helmholtz coil pair, 3D magnetic field of a Helmholtz coil model. 641 Sphere edit window 3D electrostatic potential between five cylinders model, 623, 624 3D electrostatic potential between two cylinders model, 613, 614 3D magnetic field of a Helmholtz coil model. 640 3D magnetic field of a Helmholtz coil with a magnetic test object model, 657 Spherical coordinate systems, 5 Spherical coordinate transform angle rotational sense for phi and, 4 rotational sense for theta and, 5 sqrt argument, 3 Square edit window 3D magnetic field of a Helmholtz coil model, 638, 638, 639 3D magnetic field of a Helmholtz coil with a magnetic test object model, 654, 655 3D thin layer resistance model: thin layer approximation, 581 3D thin layer resistance model: thin layer subdomain. 598 2D Concave Mirror model, without PMLs, 728 2D Concave Mirror model, with PMLs, 713 2D Dielectric Lens model, without PMLs, 695 Static electricity, study of, 611

Static methods, 572 Stationary Solver 2D axisymmetric Cylinder-Conduction\_1 model and, 236 2D Electric Impedance Sensor model: advanced and, 488 2D Electric Impedance Sensor model: basic and, 463 2D Electric Sensor model: basic and, 474 Stator and rotor composite objects, 2D AC Generator model, static. 511 paired, 2D AC Generator model, static, 512 Stator and rotor Create Pairs edit window, 2D AC Generator model, static, 512 Stator assembly, 2D AC Generator Sector model, static, 536, 538 Steady-state model, 6 Steady-state solution to 3D model, parameters and, 572 to 2D axisymmetric models, 324 to 2D coordinate system model, 461 to 2D model, 323 Steady-state value, 2D Resistive Heating 3 model and, 384 Streamline check box, 619 Subdomain edit window 3D thin layer resistance model: thin layer approximation, 585t 3D thin layer resistance model: thin layer subdomain. 601t 2D AC Generator model, static, 516t 2D AC Generator Sector model, static, 544t, 545, 546, 547 2D axisymmetric Cylinder Conduction model, first variation on, 245t 2D axisymmetric Cylinder Conduction model, second variation on, including a vacuum cavity, 258t 2D axisymmetric Cylinder Conduction 1 model, 234*t* 2D axisymmetric Inductive Heating 1 model, 401t, 403, 403t, 404 2D axisymmetric Inductive\_Heating\_2 model, 423*t*, 426*t* 

2D axisymmetric Thermos Container 1 model. 274*t* 2D axisymmetric Thermos Container 2 model, 290t, 291t, 305t, 306t, 308, 308t 2D Electric Impedance Sensor model: advanced, 485t 2D Electric Impedance Sensor model: basic, 470t 2D Inductive Heating 3 model, 442t, 446t, 447, 448 2D Resistive Heating 1 model, 332t Subdomain Expressions physics settings, 2D axisymmetric Tumor Laser Irradiation model, 755, 757 2D AC Generator Sector model, static, 539 Subdomain Expressions edit window 2D AC Generator Sector model, static, 539t, 540, 540t, 541 2D axisymmetric Tumor Laser Irradiation model. 757 Subdomain Integration Variables physics settings, 2D AC Generator model, static. 513 2D AC Generator Sector model, static, 539 Subdomain Integration Variables edit window 2D AC Generator model, static, 513, 513t 2D AC Generator Sector model, static, 540t. 541.542 Subdomain mesh, 165, 317, 388 Subdomain settings changing KdV equation model, first variation on. 77 KdV equation model, second variation on. 85 Electric Parameters edit window 2D axisymmetric Inductive Heating 2 model, 423, 424 2D axisymmetric Inductive Heating 3 model, 442, 443, 444, 445 Initial value edit window, 2D axisymmetric Tumor Laser Irradiation model, 760 Init window, 2D Resistive Heating 1 model, 336

KdV equation model, 71 1D telegraph equation model, 93–94 first variation on, 99-100 second variation on, 106 physics settings: 2 TM waves (rfwh), 2D axisymmetric Microwave Cancer Therapy model, 782 Physics tab subdomain, 2D Concave Mirror model, without PMLs, 730 subdomain 2, 2D Concave Mirror model, without PMLs, 731 subdomain 5, 2D Concave Mirror model, with PMLs, 717 subdomain 5, 2D Dielectric Lens model, with PMLs. 685 subdomain 7, 2D Concave Mirror model, with PMLs, 718 subdomain 7, 2D Dielectric Lens model, with PMLs. 685 subdomain 1, 2D Dielectric Lens model, without PMLs, 697 subdomain 2, 2D Dielectric Lens model, without PMLs, 698 3D magnetic field of a Helmholtz coil model. 641*t* 3D magnetic field of a Helmholtz coil with a magnetic test object model, 658t triple-pane window, 30t triple-pane window, initial conditions, 30t 2D axisymmetric Microwave Cancer Therapy model, 780t, 781, 782t 2D axisymmetric Tumor Laser Irradiation model, 758, 758t, 759 2D Hall effect model, 173, 175 first variation on, 191 second variation on, 208, 210 x absorption, 2D Concave Mirror model, with PMLs, 716 x absorption, 2D Dielectric Lens model, with PMLs, 684 y absorption, 2D Concave Mirror model, with PMLs, 716 y absorption, 2D Dielectric Lens model, with PMLs, 684

Subdomain settings, Conductive Media DC edit window, 355t 2D Resistive\_Heating\_2 model, 356t, 357, 358t 2D Resistive Heating 3 model, 377, 377t, 378 Init edit window 2D Resistive Heating 2 model, 358 2D Resistive Heating 3 model, 379 2D Eletrochemical Polishing model, 123t Subdomain settings, Init edit window 2D axisymmetric Inductive Heating 2 model, 429 2D axisymmetric Inductive Heating 3 model, 449 2D Resistive\_Heating\_1 model, 333 2D Resistive\_Heating\_2 model, 354 2D Resistive Heating 3 model, 375 Subdomain Settings, Moving Mesh edit window 2D AC Generator model, static, 516t 2D AC Generator Sector model, static, 543 Subdomain settings, PML type selection 2D Concave Mirror model, with PMLs, 715 2D Dielectric Lens model, with PMLs, 683 Subdomain settings (2), Electric Parameters edit window, 2D axisymmetric Inductive Heating 1 model, 400 Subdomain Settings edit window 3D electrostatic potential between five cylinders model, 627 3D electrostatic potential between two cylinders model, 616 3D magnetic field of a Helmholtz coil model. 642 3D magnetic field of a Helmholtz coil with a magnetic test object model, 658, 659 3D thin layer resistance model: thin layer approximation, 585 3D thin layer resistance model: thin layer subdomain, 602, 603, 604 2D AC Generator model, static, 516, 517, 518 2D axisymmetric Cylinder Conduction model, 234 2D axisymmetric Cylinder Conduction model, first variation on, 245

2D axisymmetric Inductive\_Heating\_2 model. 428 2D Electric Impedance Sensor model: advanced, 485 2D Electric Impedance Sensor model: basic. 470 2D Resistive\_Heating\_1 model, 333, 336 2D Resistive Heating 2 model, 352t, 353t 2D Resistive Heating 3 model, 373t, 374 Subdomain Settings Physics window settings 1D dual-pane, air gap, 20 1D dual-pane, left pane, 19 1D dual-pane, right pane, 20 Subdomain Settings window Init settings, 1D triple-pane, 30 PDE coefficients KdV equation model, 71t KdV equation model, first variation on, 79t KdV equation model, second variation on. 85 1D telegraph equation model, 93 1D telegraph equation model, first variation on, 100t, 101 1D telegraph equation model, second variation on. 107t Physics settings, 1D triple-pane, 29 2D Electrochemical Polishing model, 124 2D Electrochemical Polishing model, first variation on. 137 Suppress Boundaries selection window 3D electrostatic potential between five cylinders model, 633 3D electrostatic potential between two cylinders model, 621 3D magnetic field of a Helmholtz coil model. 645 3D magnetic field of a Helmholtz coil with a magnetic test object model, 663 Suppress Subdomains 2D Concave Mirror model, with PMLs, 722 2D Dielectric Lens model, with PMLs, 689 Surface check box, 561 Surface temperature (°C), 2D axisymmetric Microwave Cancer Therapy model, 792

Surface voltage distribution plot default, 2D Hall\_Effect\_1 model, *182* default, 2D Hall\_Effect\_2 model, *199* default, 2D Hall\_Effect\_3 model, *217* Symmetry, 2D generator sector model and use of, 460 Sze, S. M., 168

## Т

Tank Export Geometry Objects, Save As window, 2D axisymmetric Thermos\_Container\_1 model, 273 Tank profile, outer, 2D axisymmetric Thermos\_Container\_1 model, 270 Tank(s) components, 2D axisymmetric Thermos\_Container\_1 model, 271 components with lid line, 2D axisymmetric Thermos\_Container\_1 model, 272 2D axisymmetric Thermos\_Container\_1 model, 273 Tank structure creation, 2D axisymmetric Thermos\_Container\_1 model, 270t Telegraph equation, development of, 90 Telegraph equation electrical component model, 90 Temperature calculations 1D dual-pane analysis and, 26-27 1D single-pane analysis and, 13, 15 1D triple-pane analysis and, 34-35 elevations in hyperthermic oncology and, 769 tumor cell death and, 765, 769, 770, 788,794 as function of applied power versus radius, 2D axisymmetric microwave cancer therapy model, 793 Temperature and heat flux, 2D Resistive\_Heating\_1 model, 343 Temperature changes, resistance changes and, 391 Temperature uniformity, 2D axisymmetric Thermos\_Container\_1 model, 277 Temperature vs. time plot

2D Resistive\_Heating\_1 model, 344, 345 2D Resistive\_Heating\_2 model, 365 2D Resistive\_Heating\_3 model, 387 Tesla, Nikola, 494 Thales of Miletus, 611 Thermal conductivity loading, 1D single-pane heat flow model, 9 loading,1D dual-pane heat flow model, 18 Thermal insulation boundary condition, 782 Thermal losses, DC power and, 496 Thermodynamics, heat transfer and science of. 228 Thermos, origin of term, 265 Thermos laminar flow model, 265 Thermos laminar hcoeff model, 265 Thick layers, 572 Thin layer approximation, applications for, 592-593 Thin layer approximation model applications for, 577 in solution of direct current conduction model, 575-576 Thin layer orthogonal voltage dividers, in touch screen technology, 574 Thin layer resistance modeling basics, 575–577 Thin layers, 572 Thin layer technology, touch screen and, 573 Thomson, William (Lord Kelvin), 228 3D Cartesian domain, with PMLs, 672, 673 3D coordinate system, 5 plus time, 573 in steady-state solution to 3D model, 572-573 in transient solution model, 573 3D cylindrical domain, with PMLs, 672, 674 3D electrostatic potential between cylinders models, 572 3D electrostatic potential between five cylinders model, 622-633 Boundary Settings, 627t Boundary Settings (1–4, 23, 24, 28, 29) edit window, 628 Boundary Settings (5-14, 19-22, 25, 26, 31-38) edit window, 628 Boundary Settings (15–18, 27, 30) edit window, 629

3D electrostatic potential between five cylinders model (continued) Create Composite Object edit window, 626 Cylinder CYL1 edit window, 624 Cylinder CYL2 edit window, 624 Cylinder CYL3 edit window, 625 Cylinder CYL4 edit window, 625 Cylinder CYL5 edit window, 625 default solution plot, 631 Free Mesh Parameters edit window, 629 Geometry Components, 623t geometry modeling, 623, 626 mesh, 630 mesh generation, 628 Model Navigator setup, 623 physics boundary settings: electrostatics (es), 627 physics subdomain settings: electrostatics (es), 627 Plot Parameters selection window, 632 postprocessing and visualization, 630, 632 Save As edit window, 626 Solver Parameters edit window, 631 solving, 630 Sphere edit window, 624 start building, 622 streamline plot with suppressed boundaries. 633 Subdomain Settings edit window, 627 summary and conclusions about, 633 Suppress Boundaries selection window, 633 3D electrostatic potential between two cylinders model. 613-622 Boundary Settings, 616t Boundary Settings (1-4, 11-14) edit window, 617 Boundary Settings (5-10) edit window, 617 Boundary Settings (15–20) edit window, 618 Create Composite Object edit window, 615 Cylinder CYL1 edit window, 614 Cylinder CYL2 edit window, 614 default solution plot, 620 derivation of, 613 Free Mesh Parameters edit window, 618 Geometry components, 613t

geometry modeling, 613, 615 mesh. 619 mesh generation, 617 physics boundary settings: electrostatics (es), 616 physics subdomain settings: electrostatics (es), 616 Plot Parameters selection window, 621 postprocessing and visualization, 619, 622 Save As edit window. 615 Solver Parameters edit window, 620 solving, 618-619 Sphere edit window, 614 start building, 613 streamlining plot with suppressed boundaries. 622 Subdomain Settings edit window, 616 summary and conclusions about, 622 Suppress Boundaries selection window, 621 3D magnetic field of a Helmholtz coil model, 636-652 Boundary Settings (1-4, 21, 22, 31, 32) edit window, 643 Boundary Settings (2, 3) edit window, 642constants, 637 Constants edit window, 637, 637t cross-section field analysis, 648–649 cross-section plot, 651 **Cross-Section Plot Parameters** edit window. 650 derivation of, 636 Free Mesh Parameters edit window General tab. 643 Subdomain tab. 644 Geometry Components, 638t geometry modeling, 638 Helmholtz coil pair, 640 mesh. 644 mesh generation, 643-644 Model Navigator setup, 636 physics boundary settings: Magnetostatics (emqa), 642 physics subdomain settings: Magnetostatics (emga), 641 Plot Parameters selection window

Boundary tab, 648 General tab. 646 Slice tab. 647 postprocessing and visualization, 645–647 Revolve edit window, 639 Save As edit window. 637 solution plot, 650 solution plot with cross-section plot line, 651 solving, 644 sphere and Helmholtz coil pair, 641 Sphere edit window, 640 Square edit window, 638, 639 start building, 636 Subdomain settings, 641t Subdomain Settings (1) edit window, 642 summary and conclusions about, 649 Suppress Boundaries selection window, 645 Work-Plane Settings edit window, 638 3D magnetic field of a Helmholtz coil models, 572 3D magnetic field of a Helmholtz coil with a magnetic test object model, 652-669 Boundary Settings (1-4, 25, 37, 40) edit window, 660 Boundary Settings (2, 3) edit window, 659 constants, 652-653 Constants edit window, 652t, 653 cross-section field analysis, 665-666 cross-section plot, 668 default solution plot, 663 derivation of, 652 Ellipsoid edit window, 656 Free Mesh Parameters edit window General tab. 660 Subdomain (2, 3) tab, 661 Subdomain (4) tab, 661 Geometry Components, 654t geometry modeling, 653-655 Helmholtz coil pair, 656 mesh, 662 mesh generation, 660 physics boundary settings: Magnetostatics (emqa), 659 physics subdomain settings: Magnetostatics (emqa), 658

Plot Parameters edit window, 668 Plot Parameters selection window Arrow tab. 667 Boundary tab, 666 General tab, 664 Slice tab. 665 postprocessing and visualization, 662 Revolve edit window, 655 Save As edit window, 653 solution plot, 667 solution plot with cross-section plot line, 669 solving, 662 sphere, Helmholtz coil pair, and magnetic ellipsoid, 657 Sphere edit window, 657 Square edit window, 654, 655 start building, 652 Subdomain Settings, 658t Subdomain Settings (1) edit window, 658 Subdomain Settings (4) edit window, 659 summary and conclusions about, 666 Suppress Boundaries selection window, 663 Work-Plane Settings edit window, 654 3D modeling considerations for. 571–572 coordinate system, 572–573 3D spherical domain, with PMLs, 672, 673 3D thin layer resistance models, 572 thin layer approximation, 577–593 BLK1 edit window, 578 BLK2 edit windows, 578, 579 boundary pairs value edit window, 588 boundary plot, 591 Boundary Settings (1, 2, 4–10, 12, 13), 586 Boundary Settings—Conductive Media DC (dc) edit window, 586t Boundary Settings-Conductive Media (DC) (dc) edit window, pairs, 586 Boundary Settings (3) edit window, 587 Boundary Settings (11) edit window, 587 Circle edit window, 581 CO1 (intersection of circle and square), 582 Create Composite Object window, 581 Create Pairs edit window, 584 cross-section electric potential plot, 593

3D thin layer resistance models (continued) Cross-Section Line Data Parameters, 592t Cross-Section Plot Parameters edit window, 592 default slice plot, 589 derivation of, 577 EMB1 and BLK1 selected, 584 Embed edit window, 582 EMB1 on top of block BLK2, 583 Geometry Components, 578t geometry modeling, 578-583 Geom2 work-plane, 580 identity-pair-contact resistance approximation used in, 586-587 mesh. 589 mesh generation, 588 model identity pair, 585 Model Navigator setup, 577 Move edit window. 583 physics boundary settings: Conductive Media DC (dc), 586 physics subdomain settings: Conductive Media (DC) (dc), 583-584 Plot Parameters edit window, Boundary tab, 590 postprocessing and visualization, 588 solving, 588 Square edit window, 581 start building, 577 Subdomain edit window, 585t Subdomain Settings window, 585 summary and conclusions about, 591-593 TLR voltage measured across the layer, 590-591 Work-Plane Settings edit window, 580 Work-Plane settings in, 580 X-Axis Data edit window, 591 thin layer subdomain, 593-611 BLK1 edit window, 594 BLK2 edit window, 595 BLK3 edit window, 595 Boundary Settings (1, 2, 4, 5, 7, 8, 10, 12-17) edit window, 603 Boundary Settings-Conductive Media DC (dc) edit window, 601t

Boundary Settings (3) edit window, 603 Boundary Settings (11) edit window, 604 Circle edit window, 598 CO1 (intersection of circle and plane), 599 Create Composite Object window, 598 cross-section electric potential plot, 610 cross-section electric potential plot, thin layer approximation, 610 cross-section electric potential plot, thin layer subdomain, 611 Cross-Section Line Data Parameters, 609t Cross-Section Plot Parameters edit window, 609 default slice plot, 605 Embed edit window, 599 EMB1 on top of block BLK3, 600 Free Mesh Parameters edit window, 604 Geometry Components, 594t geometry modeling, 593, 595-596, 598-600 Geom2 work-plane, 597 mesh. 605 mesh generation, 603–604 Model Navigator setup, 594 model view, 596 Move edit window, 600 physics boundary settings: Conductive Media DC (dc), 601 physics subdomain settings: Conductive Media DC (dc), 600 Plot Parameters edit window, Boundary tab. 607 Plot Parameters edit window, General tab, 606 postprocessing and visualization, 606-607 Save As edit window, 596 solution, boundary plot, 608 solving, 604 Square edit window, 598 start building, 593 Subdomain Settings (1, 3) edit window, 602 Subdomain Settings (2) edit window, 602 summary and conclusions about, 608-609 TLR voltage measured across the layer, 607-608 Work-Plane Settings edit window, 597

Work-Plane Settings in, 595 X-Axis Data edit window, 608 3D touch screen divider circuit. 574 3D touch screen geometry, 574 Time coordinates in 1D models, 64 in 3D transient solution model, 573 in transient solution model, 461 in 2D models, 114, 115 Time-dependent models. See Transient (or time-dependent) models Time dependent solver, solving Kdv equation model and, 73 Time-dependent solving 2D Axisymmetric Inductive\_Heating\_1 model, 408 2D Axisymmetric Inductive\_Heating\_2 model, 431 2D Axisymmetric Inductive\_Heating\_3 model, 452 2D Resistive\_Heating\_1 model, 339 2D Resistive Heating 2 model, 361 2D Resistive\_Heating\_3 model, 383 Times edit window 2D AC Generator model: transient, 523 2D AC Generator Sector model: transient, 559 Time Stepping page Solver Parameters window COMSOL 1D telegraph equation model, 96 first variation on KdV equation model, 81 first variation on telegraph equation model. 103 KdV equation model, 74 second variation on KdV equation model. 88 second variation on telegraph equation model. 109 Time to temperature, 2D axisymmetric tumor laser irradiation model, 767 Tissue (human) transparency of, in certain infrared wavelengths, 750 2D axisymmetric microwave cancer therapy model and conductivity of, 770 TLR voltage measured across the layer

3D thin layer resistance model: thin layer approximation, 590-591 3D thin layer resistance modeling: thin layer subdomain, 607-608 T-max, comparison of, for Cylinder Conduction models 1p and 2, 249t "To the first order," meaning of, 495 Touch screen, 573 3D divider circuit, 574 3D geometry, 574 Transformed AC power, 497 for power line transmission, 498 Transient analysis 2D Axisymmetric Cylinder Conduction model. 234 2D electrochemical polishing models, 165 2D electro-thermal interaction modeling of Joule heating, 325 2D resistive heating models, 388, 454, 455 Transient calculations, physical property values and conduction calculation in, 332 Transient (or time-dependent) models dependent variables in, 390, 460 physics of, difficulty level, 459-460 quasi-static models vs., 321 Transient solution models parameter variance in, 323, 389-390 3D models and, 573 2D axisymmetric coordinate system and, 227 2D axisymmetric models and, 324 2D coordinate system models and, 461 Transparent thermally conductive materials, 306.316 Transverse electromagnetic wave propagated in z direction, voltage relationship for, 463 Triangular mesh, 165, 317, 388, 454 default, in COMSOL Multiphysics software, 140 Triple-pane windows dual- and single-pane windows compared with, 35*t* modeling, 28 workspace lines, 28t Tumor cell death, temperature elevations and, 765, 769, 770, 788, 794

Tumor laser irradiation theory, 749 2D AC Generator models static. 498-521 Application Mode Properties dialog box, 514 assembling Generator Geometry (stator and rotor), 512 circles and rectangles created, 502 circles creation, 501 composite objects, 511 constants, 499-500 Constants edit window, 500, 500t Create Composite Object edit window, 503, 504, 510 derivation of, 498-499 Free Mesh Parameters edit window, 519 generator geometry, 500, 502–503, 505-506, 508-510 Geometry Circles: Copy, Rotate, and Paste. 506t Geometry Circles Creation, 505t Geometry Rectangles creation, 502t initial solution, 521 Materials/Coefficients Library load window, 515 Materials/Coefficients Library window, 514 mesh, 519 mesh generation, 518 model CO1, 503 model CO2, 504 Model Navigator setup, 499 paired stator and rotor, 512, 513 Paste C1 and C2, 506 physics boundary settings, 516 physics settings: Subdomain Integration Variables, 513 physics subdomain settings: Moving Mesh (ALE) (ale), 515 physics subdomain settings: Perpendicular Induction Currents, Vector Potential (emga), 513–515 role of, 520 rotated, pasted circles, 507 rotated paste C3 and C4, 506 rotating portion (rotor) created, 506 rotor created circles, 508

rotor created rectangles, 509 Rotor Geometry Circles Creation, 508t Rotor Geometry Rectangles Creation, 509t Save As edit window, 501 solving, 520, 521 start building, 499 stationary portion of generator created, 506 stator and rotor composite objects, 511 stator and rotor Create Pairs edit window, 512 Stator and Rotor names assigned to composite objects in, 510 Stator Geometry Circles Creation, 500t Subdomain edit window, 516t Subdomain Integration Variables edit window, 513, 513t Subdomain Settings (1, 3-19) edit window, 516 Subdomain Settings (2, 28) edit window. 517 Subdomain Settings (20, 23, 24, 27) edit window, 517 Subdomain Settings (21, 22, 25, 26) edit window, 518 Subdomain Settings-Moving Mesh edit window, 516t Subdomain Settings-Moving Mesh (19-28) edit window, 518 summary and conclusions about, 529 two new created circles, 505 union, intersection, and difference commands, 502, 509 union of all objects, CO1, 507 transient, 498, 522–529 animation, final frame, 531 derivation of, 522 Domain Plot Parameters edit window, 528 Induced voltage vs. Time, 529 magnetic flux density and magnetic potential in, 528 Model Navigator initial solution, 523 Plot Parameters, Animate tab, 530 Plot Parameters, Contour tab, 527 Plot Parameters, General tab, 525 Plot Parameters, Surface tab, 526 postprocessing and visualization, 525–527

postprocessing animation, 529 Solver Parameters edit window, 524 Solver Parameters entered exactly as specified in, 523 solving, 522–523, 524 start building, 522 summary and conclusions about, 529 2D AC Generator Sector models static, 530, 531–557 Application Mode Properties: Perpendicular Induction Currents, Vector Potential (emqa), 542, 543 assemble the geometry (stator and rotor), 536, 538 Boundary Settings, Conditions page (1-4, 7, 8, 15, 16, 19, 20), 548Boundary Settings, Pairs page (1), 549 Boundary Settings, Weak Constr. page (1-4, 7, 8, 15, 16, 19-21, 23), 548constants, 532 Constants edit window, 533, 533t copied meshed boundaries, 553 copying mesh from one edge to the other edge, 553 coupling variables note, 542 Create Composite Object, intersection (CO1, CO3), 536 Create Composite Object, intersection (CO2, CO1), 536 created one-eighth generator sector (CO3, CO4), 537 dependent variables case changes in, 532 derivation of, 531 Free Mesh Parameters, Boundary page, 552 Free Mesh Parameters edit window, 552 generator geometry and closed polyline (solid). 535 Generator Sector Geometry, 532, 534–536 Global Equations edit window, 551, 551t Global Expressions edit window, 538t, 539 Identity Pairs window, 538 Import CAD Data From File select window, 533 imported CAD file, 534 initial solution, 557

Interchange source and destination button, 538 Line, closed polyline (solid) edit window, 535 Materials/Coefficients Library load window, 544 meshed view, 554 mesh generation, 552–554 Mesh Remaining (Free) button, 554 Model Navigator setup, 532 names in Application Mode Name edit window, 532 with omega unselected, 556 **Options: Global Expressions**, 538 **Options: Subdomain Expressions**, 539 **Options: Subdomain Integration** Variables, 539 paired stator (CO1) and rotor (CO2), 537 pairing of stator and rotor in, 538 Periodic Point Conditions, Destination page (1), 550 Periodic Point Conditions, Destination Vertices page (11), 551 Periodic Point Conditions, Source page (4), 549 Periodic Point Conditions, Source Vertices page (4), 550 Physics Boundary Settings: Perpendicular Induction Currents, Vector Potential (emqa), 547–549 Physics Settings: Global Equations, 551 Physics Settings: Periodic Point Conditions, 549-551 Physics Subdomain Settings: Moving Mesh (ALE) (ale), 542 Physics Subdomain Settings: Perpendicular Induction Currents, Vector Potential (emga), 542-544, 546 Save As edit window. 556 Solver Parameters edit window, 555 solving, 554–555 start building, 531 Subdomain edit windows, 544t Subdomain Expressions (2, 4)edit window, 541

2D AC Generator Sector models (continued) Subdomain Expressions edit window, 539t. 540t Subdomain Expressions (1) edit window, 541 Subdomain Expressions (1-4) edit window. 540 Subdomain Expressions (5-8) edit window, 540 Subdomain Integration Variables (1, 2, 4)edit window. 541 Subdomain Integration Variables (7, 8) edit window, 542 Subdomain Integration Variables edit window, 540t Subdomain Settings, Forces (1, 2, 4) edit window, 547 Subdomain Settings (1, 6) edit window, 545 Subdomain Settings (3, 5, 7, 8) edit window, 546 Subdomain Settings (7, 8) edit window, 547 Subdomain Settings (2) edit window, 545 Subdomain Settings (4) edit window, 546 Subdomain Settings-Moving Mesh (ALE) edit window (1-4), 543 summary and conclusions about, 555, 557 transient, 530, 557-568 animation, final frame, 568 derivation of, 557 global variables plot, 566 Global Variables Plot edit window, 565 magnetic flux density and magnetic potential, z component, 565 Model Navigator, initial solution, 558 Model Navigator, initial solution selection, 558 Parameters, general tab, 562 Plot Parameters, Animate tab, 567 Plot Parameters, Contour tab, 564 Plot Parameters, Surface tab, 563 postprocessing and visualization, 561-562, 564 postprocessing animation, 566 solution default plot, final frame, 561 Solver Manager, Solve For page, 560 Solver Parameters edit window, 559

solving, 559-560 start building, 557 summary and conclusions about, 568 2D axisymmetric advanced-level 2D modeling, 226 2D axisymmetric coordinate system, 389–393 basics of, 389 discussion about, 324 plus time, 322, 324, 390 2D Axisymmetric Cylinder Conduction models, 229-240,230 Boundary Settings: general heat transfer, 235 Boundary Settings (2, 5, 6) edit window, 236 Boundary Settings (3) edit window, 237 Boundary Settings-General Heat Transfer edit window, 235t consolidating calculational parameters, 231 constants, 231 Constants edit window, 231, 231t cylinder rectangle, 232 default solver used with, 238 first variation on, 240-250 Boundary Settings: General Heat Transfer edit window, 246t Boundary Settings (2, 5, 6) edit window, 247 Boundary Settings (3) edit window, 247 Constants edit window, 242, 242t cylinder rectangle, 243 derivation of, 240 final frame, 251, 252 mesh generation, 246 model animation, final frame, 241 model mesh window, 237, 248 Model Navigator setup, 230, 242 models 1p and 2, comparison of T-max for, 249, 249t Parametric (UMFPACK) used with, 246, 249 physics boundary settings: general heat transfer. 245 physics subdomain settings: general heat transfer, 245 Plot Parameters window, 240, 250 Point edit window, 244, 244t points added in, 243 Rectangle edit window, 243

rectangle with points, 244 Solver Parameters edit window, 248 solving, 246–247 start building, 241 Subdomain edit window, 245t Subdomain Settings edit window, 245 mesh generation, 235 niobium in, 229 Parametric Solver (UMFPACK), 239 parametric solving of, 236, 238 physics boundary settings: general heat transfer. 235 physics subdomain settings: general heat transfer, 233-234 Point edit window, 232t. 233 postprocessing animation, 238, 240 Rectangle edit window, 232 rectangle with points, 233 results summary, 264t second variation on, including a vacuum cavity animation, final frame, 251, 264 Boundary Settings (1, 4) edit window, 259 Boundary Settings (2, 5, 10) edit window, 260 Boundary Settings (3) edit window, 260 Boundary Settings—General Heat Transfer edit window, 259t composite object, 257 consolidating calculational parameters, 253 constants, 253 Constants edit window, 253, 253t Create Composite Object edit window, 256 cylinder rectangles (1 and 2), 255 derivation of, 250 mesh. 261 mesh generation, 261 Model Navigator setup, 252 Parametric Solver use, 262 physics boundary settings: general heat transfer, 259 physics subdomain settings: general heat transfer, 256, 258 Plot Parameters window, 263 Point Edit window, 255t points added to, 254

postprocessing animation, 261, 263 Rectangle edit window, 253t Rectangle edit window (1), 254 Rectangle edit window (2), 254rectangles with points, 256 second variation on, 250-263 setting pressure for vacuum cavity, 258 Solver Parameters edit window, 262 solving, 261 start building, 250 static and quasi-static calculations in, 259 Subdomain Edit window, 258t Subdomain Settings (1) edit window, 257 Subdomain Settings (2) edit window, 258 Solver Parameters edit window. 239 solving, 236 start building, 230 static and quasi-static calculations with, 234 Subdomain edit window, 234t Subdomain Settings edit windows, 234 summary and conclusions about, 263-264 2D axisymmetric heat conduction modeling, 229 2D axisymmetric Inductive Heating models, 393-411, 399, 422 animation, final frame, 411 Application Scalar Variables edit window, 399 Axis edit window, 395, 395t Boundary Settings (1, 3, 5) edit window, 401.405 Boundary Settings (2, 7, 9) edit window, 402, 406 Boundary Settings edit window, 401t, 402t, 404t Boundary Settings (10-13) edit window, 402 Circle edit window, 397 consolidating calculational parameters, 396 constants, 396, 398 Constants edit window, 397, 397t coupling of Joule heating and heat transfer in. 393 degrees Centigrade in, 410 derivation of, 393 first variation on, 411–433 animation, final frame, 436 **Application Scalar Variables** edit window, 420

2D axisymmetric Inductive Heating models (continued) Boundary Settings (1, 2, 4) edit window, 425 Boundary Settings (1, 3, 4), edit window, 429 Boundary Settings (2, 5, 9) edit window, 425 Boundary Settings (2, 10, 14) edit window, 430 Boundary Settings edit window, 423t, 426t, 427t Boundary Settings (12–203) edit window, 426 Circle edit window, 419t CO1, CO3, 417 CO2, 416 coil, 427 constants, 413-416, 418-419 Constants edit window, 413t, 414 Create Composite Object edit window, 416, 417, 418, 421 Create Composite Object edit windows, 415 crucible, building, 418 crucible and coil. 420 derivation of, 411 inductively heated crucible built in, 411 mesh generation, 430 mesh window, 431 Model Navigator setup, 412 options and settings, 413 physics boundary settings: Azimuthal Induction Currents, Vector Potential (emga), 423 physics boundary settings: general heat transfer (htgh), 427 physics settings, 419, 421 physics subdomain settings: Azimuthal Induction Currents, Vector Potential (emga), 422 physics subdomain settings: general heat transfer (htgh), 426-427 Plot Parameters edit window, 434 Plot Parameters window, 436 postprocessing and visualization, 431, 433 Scalar Expressions edit window, 421, 421t Solver Parameters, Advanced edit window, 432

Solver Parameters edit window, 432 solving, 430-431, 433 start building, 412 Subdomain edit window, 423t, 426t Subdomain Settings (1, 3) edit window, 428 Subdomain Settings (1), Electric Parametrs edit window. 423 Subdomain Settings (1–50), Init edit window, 429 Subdomain Settings (2), Electric Parameters edit window, 424 Subdomain Settings (3–50), Electric Parameters edit window, 424 Subdomain Settings (2–50) edit window, 428 temperature distribution shown in degrees Centigrade, 431, 435 3D rendition of, 412 time-dependent solving of, 431 Grid edit windows, 395t, 396 mesh generation, 406 mesh window, 407 Model Navigator setup, 394 options and settings, 395 physics boundary settings: Azimuthal Induction Currents, Vector Potential (emqa), 401-402 physics boundary settings: general heat transfer (htgh), 404 physics settings, 398 physics subdomain settings: Azimuthal Induction Currents, Vector Potential (emqa), 400 physics subdomain settings: general heat transfer (htgh), 403–404 Plot Parameters edit window, 409, 410 postprocessing and visualization, 408 postprocessing animation, 408 Rectangle edit windows, 398 Scalar Expressions edit window, 399t, 400 second variation on, 433-456 animation, final frame, 456 **Application Scalar Variables** edit window, 441 bismuth and building filled inductively heated crucible in, 434

bismuth subdomain 3, 440 Boundary Settings (1, 3–7) edit window, 444, 449 Boundary Settings (2, 9, 14) edit window, 445.450 Boundary Settings edit window, 444t, 448t Boundary Settings (17–208) edit window, 445 coil, 446 constants, 437, 439-440 Constants Edit window, 438, 438t coupling of Joule heating and heat transfer in. 433–434 in degrees Centigrade, 435, 455 derivation of, 433 imported crucible and coil with rectangle, 439 mesh generation, 450 mesh window, 451 Model Navigator setup, 437 mode names in Application mode window in. 435 physics boundary settings: Azimuthal Induction Currents, Vector Potential (emga), 442, 445 physics boundary settings: general heat transfer (htgh), 448 physics settings, 440 physics subdomain settings: Azimuthal Induction Currents, Vector Potential (emga), 441–442 physics subdomain settings: general heat transfer (htgh), 446, 448 Plot Parameters edit window, 434, 454 Plot Parameters window, 456 postprocessing and visualization, 452 postprocessing animation, 453 Scalar Expressions edit window, 441, 441t Solver Parameters, Advanced edit window, 452 Solver Parameters edit window, 451 solving, 450, 452, 453 start building, 435 Subdomain edit window, 442t, 446t Subdomain Settings (1), Electric Parameters edit window, 442

Subdomain Settings (1-51), Init edit window, 449 Subdomain Settings (2, 4-51) edit window, 447 Subdomain Settings (2), Electric Parameters edit window, 443 Subdomain Settings (3), Electric Parameters edit window, 443 Subdomain Settings (4–51), Electric Parameters edit window, 444 Subdomain Settings (1) edit window, 447 Subdomain Settings (3) edit window, 448 3D rendition of, 437 time-dependent solving of, 452 Solver Parameters, Advanced edit window, 408 Solver Parameters edit window, 407 solving, 406, 409 start building, 394 Subdomain edit window, 401t, 403t Subdomain Settings (1–3), Init edit window, 405 Subdomain Settings (2, 3) edit window, 404 Subdomain Settings (2), Electric Parameters edit window, 400 Subdomain Settings (1) edit window, 403 summary and conclusions about, 453-455 3D rendition of, 394 time-dependent solving of, 408 2D axisymmetric insulated container design, 265 2D axisymmetric microwave cancer therapy model, 770-794 antenna and model domain, 799 antenna plus tissue, 771 Application Scalar Variables, 782t Boundary Settings, 781, 783t Boundary Settings, (8), Port tab, 787 Boundary Settings (1, 3), 785 Boundary Settings (2, 14, 18, 20, 21), 785 Boundary Settings (5-7, 9, 11-13, 17), 786 Boundary Settings (8), 786 composite object (CO1), 776 constants, 772 Constants edit window, 773, 773t Create Composite Object edit window, 775, 777

2D axisymmetric microwave cancer therapy model (*continued*) **Cross-Section Plot Parameters** General tab. 792 Line/Extrusion tab, 793 derivation of, 770 Free Mesh Parameters Global tab. 787 subdomain 3, 788 Geometry Components, 774t, 776t, 778t geometry modeling, 772, 774, 776, 778-779 Line edit window, 778 mesh. 789 mesh generation, 784 Model Navigator Heat Transfer Module, 771 RF Module, 772 physics boundary settings: Bioheat Equation (htbh), 781 physics settings: Bioheat Equation (htbh), 779–780 physics settings: 2 TM waves (rfwh), Boundary Settings, 782 physics settings: 2TM waves (rfwh), Scalar Variables, 782 physics settings: 2TM waves (rfwh), Subdomain Settings, 782 postprocessing and visualization, 788, 790 postprocessing Plot Parameters, Surface tab (°C), 791 Rectangle edit window (R1), 774, 777, 778t Rectangle edit window (R2), 774, 777 rectangles (R1, R2), 775 Save As window, 773 solution, temperature (K), 790 Solver Parameters, 789 solving, 788 start building, 771 subdomain 1, 783 subdomain 2, 783 subdomain 3, 784 subdomain 4, 784 Subdomain Settings, 782t Subdomain 1 settings, 780t

Subdomain Settings (1), 781 Subdomain Settings (2, 3, 4), 780 summary and conclusions about, 794 surface temperature (°C), 792 temperature (T) as function of applied power versus radius (r), 793 2D axisymmetric modeling, 113 considerations for, 225 coordinate system, 227, 227-228 difficulty level with, 225 heat conduction modeling, 229 heat conduction theory, 228-229 implicit assumptions in, 226 2D axisymmetric Thermos Container models, 265-318 Boundary Integration edit window, 284 Boundary Settings (8, 14, 21), 278 Boundary Settings (2) edit window, 278 Boundary Settings (28) edit window, 279 Boundary Settings (34) edit window, 280 Boundary Settings-General Heat Transfer edit window, 277t building the 2D axisymmetric thermos container, 267 consolidating calculational parameters, 266 constants. 266 Constants edit window, 267, 267t Create Composite Object edit window, 272 derivation of, 265 Ellipse Edit window, 268t final frame, 286 first variation on, 285-301 animation, final frame, 302 Boundary Integration edit window, 300 Boundary Settings (8, 14, 21) edit window, 294 Boundary Settings (2) edit window, 293 Boundary Settings (28) edit window, 295 Boundary Settings (34) edit window, 295 Boundary Settings—General Heat Transfer edit window, 293t, 294t choosing name for modeler-defined parameters, 287 consolidating calculational parameters, 286 constants, 286

Constants edit window, 287, 288t edit window, 291 Free Mesh Parameters edit window, 296 import, 288 import and rectangle R1, 289 importing 2D axisymmetric thermos container. 288 mesh. 297 mesh generation, 296 Model Navigator setup, 287 Parametric Solver use, 298 physics boundary settings: general heat transfer, 293–294, 296 physics subdomain settings: general heat transfer, 289-290, 293 Plot Parameters window, 299, 301 postprocessing, 298, 301 Rectangle edit window, 289t Solver Parameters edit window, 297 solving, 296, 298 start building, 286 Subdomain edit window, 290t, 292 Subdomain edit window (2, 7), 291tSubdomain edit window (4), 291t Subdomain edit window (5, 9), 291tSubdomain Settings (1, 3, 6, 8) edit window, 290 Subdomain Settings (4) edit window, 292 surface temperature (°C), 299 temperature uniformity in, 293 user interface display window, 300 vacuum cavity replaces urethane foam in. 285 Free Mesh Parameters edit window, 280 half-ellipse creation, 269 Line edit window, 271 mesh. 281 mesh generation, 279 Model Navigator setup, 266 names for modeler-defined parameters, 266 outer tank profile, 270 outer tank profile creation, 269 Parametric Solver use, 282 physics boundary settings: general heat transfer, 277, 279

physics subdomain settings: general heat transfer. 274 Plot Parameters window, 283, 285 postprocessing, 282 Rectangle Edit window, 268t, 269t rectangles R1 and R2, 268 second variation on, 301–318 animation, final frame, 318 Boundary Integration edit window, 315 Boundary Settings (8, 14, 21) edit window, 310 Boundary Settings (2) edit window, 309 Boundary Settings (28) edit window, 311 Boundary Settings (34) edit window, 311 boundary settings-General Heat Transfer edit window, 309t, 310 choosing name for, 304 constants, 302 Constants edit window, 303, 303t creating domain for boundary information to be applied, 304 energy lost in this model vs. with urethane foam insulation in 2D axisymmetric Thermos Container 1 model, 316 Free Mesh Parameters edit window, 312 glass material and vacuum cavity in, 301 import, 304, 304 mesh. 313 mesh generation, 312 Model Navigator setup, 303 Parametric Solver use, 314 physics boundary settings: general heat transfer, 309-310 physics subdomain settings: general heat transfer, 305-306 Plot Parameters window, 314.317 postprocessing, 312, 315-316 postprocessing animation, 316 Rectangle edit window, 305t Solver Parameters edit window, 313 solving, 312 start building, 302 Subdomain edit windows, 305t, 306t, 308t

2D axisymmetric Thermos Container models (continued) Subdomain Settings (1, 3, 6, 8) edit window, 306 Subdomain settings (2, 7) edit window, 307 Subdomain Settings (5, 9) edit window, 307 Subdomain Settings (4) edit window, 308,308 surface temperature (°C), 315 user interface display window, 316 Solver Parameters edit window, 281 solving, 279, 282 start building, 265 subdomain edit window (1, 3, 6, 8), 274tsubdomain edit window (2, 7), 274tsubdomain edit window (4), 274t subdomain edit window (5, 9), 274tSubdomain Settings (1, 3, 6, 8) edit window. 275 Subdomain Settings (2, 7) edit window, 275 Subdomain Settings (5, 9) edit window, 276 Subdomain Settings (4) edit window, 276 summary and conclusions about, 316-317 surface temperature (°C), 283 tank, 273 tank components, 271 tank components with lid line, 272 tank Export Geometry Objects, Save As window, 273 tank structure creation steps, 270t temperature uniformity in, 277 thermos container modeling results summary, 317t user interface display window, 284 2D axisymmetric Tumor Laser Irradiation model, 749-769 Boundary Settings, 760t Boundary Settings (1, 3-5), 761 Boundary Settings (2, 8, 9), 761 Boundary Settings (7), 762 Circle edit window, 754 constants, 750 Constants edit window, 751t, 752 Create Composite Object edit window, 755 derivation of, 749

Domain Plot Parameters, Point tab, 767 Free Mesh Parameters, subdomain 2, 762 Geometry Components, 752t geometry modeling, 750, 753–755 mesh, 763 mesh generation, 761 modeling domain overview, 750 Model Navigator setup, 751 physics boundary settings: Bioheat Equation (htbh), 760 physics settings: Scalar Expressions, 755 physics settings: Subdomain Expressions, 755,757 physics subdomain settings: Bioheat Equation (htbh), 758 Plot Parameters, Animate tab, 768 Plot Parameters, animation, final frame, 769 postprocessing and visualization, 763, 765 postprocessing Plot Parameters, Surface tab (°C), 765 Rectangle edit window (R1), 752 Rectangle edit window (R2), 753 Rectangle edit window (R3), 755 rectangles (R1, R2), 753 Save As edit window, 754 Scalar Expressions edit window, 756 skin, tissue, and tumor (CO1), 756 solution, temperature (K), 764 Solver Parameters, 764 solving, 763 start building, 750 Subdomain Expressions (1) edit window, 757 Subdomain Expressions (2) edit window, 757 Subdomain Expressions (3) edit window, 757 Subdomain Settings, 758t Subdomain Settings (1), 758 Subdomain Settings (2), 759 Subdomain Settings (3), 759 Subdomain Settings, Initial value edit window, 760 summary and conclusions about, 768-769 surface temperature (°C), 766 time to temperature, 767 2D beginning-level to advanced-level 2D modeling, 113, 226

2D Bio MCT 1 model. See 2D axisymmetric microwave cancer therapy model 2D Cartesian domain, with PMLs, 672, 672 2D complex mixed-mode modeling considerations relative to, 459-460 implicit assumptions in, 460 2D concave mirror model, without PMLs, 724-744 Application Mode Properties edit window, 729 Application Scalar Variables edit window (lambda1 rfweh), 730 Boundary Settings, 732 concave mirror (CO1), 728 Create Composite Object edit window, 727 derivation of, 724 electric field, z-component, 737 electric field plot, z-component, 0.5 m, 739.740 electric field plot, z-component, 1.0 m, 741.742 electric field plot, z-component, 1.5 m, 743 Ellipse edit window (E1), 726 Ellipse edit window (E2), 726 geometry modeling, 725, 727 Materials-Coefficients Library, copper, 731 mesh generation, 732 model domain, 729 model mesh, 733 Model Navigator setup, 725 physics application mode properties: In-Plane TE waves (rfweh), 727 physics application scalar variables: In-Plane TE waves (rfweh), 730 physics boundary settings: In-Plane TE waves (rfweh), 732 physics subdomain settings: In-Plane TE waves (rfweh), 730 Plot Parameters, Animate tab, 738 postprocessing and visualization, 734–735 postprocessing Plot Parameters, Surface tab, 736 Rectangle edit window (R1), 727 Save As edit window, 726 scattered electric field, z-component (V/m), 735

Solver Parameters, 734 solving, 732 Square edit window (SQ1), 728 start building, 724 subdomain Free Mesh Parameters, 733 Subdomain Settings, Physics tab, subdomain 1.730 Subdomain Settings, Physics tab, subdomain 2.731 summary and conclusions about, 736-737 2D concave mirror model, with PMLs, 707-724 Application Mode Properties edit window, 715 Application Scalar Variables edit window (lambda0 rfweh), 715 Boundary Settings, 719 concave mirror (CO1), 713 Create Composite Object edit window, 712 derivation of, 707 domain plus PMLs, 714 electric field, z-component, 723 Ellipse edit window (E1), 711 Ellipse edit window (E2), 711 Free Mesh Parameters, subdomain, 719 Geometry Components, 708t geometry modeling, 707, 710, 712 Materials-Coefficients Library, copper, 717 mesh, 720 mesh generation, 718 Model Navigator setup, 708 physics application mode properties: In-Plane TE waves (rfweh), 714 physics application scalar variables: In-Plane TE waves (rfweh), 714 physics boundary settings: In-Plane TE waves (rfweh), 718 physics subdomain settings: In-Plane TE waves (rfweh), 714 Plot Parameters, Animate tab, 724 Plot Parameters, Surface tab, 722 PML rectangles, 710 postprocessing and visualization, 721, 723 Rectangle edit window (R1), 708 Rectangle edit window (R2), 709 Rectangle edit window (R3), 709 Rectangle edit window (R4), 709

2D concave mirror model, with PMLs (continued) Rectangle edit window (R5), 712 Save As edit widow, 711 scattered electric field, z-component (V/m), 721 Solver Parameters, 720 solving, 718 Square edit window, 713 start building, 707 Subdomain Settings, Physics tab, subdomain 5, 717 Subdomain Settings, Physics tab, subdomain 7,718 Subdomain Settings, PML type selection, 715 Subdomain Settings, x absorption, 716 Subdomain Settings, y absorption, 716 summary and conclusions about, 723 Suppress Subdomains, 722 2D coordinate system, 5 discussion about, 322-323 plus time, 322, 323, 461 2D dielectric lens model, without PMLs, 691-707 Application Mode Properties edit window, 696 Application Scalar Variables edit window (lambda0 rfweh), 697 Boundary Settings, 699 Circle edit window (C1), 693 Create Composite Object edit window, 694 derivation of, 691 dielectric lens (CO1), 695 domain, 696 electric field, z-component, 703 electric field, z-component, 0.5 m, 704 Free Mesh Parameters, subdomain, 699 geometry modeling, 693-694 mesh. 700 mesh generation, 698 Model Navigator setup, 692 model plot electric field, z-component, 0.5 m, 705 model plot electric field, z-component, 1.0 m, 705, 706 model plot electric field, z-component, 1.5 m, 706, 707

physics application mode properties: In-Plane TE waves (rfweh), 696 physics application scalar variables: In-Plane TE waves (rfweh), 697 physics boundary settings: In-Plane TE waves (rfweh), 698 physics subdomain settings: In-Plane TE waves (rfweh), 697-698 Plot Parameters, Animate tab, 704 postprocessing and visualization, 701-702 postprocessing Plot Parameters, Surface tab, 702 Rectangle edit window (R1), 694 Save As edit window, 693 scattered electric field, z-component (V/m), 701 Solver Parameters, 700 solving, 698, 701 Square edit window (SQ1), 695 start building, 691 Subdomain Settings, subdomain 1, 697 Subdomain Settings, subdomain 2, 698 summary and conclusions about, 702-703 2D dielectric lens model, with PMLs, 676-691 Application Mode Properties edit window, 682 Application Scalar Variables edit window (lambda0 rfweh), 683 Boundary Settings, 686 Circle edit window, 680 Create Composite edit window, 680 derivation of, 676 dielectric lens (CO1), 681 domain plus PMLs, 682 domain (SQ1), 681 electric field, z-component, 691 Free Mesh Parameters, 687 Geometry Components, 677t geometry modeling, 677-678, 681 mesh. 687 mesh generation, 686 Model Navigator setup, 676 physics application mode properties: In-Plane TE waves (rfweh), 682 physics application scalar variables: In-Plane TE waves (rfweh), 683

physics boundary settings: In-Plane TE waves (rfweh), 686 physics subdomain settings: In-Plane TE waves (rfweh), 683-685 Plot Parameters, Animate tab, 692 Plot Parameters, Surface tab, 690 PML rectangles, 679 postprocessing and visualization, 688, 690 Rectangle edit window (R1), 677 Rectangle edit window (R2), 677 Rectangle edit window (R3), 678 Rectangle edit window (R4), 678 Rectangle edit window (R5), 680 Save As edit window, 679 scattered electric field, z-component (V/m), 689 Solver Parameters. 688 solving, 686 start building, 676 Subdomain Settings, Physics tab, subdomain 5,685 Subdomain Settings, Physics tab, subdomain 7,685 Subdomain Settings, PML type selection, 683 Subdomain Settings, x absorption, 684 Subdomain Settings, y absorption, 684 summary and conclusions about, 690-691 Suppress Subdomains, 689 2D electric impedance sensor models advanced, 477-493 animation, final frame, 494 **Application Scalar Variables** edit window, 484 Boundary Settings (1, 4, 5, 8) edit window, 486 Boundary Settings (6, 7) edit window, 487 Boundary Settings (2, 3) edit window, 486 Boundary Settings—In-Plane Electric Currents (emqvw) edit window, 485t Boundary Settings (port) edit window, 487 Color Range edit window, 491 constants, 477, 479, 481–482 Constants edit window, 478t, 479 Create Composite Object edit window, 481 Create Composite Object result, 482 ellipse and points, 483

Ellipse edit window, 480 Free Mesh Parameters edit window, 488 high-frequency currents used in, 460 linspace function changes in, 489 mesh, 489 mesh generation, 488 Model Navigator setup, 478 physics boundary settings: In-Plane Electric Currents (emqvw), 484 physics settings: Scalar Expressions, 483 physics settings: Scalar Variables, 484 physics subdomain settings: In-Plane Electric Currents (emqvw), 484 Plot Parameters edit window, 491 Plot Parameters window, 493 Point edit window, 482t postprocessing and visualization, 489 postprocessing animation, 492 rectangle, 480 rectangle added to ellipse in, 481 Rectangle edit window, 481 Save As edit window, 479 Scalar Expressions edit window, 483t, 484 solution with detected areas, 492 Solver Parameters edit window, 490 solving, 488-489, 490 start building, 477 Subdomain edit window, 485t Subdomain Settings edit window, 485 basic, 464-477 **Application Scalar Variables** edit window, 469 Boundary Settings (1, 2, 6) edit window, 471 Boundary Settings (3, 5) edit window, 471 Boundary Settings (4) edit window, 472 Boundary Settings—In-Plane Electric Currents edit window, 471t Boundary Settings (port) edit window, 472 Color Range edit window, 476 consolidating calculational parameters, 464 constants, 464-466 Constants edit window, 465t, 466 derivation of 464 Free Mesh Parameters edit window, 473 high-frequency currents used in, 460

2D electric impedance sensor models (continued) importing/exporting expressions edit windows as text files, 465 mesh. 473 mesh generation, 472 Model Navigator setup, 465 physics boundary settings: in-plane electric currents (emqvw), 470, 472 physics settings: scalar expressions, 468 physics settings: scalar variables, 468 physics subdomain settings: in-plane electric currents (emqvw), 470 Plot Parameters edit window, 476 postprocessing and visualization, 474 rectangle, 467 rectangle and points, 468 Rectangle edit window, 467 Save As edit window. 466 Scalar Expressions edit window, 469, 469t skin depth parameters for, 464 solution with detected areas. 477 Solver Parameters edit window, 474 solving, 474, 475 start building, 464 Subdomain edit windows, 470, 470t summary and conclusions about, 475 high-frequency currents employed in, 460 summary and conclusions about, 492-493 2D electrochemical polishing (electropolishing) theory, 115-166 as inverse of electroplating, 117 Moving Mesh Application Mode, 117–118 numerical solution model and, 115-116 surface normal vector **n** and current vector **J**, 117 2D asperity (bump) on electrode, 116 2D electrochemical polishing models boundary settings, Moving Mesh (ALE) window, 125t building started for, 118-119, 122 Circle edit window, 120 Constants Edit window, 118t Constants edit window, 119 electrode with asperity, 122 electrolyte rectangle, 120

electropolishing 1 model Boundary Settings, Conductive Media DC: boundaries set, 124 electropolishing 1 model Boundary Settings, Moving Mesh (ALE): boundaries 3, 4, 6, 7 window, 125 first variation on, 130-147 animation Plot Parameters window, 131 boundary settings, Conductive Media DC window, 137t boundary settings, Moving Mesh (ALE): boundaries organized by color, 142 boundary settings, Moving Mesh (ALE) window, 140 boundary settings (1, 5), Conductive Media DC: boundaries set, 138 boundary settings (1, 5), Moving Mesh (ALE): Boundaries window, 140 boundary settings (1, 5 = blue; 3, 4, 75)green; 2 = red) Conductive Media DC: boundaries set. 139 boundary settings (2), Conductive Media DC: boundaries set, 139 boundary settings (2), Moving Mesh (ALE): Boundaries window, 141 boundary settings (3, 4, 6, 7), Conductive Media DC: boundaries set, 138 boundary settings (3, 4, 6, 7), Moving Mesh (ALE): Boundaries window, 141 Circle edit window, 134 Constants Edit window, 131t Constants edit window, 132 electrode with asperity, 135 electrolyte rectangle, 133 Free Mesh Parameters window, 142 free mesh (quad), 143 mesh generation, 140 Model Navigator setup, 132 physics boundary settings: Conductive Media DC, 137 physics boundary settings: Moving Mesh (ALE), 138, 140 physics subdomain settings: Conductive Media DC, 137 Plot Parameters window, 144

postprocessing, 144–145 postprocessing animation, 146 rectangle and circle, 134 Rectangle edit window, 133 scaled electrolyte/electrode geometry, 136 Scale edit window, 136 selected rectangle and circle, 135 Solver Parameters window, 143 solving, 141, 144 starting building for, 130–132, 134, 136 Subdomain Settings window, 137 surface Plot Parameters window: Moving Mesh (ALE), y direction, 146 surface plot window: Moving Mesh (ALE), y direction, 147 surface plot window, total normal current density, 145 mesh, 126 mesh generation, 125 modeling results summary, 166t physics boundary settings: Conductive Media DC, 123 physics boundary settings: Moving Mesh (ALE), 125 physics subdomain settings: Conductive Media DC, 123 Plot Parameters window, 127 Plot Parameters window: Moving Mesh (ALE), y direction, 129 postprocessing, 127-128 postprocessing animation, 129–130 power of, with diverse projects, 166 rectangle and circle, 121 Rectangle edit window, 119 scaled electrolyte/electrode geometry, 123 Scale edit window, 122 second variation on, 147-165 animation Plot Parameters window, 148, 167 boundary settings, Conductive Media DC window, 156t boundary settings: Moving Mesh (ALE), 159 boundary settings, Moving Mesh (ALE): boundaries organized by color, 161 boundary settings, Moving Mesh (ALE) window, 159t

boundary settings (1, 7), Conductive Media DC: boundaries set, 157 boundary settings (1, 7), Moving Mesh (ALE): Boundaries window, 159 boundary settings (1, 7 = blue; 3-6,8-13 =green; 2 =red), Conductive Media AC, boundaries set, 158 boundary settings (2), Conductive Media DC: boundaries set, 158 boundary settings (2), Moving Mesh (ALE): Boundaries window, 160 boundary settings (3-6, 8-13), Conductive Media DC: boundaries set, 157 boundary settings (3–6, 8–13), Moving Mesh (ALE): Boundaries window, 160 Circle edit window, 151 Constants edit window, 148t, 149 Create Composite Object edit window, 154 electrode with asperities, 154 electrolyte rectangle, 150 Ellipse edit window, 152 Free Mesh Parameters window, 161 free mesh (quad), 162 mesh generation, 159–160 Model Navigator setup, 149 Paste edit window, 153 physics boundary settings: Conductive Media DC, 156 physics subdomain settings: Conductive Media DC, 156 postprocessing, 162–164 postprocessing animation, 164-165 R1-C1-E1-E2 information, keying in, 153 rectangle and circle, 151 Rectangle edit window, 150 scaled electrolyte/electrode geometry, 155 Scale edit window, 155 Solver Parameters window, 162 solving, 160, 163 starting building for, 147, 149, 151–155 Subdomain Settings window, 156 surface Plot Parameter window: Moving Mesh (ALE), y direction, 165 surface Plot Parameter window, total normal current density, 163

2D electrochemical polishing models (continued) Surface Plot window, total normal current density, 164 2D electropolishing 3 model (C1, E1, E2, R1), 153 2D electropolishing 3 model (C1, E1, R1), 152 selected rectangle and circle, 121 Solver Parameters window, 126 solving, 125, 127 subdomain settings, Conductive Media DC window, 123t Subdomain Settings window, 124 summary and conclusions about, 165-166 surface plot window, 128 surface plot window: Moving Mesh (ALE), y direction, 130 2D guidelines, for new COMSOL multiphysics modelers, 113-115 coordinate system, 114-115 2D modeling considerations, 113–114 2D Hall effect models, 171-222 Application Mode Properties window, 172 boundary settings, 176, 176t, 180 Boundary Settings (1), 177 Boundary Settings (4), 178 Boundary Settings (5), 178 Boundary Settings (8), 179 Boundary Settings, Weak Constr. button. 179 Boundary Settings (2, 3, 6, 7), 177 conductivity matrix elements, 175 considerations relative to, 167-171 constants, 172 Constants edit window, 172t, 173 cross-section Line Data edit window, 185t Cross-Section Parameters, General page, 184 Cross-Section Parameters, Line/Extrusion page, 185 default surface voltage distribution plot, 182 determining exact voltage difference at any point in, 183 determining voltage difference between upper/lower surfaces, 181 final configuration, 180

first variation on, 186-203 animation Plot Parameters window, 187, 204 Application Mode Properties window, 188 boundary settings, 194-197, 194t Boundary Settings (1), 194 Boundary Settings (8), 195 Boundary Settings (12), 196 Boundary Settings (20), 196 Boundary Settings, final configuration, 197 Boundary Settings, Weak Constr. page, 197 Boundary Settings (2, 3, 5-7, 10, 13-16, 19). 195 conductivity matrix elements, 194 constants, 189 Constants edit window, 189, 189t Create Composite Object window, 191 Cross-section Line Data edit window, 202t default surface voltage distribution plot, 199 geometry, Subdomain Settings (1, 3, 4, 5), 193geometry, Subdomain Settings (2), 193 geometry with added rectangles, 192 Matrix elements edit window, 192 mesh. 198 mesh generation, 198 model plot  $V_{\rm H}$ , 203 Multiphysics Model Navigator window, 188 Plot Cross-Section Parameters, General page, 201 Plot Cross-Section Parameters, Line/Extrusion page, 202 Plot Parameters, Contour Data page, 200 postprocessing, 198, 200, 202 postprocessing animation, 202-203 Rectangle edit window, 190, 191t rectangle geometry, 190 solving, 198 start building, 186 subdomain settings, 191–192 surface voltage distribution plot, with contour lines, 201 geometry, 172-173 Matrix elements edit window, 175t mesh. 181

mesh generation, 180 model plot  $V_{\rm H}$ , 186 Multiphysics Model Navigator window, 171 Plot Parameters window, Contour Data page, 183 Points edit window, 175t postprocessing, 181 postprocessing animation, 185–186 Rectangle edit window, 173 rectangle geometry, 174 rectangle geometry with points, 174 results summary, 222t second variation on, 203-221 Animation Plot Parameters window, 204. 221 Application Mode Properties window, 205 Boundary Settings, 211–214 Boundary Settings (1), 211 Boundary Settings (8), 212 Boundary Settings (14), 213 Boundary Settings (18), 213 Boundary Settings (25), 214 Boundary Settings, final configuration, 215 Boundary Settings, Weak Constr. page, 214 Boundary Settings (2, 3, 5–7, 10–12, 15-17, 20, 21, 23, 24), 212 conductivity matrix elements, 210 constants, 206 Constants edit window, 206, 206t Create Composite Object window, 208 Cross-Section Line Data edit window, 217t default surface voltage distribution plot, 217 geometry, 206-207 geometry with added rectangles, 209 Matrix Elements edit window, 210t mesh, 215 mesh generation, 214 model plot  $V_{\rm H}, 220$ Multiphysics Model Navigator window, 205 Plot Cross-Section Parameters, General page, 219 Plot Cross-Section Parameters, Line/Extrusion page, 220 Plot Parameters, Contour Data page, 218 postprocessing, 216-218

postprocessing animation, 218, 221 purpose of, 203 Rectangle edit window, 207, 208t rectangle geometry, 207 Solver Parameters window, 216 solving, 216 start building, 205 subdomain settings, 208-210 Subdomain settings (1, 3, 4, 5, 6), 209Subdomain Settings (2), 210 surface voltage distribution plot, with contour lines. 219 Solver Parameters window, 182 solving, 180 Subdomain Settings, 173, 175, 176 summary and conclusions about, 221 - 222surface voltage distribution (2T), with contour lines. 184 2D inductive heating considerations, 2D axisymmetric coordinate system, 389-393 2D mixed-mode modeling considerations with, 321-322 implicit assumptions relative to, 322 2D coordinate systems, 322–323 2D modeling challenges with, 113 implicit assumptions with, 114 2D modeling modes, in COMSOL Multiphysics software, 113-114 2D Resistive Heating models, 327–345 with all boundary settings, 338 animation, final frame, 347 block with hole, 331 boundary conditions overview, 332 Boundary Settings (1, 4–8) edit window, 334 Boundary Settings (2, 3, 5-8) edit window, 337 Boundary Settings (2, 3) edit window, 335 Boundary Settings-Conductive Media DC (dc) Edit window, 337t Boundary Settings (1) edit window, 337 Boundary Settings (4) edit window, 338

2D Resistive Heating models (*continued*) Boundary Settings—Heat Transfer by Conduction (ht) edit window, 334t Circle edit window, 330 consolidating calculational parameters, 328 constants. 328 Constants edit window, 328, 328t coupling of Joule Heating in Conductive Media AC and Heat Transfer by Conduction Application Mode, 331 coupling of two modes within, 326, 331 Create Composite Object edit window, 331 Cross-Section Plot Parameters, General edit window. 343 Cross-Section Plot Parameters, Point edit window, 344 degrees Centigrade, 342 first variation on, 345-366 with all boundary settings, 360 animation, final frame, 367 Boundary Settings (1, 40) edit window, 355 Boundary Settings (2, 3, 5–8) edit window, 359 Boundary Settings (2, 3) edit window, 355 Boundary Settings-Conductive Media DC (dc) edit window, 359 Boundary Settings (1) edit window, 359 Boundary Settings (4) edit window, 360 Boundary Settings-Heat Transfer by Conduction (ht) edit window, 354t Combined Heat Transfer by Conduction (ht) boundary settings, 356 consolidation of calculational parameters in. 349 constants, 347 Constants edit window, 348t, 349 coupling of Joule Heating in Conductive Media DC Application and Heat Transfer by Conduction Application Mode, 351 Create Composite Object edit window, 350 Cross-Section Plot Parameters, General edit window. 364 Cross-Section Plot Parameters, Point edit window, 365

degrees Centigrade in, 361, 364 heater bar assembly, 351 materials demonstrated in, 345 mesh generation, 360 Model Navigator setup, 348 physical property values for Nichrome and copper required for conduction calculation in, 353 physics boundary settings: Conductive Media DC (dc), 358, 360 physics boundary settings: heat transfer by conduction (ht), 353-354 physics subdomain settings: Conductive Media DC (dc), 354, 356 physics subdomain settings: heat transfer by conduction (ht), 352 Plot Parameters edit window, 363 Plot Parameters window, 346, 366 postprocessing and visualization, 361,363 Rectangle edit window, 349t rectangles created, 350 Solver Parameters edit window, 362 solving, 361, 362 start building, 346-347 Subdomain Settings, Init edit window, 354 Subdomain Settings (1, 8) edit window, 352 Subdomain Settings-Conductive Media DC (dc), Init edit window, 358 Subdomain Settings—Conductive Media DC (dc) edit window, 356t, 357, 358t Subdomain Settings-Conductive Media DC (dc) edit window (1, 8), 357 Subdomain Settings edit window, 352t, 353t Subdomain Settings (2-7) edit window, 353 temperature versus time at x = 0, v = 0.1.3653D rendition of, 347 time-dependent solving of, 361 mesh generation, 338 mesh window, 339 Model Navigator setup, 327 model rectangle, 329 physics boundary settings: Conductive Media DC (dc), 335, 338

Index 851

physics boundary settings: heat transfer by conduction (ht), 334 physics subdomain settings: Conductive Media DC (dc), 334–335 physics subdomain settings: heat transfer by conduction (ht), 332 Plot Parameters, Arrow edit window, 342 Plot Parameters edit window, 341 Plot Parameters window, 346 postprocessing and visualization, 339, 341, 344 postprocessing animation, 344–345 rectangle and circle, 330 Rectangle edit window, 329 second variation on. 366-388 with all boundary settings, 381 Alumina, boundary conditions overview, 372 Alumina introduced in, 366 animation, final frame, 390 applications for materials and configuration in. 367 boundary conditions overview, 372 Boundary Settings (1, 40) edit window, 376 Boundary Settings (2, 3, 5, 26, 28, 39) edit window, 380 Boundary Settings (2, 3, 5, 28, 39) edit window, 376 Boundary Settings—Conductive Media DC (emdc) edit window, 379t Boundary Settings (1) edit window, 380 Boundary Settings (40) edit window, 381 Boundary Settings—Heat Transfer by Conduction (ht) edit window, 375t constants, 368 Constants edit window, 369, 369t coupling of Joule Heating in Conductive Media DC Application Mode and Heat Transfer by Conduction Application Mode. 371 Create Composite Object edit window, 371 Cross-Section Plot Parameter, General edit window. 386 Cross-Section Plot Parameters, Point edit window. 387 degrees Centigrade in, 383-384, 386

derivation of, 366 Free Mesh Parameters window, 382 heater bar assembly, 372 heat flux (proportional arrows), 388 import paths for, 369 mesh generation, 380 model mesh, 382 Navigator setup, 368 physical property values for copper, Nichrome, and Alumina required for conduction calculation in, 375 physics boundary settings: Conductive Media DC (emdc), 379 physics boundary settings: general heat transfer (htgh), 375 physics domain settings: general heat transfer (htgh), 371–372 physics subdomain settings: Conductive Media DC (emdc), 377-378 Plot Parameters edit window, 385 Plot Parameters window, 389 postprocessing and visualization, 383–385 postprocessing animation, 385, 388 Rectangle edit window, 370t, 371t rectangles created, 370 solving, 382, 383, 384 start building, 367 Subdomain Settings, Conductive Media DC (emdc) edit window (2, 4, 8, 10, 12), 378 Subdomain Settings (3, 5, 7, 9, 11) edit window, 373 Subdomain Settings—Conductive Media DC (emdc), Init edit window, 379 Subdomain Settings-Conductive Media DC (emdc) edit window, 378t Subdomain Settings—Conductive Media DC (emdc) edit window (1, 13), 377 Subdomain Settings edit windows, 373t, 374*t*, 375*t* temperature versus time at x = 0, y = 0.1, 3873D rendition of, 368 time-dependent solving of, 383 Solver Parameters edit window, 340 solving, 339, 340

2D Resistive Heating models (*continued*) start building, 327 Subdomain edit window, 332*t* Subdomain Settings: Conductive Media DC (dc) Edit window, 335*t* Subdomain Settings, Init edit window, 333, 336 Subdomain Settings edit window, 333, 336 summary and conclusions about, 388 temperature and heat flux, 343 temperature versus time at x = 0, y = 0.4, 3453D rendition of, 327 time-dependent solving of, 339

# U

UMFPACK. See Parametric solver (UMFPACK) Uniform Color radio button, 181, 526, 562, 647 Union button, 506 Union command, 2D AC Generator model, static, 502, 509 Unregistered class of access, MatWeb, 43, 45 UNS C10100 properties, Materials/Coefficient Library, 43 Untransformed AC power, 497 Use selected points as destination check box, 550 Use weak constraints check box, 176, 548

# V

Vacuum, 317 Vacuum flask, invention of, 265 Vacuum flask containers, 265 Vector dot product current (K\*nJ\_dc), 165 Voltage difference methods, determining between upper/lower surfaces, 181 Voltage divider, 574 von Guericke, Otto, 611

### W

"War of Currents," 494–498
Wave equation PDE, achieving desired behavior for and transformation of, 672, 674
Wave equation problems, PML methodology applied to, 671
Wave equation solution examples inside modeling domain, 675
inside PML domain, 675 Weak constraints, 176, 221 Weak constraints pull-down list, 172 Westinghouse, George, 494 Work-Plane settings 3D thin layer resistance model: thin layer approximation and use of, 580 3D thin layer resistance modeling, thin layer subdomain. 595 Work-Plane Settings edit window 3D magnetic field of a Helmholtz coil model. 638 3D magnetic field of a Helmholtz coil with a magnetic test object model, 654 3D thin layer resistance model: thin layer approximation, 580 3D thin layer resistance model: thin layer subdomain, 597

Weak Constr. tab, 176

# Х

X-Axis Data edit window
3D thin layer resistance model: thin layer approximation, *591*3D thin layer resistance model: thin layer subdomain, *608*X-ray tubes, 3D electrostatic potential models and, 612, 622, 633
x-y-z coordinate orientation, for COMSOL modeling calculations, 2, *3*x-y-z to spherical coordinate conversion, equations for, 2–3

# Ζ

Zoom Extents button, 136, 155, 172, 189, 191, 207, 231, 243, 254, 268, 329, 350, 370, 398, 466, 479, 505, 536, 579, 580, 595, 596, 615, 626, 639, 655, 677, 694, 710, 727, 753, 774 COMSOL Multiphysics workspace window, 92 Zoom Extents icon, 28 1D dual-pane workspace after clicking, 19 1D triple-pane workspace after clicking, 29

