COMSOL Multiphysics

Reference Manual

5.5

COMSOL

COMSOL Multiphysics Reference Manual

Version: COMSOL 5.5

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case. Other useful links include:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM020005

Contents

Chapter 1: Introduction

About COMSOL Multiphysics 39

The COMSOL Multiphysics Modules and Interfacing Options 40

COMSOL Documentation and Help 41

About the Documentation Set 41

The Help Window and Topic-Based Help 43

The Documentation Window 46

Searching Help and Documentation Content 47

The About COMSOL Multiphysics Box 49

Checking for Product Software Updates 50

Typographical Conventions 50

Overview of the Reference Manual 53

Chapter 2: The COMSOL Modeling Environment

The COMSOL Desktop 58

Basic Navigation 61

Adjusting Window Location and Size on the Desktop 62

The COMSOL Desktop Windows 66

The COMSOL Desktop Menus and Toolbars 68

Windows Toolbars and Menus 68

Cross Platform (macOS and Linux) Toolbars and Menus 69

Features Available on Toolbars and From Menus. 70

The Messages Window 75

About the COMSOL Model File Formats. 76

Saving COMSOL Files 77

Saving and Opening Recovery Files 79

The Root Settings and Properties Windows 80

Unit Systems 81

Searching and Finding Text 82

The Application Libraries Window 84

The Applications Folders 86

Running or Opening a Model or Application and Its Documentation . . . 86

Downloading MPH-Files With or Without Solutions 88

Searching the Application Libraries 88

The Application Library Update Window. 90

The Part Library Update Window 92

The Physics Interfaces 93

Introduction to the Physics Interfaces 93

Physics Groups and Subgroups. 93

Adding and Inserting Physics Interfaces 96

Physics Interface Guide 98

Common Physics Interface and Feature Settings and Nodes. 101

Creating a New Model 105
Open a New Window to Begin Modeling. 105
The Model Wizard 105

Toolbars and Keyboard Shortcuts 110
Home Toolbar. 110
Definitions Toolbar 113
Geometry Toolbar 115
Sketch Toolbar 119
Work Plane Modal Toolbar 121
Materials Toolbar. 122
Physics Toolbar 123
0D Component Toolbar 124
Mesh Toolbar 125
Study Toolbar 127
Results Toolbar 128
Plot Group Contextual Toolbar 131
Report Group Contextual Toolbar 132
Template Group Contextual Toolbar 133
View Toolbar 133
Developer Toolbar 134
Keyboard Shortcuts. 135

Chapter 3: Building a COMSOL Multiphysics Model

Building Models in the Model Builder 141
The Model Builder 141
About the Sequence of Operations 142
The Global Definitions Node 142
The Component Node 142
Adding Extra Dimensions to a Model 146
Branches and Subbranches in the Tree Structure. 147
Settings and Properties Windows for Feature Nodes 150
Displaying Node Names, Tags, and Types in the Model Builder 152
Opening Context Menus and Adding Nodes 154
The Physics Nodes 155
Physics Interface Default Nodes 157
Physics Interface Node Context Menu Layout 157
Physics Exclusive and Contributing Node Types 159
Physics Node Status. 161
Dynamic Nodes in the Model Builder 162
Physics Symbols 164
Errors and Warnings 167

Modeling Development Tools 170
Overview. 170
Creating and Running Methods in Models 170
Method Calls 171
Creating and Using Settings Forms and Dialogs 171
Creating Add-ins 172
Using Add-ins 172
The Add-in Libraries Window 172

Comparing Models and Applications 173

Working with Nodes in the Model Builder 175

Moving Nodes in the Model Builder. 175
Copying, Pasting, and Duplicating Nodes 175
Undoing and Redoing Operations 177
Going to the Source Node 177
Clearing Sequences and Deleting Sequences or Nodes 178
Disabling or Enabling Nodes 178

Modeling Guidelines 180

Selecting Physics Interfaces 180
Using Symmetries 181
Effective Memory Management 181
Selecting an Element Type 182
Analyzing Model Convergence and Accuracy 183
Achieving Convergence When Solving Nonlinear Equations. 183
Avoiding Strong Transients 184
Physics-Related Checks and Guidelines 184
Results With Unphysical Values 185

Multiphysics Modeling Workflow 188

Creating a Multiphysics Model 188
Advantages of Using the Predefined Multiphysics Interfaces 190
The Add Multiphysics Window 190
The Multiphysics Branch 191
Uncoupling a Multiphysics Coupling 191
Model Inputs and Multiphysics Couplings 192

Specifying Model Equation Settings 194

Specifying Equation Coefficients and Material Properties 194
Modeling Anisotropic Materials 195
Specifying Initial Values. 195
Equation View 196
Physics Nodes — Equation Section 199

Boundary Conditions 201

Boundary Condition Types 201
Physics Interface Boundary Types 203
Continuity on Interior Boundaries 203
Physics Interface Axial Symmetry Node 204
Constraint Reaction Terms 204
Constraint Settings 205
Excluded Surfaces, Excluded Edges, and Excluded Points 206
Weak Constraints 206
Periodic Boundary Conditions 208

Computing Accurate Fluxes 211

Flux Computation Methods 211
Flux Calculation Example — Heat Transfer Model 212

Using Load Cases 214

About Load Cases 214
Defining Load Groups and Constraint Groups. 214

Load Group 216
Constraint Group 216
Defining and Evaluating Load Cases 217

Numerical Stabilization **219**
About Numerical Stabilization in COMSOL 219
Consistent Stabilization and Inconsistent Stabilization Sections on Settings Windows 219
An Example of Stabilization 220
Stabilization Techniques 222
References for Stabilization Techniques 225

Using Units **226**
Using Standard Unit Prefixes and Syntax 226
SI Base, Derived, and Other Units 228
Special British Engineering Units 232
Special CGSA Units 232
Special EMU Units 233
Special ESU Units. 233
Special FPS Units 234
Special IPS Units 234
Special MPa Units. 234
Special Gravitational IPS Units 235
Switching Unit System 235
About Temperature Units 235
About Editing Geometry Length and Angular Units 236
Units and Space Dimensions 237

Chapter 4: Customizing the COMSOL Desktop

Customizing a Model **240**
Customizing the Desktop Layout. 240
Changing Fonts and the Desktop Language 241
Editing Node Properties, Names, and Labels 241
Custom Grouping of Nodes 243
Grouping Nodes by Space Dimension and Type 244
Setting the Unit System for Models 246
Checking and Controlling Products and Licenses Used 246

Preferences Settings **248**
The Preferences Dialog Box. 248
Importing and Exporting Preferences 249

Showing More Options **250**

Chapter 5: Global and Local Definitions

Global Definitions, Geometry, Mesh, and Materials **254**
Global Definitions 254
Geometry Parts 254
Mesh Parts 255
Global Materials 255

Definitions 256
About Parameters, Variables, Variable Utilities, and Expressions 257
Parameters 258
Parameter Cases 260
Variables 260
Variable Utilities 262
Expression Operator 262
Operator Contribution 263
Default Model Inputs 263
Model Input 265
State Variables 265
Participation Factors 266
Response Spectrum 266
Common Settings for the Definitions Nodes 267

Operators, Functions, and Constants 269
Unary, Binary, and List Operators and Their Precedence Rules 269
Mathematical and Numerical Constants 270
Mathematical Functions 270
Physical Constants 272
Built-In Operators 273

Predefined and Built-In Variables 293
Predefined Physics Variables. 293
Variable Naming Convention and Namespace 293
Variable Classification and Geometric Scope 294
Built-In Global Variables 294
Geometric Variables, Mesh Variables, and Variables Created by Frames . 296
Material Group Indicator Variables 301
Shape Function Variables 302
Solver Variables 304
Entering Ranges and Vector-Valued Expressions 304
Summary of Built-In Variables With Reserved Names 306

Mass Properties 313
Introduction. 313
Mass Properties 313
Mass Contributions 315

Functions 316
Switch for Functions 316
About User-Defined Functions 317
Common Settings for the Function Nodes 317
Analytic 319
Elevation (DEM) 319
External 320
Gaussian Pulse 321
Image 321
Interpolation 322
MATLAB 328
Piecewise. 328
Ramp 330
Random 330
Rectangle. 331

Step 331
Thermodynamics Property Package 332
Triangle 332
Waveform 332
Specifying Discontinuous Functions 333

Matrices and Matrix Operations **335**
Matrix 335
Matrix Inverse 335
Matrix Diagonalization 336
Matrix Decomposition (SVD) 337
Vector Transform 338
Matrix Transform 339
Tensors in COMSOL Multiphysics 341

Nonlocal Couplings and Coupling Operators **345**
About Nonlocal Couplings 345
General Extrusion 351
Linear Extrusion 352
Boundary Similarity 352
One-Point Map 353
Two-Point Map 353
Edge Map 354
Identity Mapping 354
General Projection 354
Linear Projection 355
Integration 356
Average 357
Maximum and Minimum 357
Common Settings for Nonlocal Couplings 358

Coordinate Systems **360**
About Coordinate Systems 360
Base Vector System 361
Boundary System 363
Reverse Normal 364
Domain Normal 364
Combined System 365
Composite System 365
Cylindrical System 366
Mapped System 367
Rotated System 367
Scaling System 369
Spherical System 370
System from Geometry 371

Identity and Contact Pairs **372**
About Identity and Contact Pairs 372
Identity Pairs 373
Contact Pair 375

Probes **377**
About Probes 377
Common Settings for Probes 378

Domain Probe, Boundary Probe, and Edge Probe 379
Domain Point Probe 380
Boundary Point Probe 381
Point Probe Expression 381
Global Variable Probe 381

Infinite Elements, Perfectly Matched Layers, and Absorbing Layers 382
Simulation of Infinite Domains 382
Standard Geometry Configurations 383
Manual Settings for Nonstandard Geometries 386
Note on Availability 387
PML Implementation 387
PMLs in Multiphysics 389
Perfectly Matched Layer 389
Known Issues When Modeling Using PMLs 391
Infinite Element Implementation 392
Infinite Element Domain 393
Known Issues When Modeling Using Infinite Elements 394
Absorbing Layer 395
References for PMLs and Infinite Element Domains 396

Reduced-Order Modeling 397
Introduction. 397
Reduced-Order . . . . Model Inputs 397
Reduced-Order Model Outputs 398
Modal Reduced-Order Models. 399
Global Reduced Model Inputs 400
Frequency Domain, Modal Reduced-Order Model 401
Time Dependent, Modal Reduced-Order Model 402
Frequency Domain, AWE Reduced-Order Model 403
Random Vibration 404
Importing Reduced-Order Models 404
The Reduced-Order Model Toolbar 404

Chapter 6: Visualization and Selection Tools

Working with Geometric Entities 408
About Geometric Entities 408
The Graphics Window. 410
Basic Selection Concepts 411
About Highlighted Geometric Entities in the Graphics Window 411
Selection Colors 415
About Selecting Geometric Entities 416
The Selection List Window 421
Selecting and Clearing Selection of Geometric Entities. 423
The Graphics Window Toolbar Buttons and Navigation 426
Using A 3D Mouse from 3Dconnexion 435
Customizing the Graphics Toolbars 436

Creating Named Selections 437
Introduction. 437
Creating Named Selections 439

Copying and Pasting Selection Lists 441
Adjacent 442
Ball 442
Box 444
Cylinder 444
Disk 445
Explicit. 446
Union, Intersection, Difference, and Complement 447
Creating Named Selections in the Geometry Sequence 448
Adjacent Selection (Geometry Sequences) 450
Ball Selection (Geometry Sequences) 451
Box Selection (Geometry Sequences) 451
Cylinder Selection (Geometry Sequences) 452
Disk Selection (Geometry Sequences) 453
Explicit Selection (Geometry Sequences) 453
Union Selection, Intersection Selection, Difference Selection, and Complement Selection (Geometry Sequences) 455
Cumulative Selections 456

User-Defined Views **457**
View (1D and 2D) 457
Axis (2D and 2D Axisymmetric) 458
Axis (1D and 1D Axisymmetric) 459
View (3D) 459
Camera 461
About the 3D View Light Sources and Attributes. 463
Directional Light 464
Point Light 466
Spotlight 467
Headlight 468
Hide for Geometry 469
Hide for Physics 470
Hide for Mesh Import 470

Chapter 7: Geometry Modeling and CAD Tools

Creating a Geometry for Analysis **474**
Overview of Geometry Modeling Concepts. 474
Techniques for Creating Geometries 475
Associative Geometry and Selections of Geometry Objects 476
Choosing the Right Space Dimension 476
Removing Interior Boundaries 478

Working with Geometry Sequences **479**
The Geometry Nodes 479
The Geometry Toolbar 479
The Sketch Toolbar 481
The Geometry Node 483
Plane Geometry 486
Creating a Geometry Sequence 487
Editing and Building Geometry Nodes 488
Exporting a Geometry 490

Measuring Geometry Objects 491
The Form Union/Assembly Node — Uniting the Geometry. 491
Using Geometry Parts 494
Geometry Part Settings 495
Loaded Part Settings 496
Local Parameters 496

Part Libraries **497**
Introduction. 497
Using Part Libraries 497
Creating a Part. 500

Geometric Primitives **502**
Bézier Polygon. 503
Block 505
Circle 506
Circular Arc. 507
Composite Curve 508
Cone 509
Cubic Bézier 510
Cylinder 511
Eccentric Cone 512
Ellipse 514
Ellipsoid 515
Helix 516
Hexahedron. 519
Interpolation Curve 519
Interval 521
Line Segment 522
Parametric Curve. 523
Parametric Surface 524
Point 526
Polygon 526
Pyramid 528
Quadratic Bézier 529
Rectangle. 530
Sphere 531
Square 532
Tetrahedron 533
Torus 534
Composite Object (Backward Compatibility) 535

Geometry Operations **536**
Array 539
Chamfer 540
Compose. 541
Convert to Curve 542
Convert to Point 543
Convert to Solid 544
Convert to Surface 545
Copy 546
Cross Section 547
Deformed Configuration 548
Delete Entities. 549

Difference 549
Edit Object 550
Extrude 552
Fillet. 553
If, Else If, Else, End If. 554
Import 555
Intersection 558
Mirror 559
Move 560
Parameter Check. 561
Part Instance 561
Partition Objects 563
Partition Domains 564
Partition Edges. 566
Partition Faces 567
Revolve 568
Rigid Transform 569
Rotate 570
Scale 572
Split 573
Sweep 573
Tangent 576
Union 576
Work Plane 577
Using Work Planes 582

Virtual Geometry and Mesh Control Operations **586**
Collapse Edges. 587
Collapse Faces 587
Collapse Face Regions 588
Form Composite Domains 588
Form Composite Edges 588
Form Composite Faces 589
Ignore Edges 589
Ignore Faces. 590
Ignore Vertices 590
Merge Edges 591
Merge Vertices 591
Mesh Control Domains 591
Mesh Control Edges. 592
Mesh Control Faces. 592
Mesh Control Vertices. 592
Remove Details 593

Geometry Modeling Examples **595**
Creating a 2D Geometry Model 595
Creating a 3D Geometry Model 600
Forming Composite Edges and Faces by Ignoring Vertices and Edges . . 608
Merging Vertices by Collapsing Edges 612

Chapter 8: Meshing

Creating a Mesh for Analysis **616**

Meshing Concepts . . . 616
Mesh Elements for 1D, 2D, and 3D Geometries . . . 617
Free (Unstructured) Meshing . . . 618
Structured Meshes . . . 619
About Swept Meshes . . . 620
Mesh Control Entities . . . 621
The Mesh Toolbar . . . 622
Adding, Editing, and Building Meshing Sequences . . . 623
Mesh (Node) . . . 624
Using Mesh Parts . . . 625
Mesh Part Settings . . . 625
Meshes Generated by Adaptation . . . 625
The Mesh Statistics Window . . . 626

Meshing Techniques **628**

Choosing a Meshing Sequence Type. . . . 628
Mesh Element Quality and Size . . . 629
Avoiding Inverted Mesh Elements . . . 631
Troubleshooting Boundary Layer Mesh Generation . . . 633
Troubleshooting Free Tetrahedral Mesh Generation . . . 633

Meshing Operations and Attributes **635**

Adapt . . . 636
Boundary Layers . . . 639
Boundary Layer Properties . . . 640
Convert . . . 641
Corner Refinement . . . 642
Copy Domain . . . 642
Copy Edge . . . 644
Copy Face . . . 645
Copy . . . 646
Distribution . . . 647
Edge. . . . 648
Edge Groups . . . 649
Edge Map . . . 649
Free Quad . . . 650
Free Tetrahedral . . . 651
Free Triangular . . . 653
Mapped . . . 654
One-Point Map . . . 655
Reference . . . 655
Refine . . . 656
Scale . . . 657
Size . . . 657
Size Expression . . . 659
Swept . . . 663
Two-Point Map . . . 665

Importing and Exporting Meshes **666**

About Mesh Export, Import, and Operations on Imported Meshes . . . 666

Exporting Meshes 666
Importing Meshes 668
Creating Geometry from Mesh 670
Creating or Modifying Entities of Imported Meshes 671
Creating or Modifying Elements of Imported Meshes 672
Using Operations on an Imported Mesh 672
Ball 674
Box 675
Create Domains 675
Create Edges 676
Create Faces 677
Create Vertices 677
Cylinder 677
Delete Entities 678
Detect Faces 678
Fill Holes 679
Finalize. 679
Import 679
Join Entities 682
Logical Expression 683

Meshing Examples **684**
Generating a 3D Swept Mesh 684
Using Mesh Control Entities to Control Element Size 686
Using Structured and Unstructured Mesh with Boundary Layers 687

Chapter 9: Materials

Materials Overview **692**
About Materials and Material Properties 692
About the Material Databases 693
About Using Materials in COMSOL Multiphysics 694

Working with Materials **698**
The Material Browser Window 698
The Add Material Window 699
Materials 701
The Settings Window for Material 702
Property Groups 706
Material Link 709
Switch for Materials 709
Layered Material 710
Layered Material Link 714
Layered Material Stack. 717
Layered Material Link (Subnode) 719
Single-Layer Materials 720

Material Properties Reference **722**
About Model Inputs 722
About the Output Material Properties 722
Acoustics Material Properties 724
Electrochemistry Material Properties 725

Electromagnetic Models 726
Equilibrium Discharge 727
Gas Models 727
Geometric Properties (Shell) 728
Magnetostrictive Models 728
Piezoelectric Models 729
Piezoresistive Models 729
Semiconductors Material Properties 729
Solid Mechanics Material Properties. 733
Solid Mechanics Material Properties: Nonlinear Structural Materials Module 735
Solid Mechanics Material Properties: Fatigue Module 739
Solid Mechanics Material Properties: Geomechanics Material Model . . 740
Thermal Expansion Material Properties 741
External Material Properties 741

User-Defined Materials and Libraries **742**
Importing a Material Library. 742
Creating a New Material Library and Adding and Editing Materials . . . 742
Restoring a Deleted User-Defined Library 745

Using Functions in Materials **746**
Adding a Function to the Material 746
Defining an Analytic Function 746

Working with External Materials **749**
The External Material Model 749
Using External Materials in Physics Interfaces 749
Built-in Material Function Interface Types 751
Library of Utility Functions for Structural Mechanics 757
How to Compile and Link an External Material Model. 757
Known Issues for External Materials 759
External Material 759

Module-Specific Material Databases **762**
AC/DC Materials Database 762
Batteries and Fuel Cells Materials Database 763
Bioheat Materials Database 764
Building Materials Database 764
Equilibrium Discharge Materials Database 765
Liquids and Gases Materials Database 765
MEMS Materials Database 766
Nonlinear Magnetic Materials Database 767
Piezoelectric Materials Database 770
Piezoresistivity Materials Database 771
Optical Materials Database 771
RF Materials Database 772
Semiconductor Materials Database 772
Thermoelectric Materials Database 772
References for the Materials Databases 772

Chapter 10: The AC/DC Interfaces

The Electromagnetics Interfaces 778

Fundamentals of Electromagnetics 779

Maxwell's Equations . . . 779
Constitutive Relations . . . 779
Potentials . . . 780
Material Properties . . . 781
About the Boundary and Physics Interface Conditions . . . 781
Phasors . . . 782
Effective Nonlinear Magnetic Constitutive Relations . . . 783
Electromagnetic Forces . . . 783
References for Electromagnetic Theory . . . 783

Theory of Electrostatics 784

Charge Relaxation Theory . . . 784
Electrostatics Equations . . . 785
The Electrostatics Interface in Time Dependent or Frequency Domain Studies 786

Theory of Electric Currents 787

Electric Currents Equations in Steady State . . . 787

Theory of Magnetic Fields 788

Magnetostatics Equation . . . 788
Frequency Domain Equation . . . 788
Transient Equation . . . 789
Maxwell's Equations . . . 789
Magnetic and Electric Potentials . . . 789
Gauge Transformations . . . 790
Selecting a Particular Gauge . . . 790
The Gauge and Equation of Continuity for Dynamic Fields . . . 790
Time-Harmonic Magnetic Fields . . . 790

The Electrostatics Interface 792

Domain, Boundary, Edge, Point, and Pair Nodes for the Electrostatics Interface 793
Charge Conservation . . . 795
Conduction Loss (Time-Harmonic) . . . 795
Initial Values . . . 796
Space Charge Density . . . 796
Zero Charge . . . 796
Ground . . . 796
Electric Potential . . . 797
Surface Charge Density . . . 797
External Surface Charge Accumulation . . . 797
Electric Displacement Field . . . 798
Periodic Condition . . . 798
Thin Low Permittivity Gap . . . 799
Line Charge . . . 799
Line Charge (on Axis) . . . 799
Line Charge (Out-of-Plane) . . . 800
Point Charge . . . 800
Point Charge (on Axis) . . . 801

Change Cross Section 801
Change Thickness (Out-of-Plane). 801
Charge Conservation, Piezoelectric 802

The Electric Currents Interface 803
Domain, Boundary, Edge, Point, and Pair Nodes for the Electric Currents Interface 804
Current Conservation 806
Initial Values. 807
External Current Density. 807
Current Source 807
Electric Insulation. 808
Boundary Current Source 808
Normal Current Density 808
Distributed Impedance. 809
Contact Impedance 809
Sector Symmetry 810
Line Current Source 811
Line Current Source (on Axis). 811
Point Current Source 811
Point Current Source (on Axis) 812
Piezoresistive Material 812

The Magnetic Fields Interface 814
Domain, Boundary, Point, and Pair Nodes for the Magnetic Fields Interface 815
Ampère's Law 816
Ampère's Law, Magnetostrictive 818
Initial Values. 819
External Current Density. 819
Velocity (Lorentz Term) 819
Magnetic Insulation 820
Magnetic Field 821
Surface Current Density 821
Magnetic Potential 821
Perfect Magnetic Conductor 822
Line Current (Out-of-Plane) 822
Line Current (on Axis). 822

Chapter 11: The Pressure Acoustics Interface

Fundamentals of Acoustics 824
Acoustics Explained 824
Examples of Standard Acoustics Problems 824
Different Acoustic Interfaces 825
Mathematical Models for Acoustic Analysis 826

The Pressure Acoustics, Frequency Domain Interface 828
Domain, Boundary, Edge, Point, and Pair Nodes for the Pressure Acoustics, Frequency Domain Interface 830
Pressure Acoustics 831
Sound Hard Boundary (Wall) 831
Initial Values. 832
Monopole Source 832
Dipole Source 832

Normal Acceleration 832
Sound Soft Boundary 833
Pressure 833
Impedance 833
Symmetry 834
Plane Wave Radiation 834
Spherical Wave Radiation. 834
Cylindrical Wave Radiation 834
Incident Pressure Field. 835
Interior Sound Hard Boundary (Wall) 835
Periodic Condition 835
Axial Symmetry 836
Continuity 836

Theory for Pressure Acoustics, Frequency Domain **837**
Frequency Domain Study. 837
Eigenfrequency Study 838
References for the Pressure Acoustics, Frequency Domain Interface . . 838

Chapter 12: The Chemical Species Transport Interfaces

Theory for Transport of Diluted Species **842**
Mass Balance Equation 842
Equilibrium Reaction Theory 843
Convective Term Formulation. 844
Solving a Diffusion Equation Only. 845
Mass Sources for Species Transport. 845
Adding Transport Through Migration 846
Supporting Electrolytes 847
Crosswind Diffusion 848
Danckwerts Inflow Boundary Condition 848
Mass Balance Equation for Transport of Diluted Species in Porous Media 849
Convection in Porous Media 850
Diffusion in Porous Media 851
Dispersion 852
Adsorption 853
Reactions. 854
Mass Transport in Fractures 855
References 856

The Transport of Diluted Species Interface **857**
The Transport of Diluted Species in Porous Media Interface 860
Domain, Boundary, and Pair Nodes for the Transport of Diluted Species Interface 861
Transport Properties 862
Turbulent Mixing 863
Initial Values. 864
Mass-Based Concentrations 864
Reactions. 864
No Flux 865
Inflow 865
Outflow 866
Concentration 866

Flux . . . 867
Symmetry . . . 867
Flux Discontinuity . . . 868
Partition Condition . . . 868
Periodic Condition . . . 868
Line Mass Source . . . 869
Point Mass Source . . . 869
Open Boundary . . . 870
Thin Diffusion Barrier . . . 870
Thin Impermeable Barrier . . . 870
Equilibrium Reaction . . . 870
Surface Reactions. . . . 871
Surface Equilibrium Reaction . . . 872
Fast Irreversible Surface Reaction . . . 872
Porous Electrode Coupling . . . 872
Reaction Coefficients . . . 872
Electrode Surface Coupling . . . 873
Porous Media Transport Properties. . . . 873
Adsorption . . . 875
Partially Saturated Porous Media . . . 876
Volatilization . . . 877
Reactive Pellet Bed . . . 878
Reactions (Reactive Pellet Bed) . . . 880
Species Source. . . . 881
Hygroscopic Swelling . . . 881
Fracture . . . 881

The Reacting Flow, Diluted Species Multiphysics Interface 883
The Reacting Laminar Flow, Diluted Species Interface . . . 883
The Reacting Flow, Diluted Species Coupling Feature . . . 884
Physics Interface Features . . . 885

Chapter 13: The Fluid Flow Interface

Theory of Laminar Flow 888
General Single-Phase Flow Theory . . . 888
Compressible Flow . . . 890
Weakly Compressible Flow . . . 890
The Mach Number Limit . . . 890
Incompressible Flow . . . 891
The Reynolds Number. . . . 891
Theory for the Wall Boundary Condition . . . 892
Prescribing Inlet and Outlet Conditions . . . 893
Normal Stress Boundary Condition . . . 894
Pressure Boundary Condition . . . 895
Mass Sources for Fluid Flow. . . . 896
Numerical Stability — Stabilization Techniques for Fluid Flow . . . 897
Solvers for Laminar Flow . . . 898
Pseudo Time Stepping for Laminar Flow Models . . . 900
Discontinuous Galerkin Formulation . . . 901
Particle Tracing in Fluid Flow . . . 901
References for the Single-Phase Flow, Laminar Flow Interfaces . . . 902

The Single-Phase Flow, Laminar Flow Interface **904**
The Laminar Flow Interface 904
Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow 907
Fluid Properties 907
Volume Force 908
Initial Values. 909
Wall. 909
Inlet 910
Outlet 911
Symmetry 912
Open Boundary 912
Boundary Stress 913
Periodic Flow Condition 914
Flow Continuity 914
Pressure Point Constraint 915
Point Mass Source 915
Line Mass Source 915
Gravity. 916

Chapter 14: The Heat Transfer Interfaces

Theory for Heat Transfer **918**
Theory for Heat Transfer in Solids 918
Theory for Heat Transfer in Fluids 918

About the Heat Transfer Interfaces **920**
Space Dimensions 920
Study Types 920
Versions of the Heat Transfer Physics Interface 920
Settings for the Heat Transfer Interface 921
References for the Heat Transfer Interfaces. 922

The Heat Transfer in Solids Interface **923**
Feature Nodes for the Heat Transfer in Solids Interface 923

The Heat Transfer in Fluids Interface **925**
Feature Nodes for the Heat Transfer in Fluids Interface 925

The Heat Transfer in Solids and Fluids Interface **927**

The Joule Heating Interface **928**
Electromagnetic Heating 929

The Nonisothermal Flow and Conjugate Heat Transfer Interfaces **931**
Settings for Physics Interfaces and Coupling Feature 931
Nonisothermal Flow 931
Physics Interface Features 933

Domain Features **934**
Cross Section 934
Thickness. 934
Heat Source. 935

Fluid. 936
Solid 938
Initial Values. 939
Translational Motion 940

Boundary Features **941**
Boundary Heat Source. 941
Continuity 942
Heat Flux. 942
Line Heat Source on Axis 943
Outflow 943
Periodic Condition 944
Surface-to-Ambient Radiation 944
Symmetry 945
Temperature 946
Thermal Insulation 946
Thin Layer 947

Edge and Point Features **949**
Line Heat Source. 949
Point Heat Source 949
Point Heat Source on Axis 950

Heat Transfer Variables **951**
Predefined Variables 951
Global Variables 952
Domain Heat Fluxes 954
Boundary Heat Fluxes 955
Internal Boundary Heat Fluxes. 955
Domain Heat Sources 956
Boundary Heat Sources 956
Line and Point Heat Sources 957

Using the Boundary Conditions for the Heat Transfer Interfaces **958**
Temperature and Heat Flux Boundary Conditions 958
Overriding Mechanism for Heat Transfer Boundary Conditions 958

Heat Transfer Consistent and Inconsistent Stabilization Methods **961**
Consistent Stabilization 961
Inconsistent Stabilization 961

Handling Frames in Heat Transfer **962**
Physics Feature Nodes and Definition Frame 962
Definition Frame of Domain Nodes 963
Definition Frame of Boundary Nodes 963
Definition Frame of Edge and Point Nodes 963

Solver Settings **964**
Linear Solver 964
Nonlinear Solver 965
Time-Dependent Study Step 966
Guidelines for Solving Multiphysics Problems 967

Chapter 15: Solid Mechanics

The Solid Mechanics Interface 970

Domain, Boundary, Edge, Point, and Pair Nodes for Solid Mechanics . . 972
Initial Values. 973
Change Thickness 974
Linear Elastic Material 974
Damping 976
Free. 977
Prescribed Displacement 977
Fixed Constraint 978
Roller 979
Symmetry 979
Rigid Motion Suppression. 980
Body Load 980
Boundary Load. 981
Edge Load 982
Point Load 982
Point Load (on Axis) 983
Periodic Condition 983
Ring Load. 984

Chapter 16: Equation-Based Modeling

The Mathematics Interfaces 988

Modeling with PDEs 990

About Equation Forms. 990
Notational Conventions 990
PDE Interface Variables 992
The General Form PDE 992
The Coefficient Form PDE 993
Multiple Dependent Variables — Equation Systems 996
Solving Time-Dependent Problems 1001
Solving Eigenvalue Problems. 1003
About Weak Form Modeling 1004
Introduction to the Weak Form 1005
The Weak Form PDE 1006
Specifying and Interpreting Boundary Conditions 1007
Symmetric and Nonsymmetric Constraints 1011

The PDE Interfaces 1013

Adding a PDE Interface to a Component 1013
Settings for the Discretization Sections 1014
The Coefficient Form PDE Interfaces 1017
The General Form PDE Interfaces 1018
The Weak Form PDE Interfaces 1019
The Classical PDE Interfaces 1020
Domain, Boundary, Pair, Edge, and Point Conditions for PDEs. 1021
Initial Values. 1022
Coefficient Form PDE 1022
General Form PDE 1023

Weak Form PDE 1023
Source, Edge Source, and Point Source 1024
Classical PDE Domain Nodes 1024
Dirichlet Boundary Condition 1025
Constraint 1025
Excluded Points, Excluded Edges, Excluded Surfaces 1026
Flux/Source 1026
Zero Flux. 1026
No Flux 1027
No Diffusive Flux. 1027
Interior Dirichlet Boundary Condition. 1027
Interior Flux/Source. 1027
Periodic Condition 1028
Destination Selection 1029

The PDE, Boundary Elements Interface 1030
About Boundary Elements Interfaces 1030
Finite and Infinite Voids 1030
The PDE, Boundary Elements Interface Main Node 1031
Domain and Boundary Physics for the PDE, Boundary Elements Interface 1034
PDE, Boundary Elements 1034
Initial Values. 1035
Zero Flux. 1035
Dirichlet Boundary Condition 1035
Flux/Source 1035

Theory for the Boundary Elements PDE 1037
About the Boundary Element Method 1037
The Fundamental Solution to Laplace's Equation 1037
Derivation of the Boundary Integral Representation of the Solution to Laplace's Equation 1038
Derivation of the Boundary Integral Equations. 1039
The Weak Form Implementation of the Boundary Integral Equations . . 1040
Reference 1041

The Wave Form PDE Interface 1042
Domain and Boundary Physics for the Wave Form PDE Interface . . . 1043
Wave Form PDE 1043
Initial Values. 1046
Zero Flux. 1046
Flux/Source 1046
Interior Flux. 1046
Interior Flux Split. 1047
Interior Source 1048

Theory for the Wave Form PDE 1049
Derivation of the Weak Form of the Wave Form PDE 1049
Time Explicit Integrator 1050
Local Time Stepping. 1051
Reference for the Wave Form PDE Interface 1052

About Auxiliary Equation-Based Nodes 1053
Weak Contribution (PDEs and Physics) 1053
Weak Contribution on Mesh Boundaries. 1054
Auxiliary Dependent Variable 1055

About Explicit Constraint Reaction Terms 1055
Pointwise Constraint 1057
Weak Constraint 1058
Discretization (Node) 1060

Modeling with ODEs and DAEs **1061**
Adding ODEs, DAEs, and Other Global Equations 1061
Solving ODEs: An Example 1062
Solving Algebraic and Transcendental Equations: An Example 1062
Distributed ODEs and DAEs 1063

The ODE and DAE Interfaces **1064**
The Global ODEs and DAEs Interface 1064
About ODEs, Initial-Value Problems, and Boundary-Value Problems . . 1064
Global Equations 1065
Global Constraint 1067
Weak Contribution (ODEs and DAEs) 1067
The Distributed ODEs and DAEs Interfaces. 1067
Distributed ODE 1068
Algebraic Equation 1068

The Events Interface **1069**
Discrete States 1069
Indicator States 1070
Explicit Event 1070
Implicit Event 1070
Reinitialization on Domains, Boundaries, Edges, or Points 1071

The Wall Distance Interface **1072**
Domain and Boundary Nodes for the Wall Distance Interface. 1072
Distance Equation 1073
Initial Values. 1073
Wall. 1073
Wall Distance Continuity. 1073
Periodic Condition 1074

Theory for Wall Distance **1075**
The Eikonal Equation 1075
Modified Eikonal Equation 1075
Reference for the Wall Distance Interface 1076

Curvilinear Coordinates **1077**
Introduction. 1077
The Curvilinear Coordinates Interface. 1077
Diffusion Method 1078
Adaptive Method 1079
Elasticity Method 1079
Flow Method 1080
User Defined 1080
Inlet 1081
Jump 1081
Outlet 1082
Wall. 1082
Interior Wall 1082

Coordinate System Settings 1082

Using Extra Dimensions **1083**
Attached Dimensions 1084
Points to Attach 1086
Integration Over Extra Dimension 1087
Example: Solving Poisson's Equation in a Cylinder by Means of Extra Dimensions 1087

Chapter 17: Sensitivity Analysis

Theory for the Sensitivity Interface **1090**
About Sensitivity Analysis. 1090
Sensitivity Problem Formulation 1090
Theory for Stationary Sensitivity Analysis. 1091
Specification of the Objective Function 1092
Choosing a Sensitivity Method 1093
Postprocessing Sensitivities 1094
Issues to Consider Regarding the Control Variables 1094
Issues to Consider Regarding the Objective Function 1095
Issues to Consider Regarding Constraints 1096

The Sensitivity Interface **1097**
Integral Objective 1098
Probe Objective 1098
Control Variable Field 1098
Global Objective 1099
Global Control Variables 1099

Chapter 18: Deformed Geometry and Moving Mesh

Deformed Mesh Fundamentals **1102**
About Deformed Meshes. 1102
Deformed Geometry vs. Moving Mesh. 1102
Arbitrary Lagrangian-Eulerian Formulation (ALE) 1103
About Frames 1104
Mathematical Description of the Mesh Movement 1105
Derivatives of Dependent Variables 1106
Transformation Matrices and Volume Factors 1108
Smoothing Methods. 1110
Limitations of the ALE Method 1112
Tips for Modeling Using Deformed Meshes 1112
Remeshing a Deformed Mesh 1113

Deformed Mesh Definition Features **1115**
Moving Mesh Features 1115
Deformed Geometry Features. 1116
Prescribed Deformation 1116
Rotating Domain 1117
Deforming Domain 1118
Fixed Boundary 1118
Prescribed Mesh Displacement 1118

Prescribed Normal Mesh Displacement 1118
Prescribed Normal Mesh Velocity 1119
Rotating Boundary 1119
Mesh Slip 1120
Symmetry/Roller 1120

The Moving Mesh Interface **1121**
Domain and Boundary Nodes in the Moving Mesh Interface 1122
Fixed Mesh 1122
Prescribed Mesh Displacement 1122
Free Deformation 1123
Prescribed Deformation 1123
Prescribed Mesh Velocity. 1123
Prescribed Normal Mesh Velocity 1123
Zero Normal Mesh Velocity 1124
Zero Normal Mesh Displacement 1124

The Deformed Geometry Interface **1125**
Domain and Boundary Nodes for Deformed Geometry 1126
Fixed Mesh 1126
Prescribed Mesh Displacement 1126
Free Deformation 1126
Prescribed Deformation 1126
Prescribed Mesh Velocity. 1127
Prescribed Normal Mesh Velocity 1127
Zero Normal Mesh Velocity 1127
Zero Normal Mesh Displacement 1128

Chapter 19: Elements and Shape Functions

Shape Functions and Element Types **1130**
Finite Elements 1130
Shape Function Types (Element Types) 1132

Chapter 20: Studies and Solvers

Introduction to Solvers and Studies **1150**
The Add Study Window 1151
Study 1153
Solver Configurations 1153
The Relationship Between Study Steps and Solver Configurations . . . 1155
Combine Solutions 1157
Study Reference 1158
Model Reduction 1159

Study and Study Step Types **1162**
Common Study Step Settings 1170
Using a Solution From Previous Study Steps. 1177
Physics and Variables Selection 1178
Error Estimation — Theory and Variables 1182
Stationary 1184

Time Dependent 1186
Time Discrete 1188
Time Dependent, Modal 1188
EEDF Initialization 1188
Time Periodic 1189
Time Periodic to Time Dependent 1189
Frequency to Time FFT 1189
Eigenfrequency. 1189
Eigenvalue 1191
Frequency Domain 1193
Frequency Domain, Modal 1194
Frequency Domain, AWE Reduced-Order Model 1195
Frequency Domain, Modal Reduced-Order Model 1196
Time Dependent, Modal Reduced-Order Model 1196
Adaptive Frequency Sweep 1196
Time to Frequency FFT 1197
Batch 1197
Batch Sweep 1198
Bidirectionally Coupled Particle Tracing 1200
Bidirectionally Coupled Ray Tracing. 1201
Cluster Computing 1201
Cluster Sweep 1206
Function Sweep 1208
Material Sweep 1209
Modal Reduced-Order Model 1209
Multigrid Level 1209
Parametric Sweep 1210
Sensitivity. 1213
Bolt Pretension 1215
Boundary Mode Analysis 1215
TEM Boundary Mode Analysis 1216
Coil Geometry Analysis 1216
Electrochemistry Studies and Study Steps 1217
Fatigue 1219
Frequency Domain Perturbation 1220
Frequency-Stationary 1221
Frequency-Transient 1221
Frozen Rotor 1222
Frozen Rotor with Initialization 1222
Stationary Free Surface 1223
Frozen Rotor with Stationary Free Surface 1223
Frozen Rotor with Initialization and Stationary Free Surface 1223
Linear Buckling. 1223
Mapping 1225
Mean Energies 1225
Mode Analysis 1225
Optimization 1227
Parameter Estimation 1227
Prestressed Analyses Studies 1227
Ray Tracing 1229
Reduced Electric Fields 1230
Schrödinger-Poisson 1230
Semiconductor Equilibrium 1231
Semiconductor Initialization 1231

Frequency-Stationary, One-Way Coupled, Electromagnetic Heating . . 1232
Frequency-Transient, One-Way Coupled, Electromagnetic Heating . . . 1232
Small-Signal Analysis, Frequency Domain 1232
Stationary and Time Dependent One-Way Coupled Studies for Fluid-Structure Interaction 1232
Stationary, Time Dependent, and Frozen Rotor One-Way Coupled Studies for Nonisothermal Flow 1234
Stationary and Time Dependent One-Way Coupled Studies for Moisture Flow 1235
Stationary Source Sweep 1235
Stationary Plug Flow. 1235
Stationary with Initialization and Time Dependent with Initialization . . 1236
Stationary, Time Dependent, and Frozen Rotor One-Way Coupled with Initialization Studies for Nonisothermal Flow 1236
Thermal Perturbation, Eigenfrequency 1237
Thermal Perturbation, Frequency Domain 1237
Time Dependent with FFT 1237
Time Dependent with Phase Initialization. 1238
Wavelength Domain 1238

Computing a Solution **1240**
Getting Results While Solving 1243
Computing the Initial Values 1243
The Progress Window. 1243
Convergence Plots 1245
The Log Window. 1246
The External Process Window. 1250

Solution Operation Nodes and Solvers **1251**
Selecting a Stationary, Time-Dependent, or Eigenvalue Solver 1252
Remarks on Solver-Related Model Characteristics 1252
Scaling of Variables and Equations 1253
About the Stationary Solver. 1254
About the Parametric Solver 1257
About the Time-Dependent Solver 1257
About the Time Discrete Solver 1260
The Eigenvalue Solver Algorithm 1261
The Modal Solver Algorithm 1262
The Time Explicit Solver Algorithms 1264
The AWE Solver Algorithm 1265
AWE Solver. 1266
Dependent Variables 1269
Eigenvalue Solver 1271
FFT Solver 1273
Modal Solver 1279
Optimization Solver 1282
Plug Flow Solver 1282
Stationary Solver 1282
Time-Dependent Solver 1284
Time Discrete Solver 1292
Time Explicit Solver 1293
References for the Solution Operation Nodes and Solvers 1294

Solution Attribute Nodes **1296**
About the Advanced Attribute Settings 1299
Choosing the Right Linear System Solver 1300
About Incomplete LU 1302

The Adaptive Mesh Refinement Solver. 1302
The Domain Decomposition Solvers 1304
The Fully Coupled Attribute and the Double Dogleg Method 1306
The Iterative Solvers 1307
The Multigrid Solvers 1311
The Parametric Solver Algorithm. 1313
The SCGS Solver 1314
The Segregated Solver 1314
The Sensitivity Analysis Algorithm 1315
About the SOR, SOR Gauge, SOR Line, and SOR Vector Iterative Solver Algorithms 1317
The Sparse Approximate Inverse (SAI) Preconditioner 1319
The Vanka Algorithm 1319
Complex Shifted Laplacian for Large Helmholtz Problems 1320
The WENO Limiter. 1321
Adaptive Mesh Refinement 1321
Advanced. 1324
Auxiliary-Space Maxwell (AMS) 1327
Automatic Remeshing 1328
Coarse Solver 1329
Control Field 1329
Control State 1330
Direct 1330
Direct Preconditioner 1334
Domain Decomposition (Schur) 1335
Domain Decomposition (Schwarz) 1336
Domain Solver. 1341
Eigenvalue Parametric 1341
Error Estimation 1342
Field. 1342
Fully Coupled 1343
Hierarchical LU 1347
Incomplete LU. 1348
Iterative 1349
Jacobi 1352
Krylov Preconditioner 1352
Localized Schur 1353
Lower Limit. 1354
Lumped Step 1354
Multigrid 1354
Parametric 1357
Postsmoother 1360
Presmoother 1360
Previous Solution. 1360
Sparse Approximate Inverse (SAI) 1361
Sparse Localized Schur. 1362
SCGS 1362
Schur Solver 1364
Segregated 1364
Segregated Step 1366
Sensitivity. 1367
SOR. 1367
SOR Gauge 1368
SOR Line 1369
SOR Vector. 1369

State 1370
Stationary Acceleration 1371
Stop Condition 1372
Time Parametric 1373
Upper Limit 1374
Vanka 1375
References for the Linear System Solvers and the Preconditioners . . . 1376

Solution Utility Nodes **1379**
Adaptive Mesh Refinement (Utility Node) 1379
Assemble 1379
Compile Equations 1381
Copy Solution 1381
For and End For 1382
The Statistics Page 1383
Input Matrix 1383
Solution Store 1383
State Space 1384

Job Configurations **1386**
Parametric Sweep (Job Configurations) 1387
Batch (Job Configurations) 1389
Cluster Computing (Job Configurations) 1392
Function Sweep (Job Configurations) 1395
Material Sweep (Job Configurations) 1396
Optimization (Job Configurations) 1396
Sequence 1397
Using a Job Configuration to Store Parametric Results on File 1397
Batch Data 1399
Derived Value 1399
Evaluate Derived Value 1400
Export to File 1400
External Class 1401
External Process 1401
Method Call 1402
Geometry 1402
Job 1402
Mesh 1402
Plot Group 1402
Save Model to File 1403
Solution 1403

Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis **1404**
Frequency Domain Perturbation Study Step. 1404
Harmonic Perturbation — Exclusive and Contributing Nodes 1404

Chapter 21: Results Analysis and Plots

Results Overview **1408**
About the Results Branch 1408
Common Results Node Settings 1410
Selecting a Dataset for Plots and Postprocessing 1412

Inputs for Parametric Solver and Parametric Sweep Studies 1413
Entering Axis Data for a Dataset 1414
Expressions and Predefined Quantities. 1414
Defining Plane Data for a Dataset 1416
Plot Titles for Plot Groups and Plot Types 1417
Using Special Formats and Symbols in Titles. 1419
Arrow Positioning 1421
Principal Components and Positioning 1422
Defining the Number of Levels 1422
Selecting Color Tables 1423
Defining the Color and Data Ranges 1424
Defining the Coloring and Style 1424
Defining Element Filters 1430
Defining Shrinking of Elements. 1430
Entering Quality Settings for Plot Settings Windows 1430
Inheriting Style Options 1433
Integration Settings for a Derived Value 1433
Data Series Operation Settings for a Derived Value. 1434
Small-Signal Analysis, Prestressed Analysis, and Harmonic Perturbation Plot Settings 1435
Extra Time Steps for Trajectory Plots and Intersection Point Datasets . 1437
Node Properties for Reports 1438
Through-Thickness Location for Layered Materials 1438

Datasets **1439**

About Datasets 1439
Dataset Types 1440
Array 1D, Array 2D, and Array 3D 1442
Average and Integral 1444
Contour (Dataset) 1444
Cut Line 2D and Cut Line 3D 1445
Cut Plane. 1445
Cut Point 1D, Cut Point 2D, and Cut Point 3D 1446
Edge 2D and Edge 3D 1447
Extrusion 1D and Extrusion 2D 1448
Filter 1448
Grid 1D, Grid 2D, and Grid 3D 1449
Intersection Point 2D and Intersection Point 3D 1450
Isosurface (Dataset). 1452
Join 1452
Layered Material 1453
Maximum and Minimum 1454
Mesh (Dataset) 1455
Mirror 2D and Mirror 3D 1455
Parameterized Curve 2D and Parameterized Curve 3D 1456
Parameterized Surface 1456
Parametric Extrusion 1D and Parametric Extrusion 2D 1457
Particle (Dataset). 1458
Particle Bin 1458
Ray (Dataset) 1459
Ray Bin 1460
Receiver 2D and Receiver 3D 1460
Response Spectrum 2D and Response Spectrum 3D 1461
Revolution 1D and Revolution 2D 1465
Sector 2D and Sector 3D. 1465

Shell. 1467
Solution 1468
Surface (Dataset) 1468
Time Average and Time Integral 1469

Derived Values, Evaluation Groups, and Tables 1470
About Derived Values 1470
About Evaluation Groups. 1470
The Table Window and Tables Node 1472
Derived Value Types 1474
Volume Average, Surface Average, and Line Average 1475
Volume Integration, Surface Integration, and Line Integration 1476
Volume Maximum, Volume Minimum, Surface Maximum, Surface Minimum, Line Maximum, and Line Minimum 1476
Point Evaluation 1477
Global Evaluation 1478
Global Matrix Evaluation 1478
Particle Evaluation 1479
Point Matrix Evaluation 1480
Ray Evaluation 1480
Aberration Evaluation 1481
Global Evaluation Sweep 1482
System Matrix 1482
Table 1483

Plot Groups and Plots 1486
About Plot Groups 1486
Plot Types 1487
The Plot Windows 1492
Creating Cross-Section Plots and Combining Plots 1493
Plotting and Cross-Section Interactive Toolbar 1494
1D, 2D, and 3D Cross-Section Point Plots 1495
2D Cross-Section Line Plots 1497
3D Cross-Section Line Plots 1498
3D Cross-Section Surface Plot. 1500
1D Plot Group and Polar Plot Group 1501
Smith Plot Group. 1505
2D Plot Group and 3D Plot Group 1507
Admittance Graph 1510
Annotation 1511
Annotation Data 1513
Arrow Data 1513
Arrow Line 1513
Arrow Point 1514
Arrow Surface 1514
Arrow Volume. 1514
Contour (Plot). 1514
Coordinate System Volume, Coordinate System Surface, and Coordinate System Line 1515
Directivity 1516
Global 1518
Histogram 1520
Impedance Graph. 1521
Impulse Response 1522
Interference Pattern. 1523

Isosurface (Plot) 1524
Layered Material Slice 1524
Line 1526
Line Data 1526
Line Graph 1526
Matrix Histogram. 1527
Max/Min Volume, Max/Min Surface, Max/Min Line, Max/Min Point . . . 1527
Mesh (Plot) 1529
Multislice 1530
Nyquist 1531
Octave Band 1531
Optical Aberration 1533
Particle (Plot) 1536
Particle Tracing 1536
Particle Tracing with Mass 1539
Particle Trajectories. 1541
Filter for Particle Trajectories 1541
Phase Portrait 1542
Poincaré Map 1542
Point Data 1543
Point Graph 1543
Point Trajectories 1544
Filter for Point Trajectories 1545
Principal Stress Volume 1545
Principal Stress Surface 1546
Principal Stress Line 1546
Radiation Pattern 1546
Ray (Plot). 1549
Ray Trajectories 1550
Filter for Ray and Ray Trajectories 1550
Reflection Graph 1551
Scatter Surface and Scatter Volume 1551
Slice 1552
Spot Diagram 1553
Streamline 1556
Streamline Surface 1561
Surface (Plot) 1564
Surface Data 1564
Surface Slit 1565
Table Graph. 1565
Table Surface 1567
Through Thickness 1568
Tube Data 1570
Volume 1570
Waterfall 1570
Whirl 1571
Color Expression. 1571
Deformation 1571
Export Expressions 1572
Filter 1572
Height Expression 1573
Selection (Plot Attribute). 1574

Exporting Data and Images **1575**
Export Types . . . 1575
About the Sectionwise Data Format for Data Export . . . 1575
Animation . . . 1576
Data. . . . 1580
Mesh (Export) . . . 1582
Table . . . 1583
Image . . . 1583
Plot . . . 1586

Reports **1588**
About the Report Generator . . . 1588
Generating a Model Report . . . 1588
Creating, Exporting, and Using Custom Report Templates . . . 1589
The Template Node . . . 1589
The Report Node . . . 1590
The Title Page . . . 1593
The Table of Contents. . . . 1594
Sections in the Report . . . 1594
Custom Report Components . . . 1594
Declaration Components. . . . 1596
Arrays and Scalars . . . 1596
Mathematical Symbols and Special Characters . . . 1596
Model Contents — Report Components. . . . 1604
Root Report Node . . . 1604
Component Report Node . . . 1604
Definitions Report Nodes . . . 1604
Geometry Report Node . . . 1608
Material Report Node . . . 1608
Physics Interface Report Node. . . . 1609
Multiphysics Coupling Report Node. . . . 1609
Mesh Report Node . . . 1610
Study Report Node . . . 1610
Solver Report Node . . . 1610
Results Report Nodes . . . 1611
Declaration Contents . . . 1612

Printing and Capturing Screenshots **1614**
Printing from the COMSOL Desktop . . . 1614
Capturing and Copying Screenshots. . . . 1614
Creating PowerPoint Presentations with Images Linked to Models . . . 1616
Setting and Clearing the Thumbnail Image . . . 1619

Chapter 22: Running COMSOL Multiphysics

Running COMSOL Multiphysics **1622**
Windows and the Cross-Platform Desktop . . . 1622
COMSOL Multiphysics Client-Server Architecture . . . 1622
Parallel Computing with COMSOL Multiphysics . . . 1622
LiveLink for MATLAB . . . 1623
LiveLink for Excel. . . . 1623
COMSOL Batch . . . 1623

COMSOL API 1624
Security Settings 1624
COMSOL and the Java Heap Space 1625

COMSOL Multiphysics Client-Server Architecture **1627**
Standalone COMSOL 1627
Running COMSOL Multiphysics in Client-Server Mode 1627
Running COMSOL with MATLAB or Excel 1627

Running COMSOL Multiphysics in Client-Server Mode **1629**
Advantages of Using COMSOL Multiphysics in Client-Server Mode . . . 1629
Running COMSOL Multiphysics in Client-Server Mode 1629
Connecting, Disconnecting, and Reconnecting to and from the Desktop . 1630
Shared Libraries 1632

Running COMSOL in Parallel **1633**
Overview of Simulations on Shared- and Distributed-Memory Computers 1633
Shared-Memory Parallel COMSOL 1634
COMSOL and BLAS. 1635
Distributed-Memory Parallel COMSOL 1635
Benefits of Running COMSOL in a Distributed Mode 1636
Running COMSOL in Parallel on Clusters 1636
Grid Computing and Remote Computing in COMSOL Multiphysics. . . . 1639

The COMSOL Commands **1641**
COMSOL Commands on Windows. 1641
COMSOL Commands on Linux 1655
COMSOL Commands on macOS. 1669

Chapter 23: Glossary

Glossary of Terms **1680**

1

Introduction

Welcome to the COMSOL Multiphysics® simulation software! This book details features and techniques that help you throughout all COMSOL Multiphysics modeling in version 5.5 using the COMSOL Desktop® environment. For example, detailed information about:

- How to build geometries in COMSOL Multiphysics.
- How to create a mesh for the analysis.
- How to create parameters and variables used within a model.
- How to add the physics interfaces and material properties.
- How to solve and display the results.

The full set of documentation shows you, step by step, how to tap into the functions and capabilities in the COMSOL Multiphysics software. This introductory chapter provides an overview of COMSOL Multiphysics and its product family, documentation set, and other resources.

Version 5.5 contains further enhanced *COMSOL Compiler*™, *Application Builder*, and *COMSOL Server*™ products for creating and deploying custom applications based on COMSOL Multiphysics.

In this chapter:

- About COMSOL Multiphysics
- COMSOL Documentation and Help
- Overview of the Reference Manual

About COMSOL Multiphysics

COMSOL Multiphysics is a powerful interactive simulation environment used to model and solve all kinds of scientific and engineering problems. The software provides a powerful integrated desktop environment with a *Model Builder* that gives you a full overview of the model and access to all functionality. With COMSOL Multiphysics you can easily extend conventional models for one type of physics into multiphysics models that solve coupled physics phenomena — and that do so simultaneously. Accessing this power does not require an in-depth knowledge of mathematics or numerical analysis.

Using the built-in *physics interfaces* and the advanced support for material properties, you can build models by defining the relevant physical quantities — such as material properties, loads, constraints, sources, and fluxes — rather than by defining the underlying equations. You can always apply these variables, expressions, or numbers directly to solid and fluid domains, boundaries, edges, and points independently of the computational mesh. The COMSOL Multiphysics software then internally compiles a set of equations representing the entire model.

You access the power of COMSOL Multiphysics as a standalone product through a flexible graphical user interface (GUI), in applications created using the Application Builder and deployed using COMSOL Compiler™ or COMSOL Server™, or by script programming in Java® or the MATLAB® language (this requires a LiveLink™ *for* MATLAB® license).

Using these physics interfaces, you can perform various types of studies including:

- Stationary and time-dependent (transient) studies
- Linear and nonlinear studies
- Eigenfrequency, modal, and frequency response studies

When solving the models, the COMSOL Multiphysics software assembles and solves the problem using a set of advanced numerical analysis tools. The software runs the analysis together with adaptive mesh refinement (if selected) and error control using a variety of numerical solvers. The studies can make use of multiprocessor systems and cluster computing, and you can run batch jobs and parametric sweeps.

The COMSOL Multiphysics software creates *sequences* to record all steps that create the geometry, mesh, physics, studies and solver settings, and visualization and results presentation. This makes it easy to parameterize any part of the model; simply change a node in the model tree and rerun the sequences. The program remembers and reapplies all other information and data in the model.

REAL-WORLD APPLICATIONS

Partial differential equations (PDEs) form the basis for the laws of science and provide the foundation for modeling a wide range of scientific and engineering phenomena. You can use the COMSOL Multiphysics software in many application areas, including the following, combining physics freely and incorporating user-defined PDEs, ODEs, and DAEs if needed:

- Acoustics
- Bioscience
- Chemical reactions
- Corrosion and corrosion protection
- Diffusion
- Electrochemistry
- Electrodeposition
- Electromagnetics
- Fatigue analysis
- Fluid dynamics
- Fuel cells and batteries
- Geophysics and geomechanics
- Heat transfer
- Layered shells and composite materials
- Metal processing
- Microelectromechanical systems (MEMS)
- Microfluidics
- Microwave engineering
- Multibody dynamics
- Optics
- Optimization and sensitivity analysis
- Particle tracing
- Photonics
- Piezoelectric devices
- Pipe flow
- Plasma physics
- Porous media flow
- Quantum mechanics
- Radio-frequency components
- Ray tracing and ray optics
- Rotordynamics
- Semiconductor devices
- Structural mechanics
- Subsurface flow
- Transport phenomena
- Wave propagation

Many real-world applications involve simultaneous couplings of physics, represented in a system of PDEs — *multiphysics*. For instance, the electric resistance of a conductor often varies with temperature, and a model of a conductor carrying current should include resistive-heating effects. The Multiphysics Modeling Workflow section discusses multiphysics modeling techniques. Predefined multiphysics interfaces provide easy-to-use entry points for common multiphysics applications.

In its base configuration, COMSOL Multiphysics offers modeling and analysis power for many application areas. For several of the key application areas there are also optional add-on modules. These application-specific modules use terminology, solution methods, and plot types specific to the particular discipline, which simplifies creating and analyzing models. The modules also include comprehensive Application Libraries with example models that show the use of the product within its application areas.

The COMSOL Multiphysics Modules and Interfacing Options

The optional modules, including interfacing options such as the CAD Import Module and bidirectional interfaces such as the LiveLink™ products, are optimized for specific application areas and offer discipline-standard terminology and physics interfaces. For some modules, additional material libraries, specialized solvers, element types, and visualization tools are also available.

> For up-to-date module availability, product descriptions, and a specification chart, go to www.comsol.com/products.

COMSOL Documentation and Help

About the Documentation Set

The full documentation set that ships with COMSOL Multiphysics consists of the following titles:

- *Introduction to COMSOL Multiphysics* — information about version 5.5 and how to build models using the desktop environment, including quick references to keyboard shortcuts and common commands and functions.
- *COMSOL License Agreement*.
- *COMSOL Multiphysics Installation Guide* — besides covering various installation options, it describes system requirements and how to configure and run the COMSOL Multiphysics software on different platforms, including client-server architectures as well as shared-memory and distributed (cluster) parallel versions.
- *COMSOL Multiphysics Reference Manual* — this book, which covers the functionality of COMSOL Multiphysics across its entire range from geometry modeling to results evaluation and visualization, including the physics interfaces for physics and equation-based modeling. It serves as a tutorial and a reference guide to use COMSOL Multiphysics. This book reviews geometry, mesh, solver, and results functionality and provides detailed information about the settings and options. Additionally, it describes some advanced functionality and settings in COMSOL Multiphysics and provides background material and references.
- *COMSOL Multiphysics Programming Reference Manual* — this book provides details about features and techniques that help you control COMSOL Multiphysics using its application programming interface (API). The COMSOL API can be used from the Application Builder, in a standalone Java® application, and from MATLAB® using the LiveLink™ *for* MATLAB® interface. For the Application Builder, the *Application Programming Guide* provides information about using the COMSOL API and the API of the Application Builder components to create methods for custom applications.
- The *Introduction to the Application Builder* and the *Application Builder Reference Manual* provide documentation related to the Application Builder and how to create and deploy simulation applications and how to use COMSOL Compiler™ to create standalone runnable applications. See also the *COMSOL Server Manual* for configure a server and clients for COMSOL Multiphysics applications.
- *COMSOL Server Manual* — information about setting up, configuring, and running a COMSOL Server for running and deploying applications within an organization.
- The *Physics Builder Manual* provides documentation related to the Physics Builder for creating custom physics interfaces.
- The *Essentials of Postprocessing and Visualization* and *Specialized Techniques for Postprocessing and Visualization* provide tips and information that help you get the most out of the postprocessing and visualization tools in COMSOL Multiphysics.
- *COMSOL Multiphysics Release Notes* — information about new functionality and changes in the 5.5 release and about compatibility with earlier versions of COMSOL Multiphysics.

In addition, each of the optional modules includes a manual as described in The COMSOL Multiphysics Modules and Interfacing Options. The documentation for the optional CAD Import Module and LiveLinks to CAD packages is available in separate manuals, and the documentation for the optional Material Library in the *Material Library User's Guide*.

The *COMSOL LiveLink™ for MATLAB® User's Guide* shows how to access the capabilities of COMSOL from the MATLAB programming environment.

DIFFERENT INSTRUCTIONS FOR DIFFERENT OPERATING SYSTEMS

The Windows® platform uses a ribbon layout, a style familiar to Microsoft® Office users and integrated into many other Windows software designs. The ribbon-style layout is intuitive and makes it easy to locate similar and frequently used features. For the Linux® and macOS platforms, there are extended toolbars that provide almost identical single-click access to most functionality in the software.

The use of the ribbon for Windows users means that there are slightly different instructions about how to access some features compared to macOS or Linux users. When specific instructions are included about where to find a particular feature, the instructions distinguish between the operating systems using different icons.

- Where there are no differences, the icons are not used.
- Where there are minor differences in appearance or accessibility, but the functionality is the same, no icons are used.
- In general, instructions for all platforms imply that the feature is available from a named toolbar. For example, the **Home** toolbar, **Physics** toolbar, **Mesh** toolbar, or **Geometry** toolbar. See Toolbars and Keyboard Shortcuts for information about each toolbar.

Win

A *ribbon tab*, *ribbon group*, or *modal ribbon tab*, are available in the Windows version. See Figure 2-1 for an example of the Windows **Home** toolbar.

Mac

Linux

The *Model Toolbar* and *Contextual Toolbar* are available in the cross-platform version, primarily for macOS and Linux users. See Figure 2-12 for an example of these toolbars.

ABOUT THE SCREENSHOTS USED IN THIS MANUAL

The screenshots used throughout this reference manual are captured using the Windows platform except when there are clear differences other than fonts or cosmetic appearance.

ADDITIONAL INTERNET RESOURCES

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries are all accessed through the COMSOL Desktop.

!

If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules, application examples, and documentation sets.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

COMSOL ACCESS AND TECHNICAL SUPPORT

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and a case number are sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/support/updates
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/community
Events	www.comsol.com/events
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

The Help Window and Topic-Based Help

The **Help** window is useful as it is connected to many of the features in the COMSOL Desktop. This concept is called *topic-based help* or *context help*. You can also search and access all the HTML documentation content from this window.

The **Help** system automatically starts a web server using port 8090 on the computer where COMSOL Multiphysics is installed. Depending on the security settings, you might get a question to allow that port to be used the first time the help system is started.

The operating system might also issue a firewall security warning. To use Help, allow COMSOL Multiphysics access through the firewall.

To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1. The **Help** window opens and displays the topic information about the selected feature.

OPENING THE HELP WINDOW AND THE TOPIC-BASED HELP

There are several ways to open the **Help** window:

- Press F1.
- In the main toolbar, click **Help** ( ).
- In the upper-right corner of the COMSOL Desktop, click the ( ) button.
- From the main menu, select **File>Help** (Windows) **Help>Help** (Linux and macOS).
- Right-click any node in the **Model Builder** and select **Help**.

ABOUT USING THE F1 KEY TO ACCESS CONTEXT HELP

To display topic-based (context) information in the **Help** window, on the COMSOL Desktop:

- Click to highlight a node in the Model Builder tree. For example, the **Component** or **Geometry** node.

- Click a window tab, for example, **Model Builder**, **Add Study**, or **Messages**.
- For Windows users, hover over ribbon buttons or menu items to display a *tooltip*. While the tooltip is showing, you can press F1 to display more detail.

At the top of the **Help** window you find the tools and search functionality listed in the table below. Also, above the help text, a clickable breadcrumb trail shows the location of the displayed contents in the COMSOL documentation set. Click any part of the trail to move to that level in the COMSOL documentation.

In some cases you need to refocus the context help on its target before pressing F1. Try clicking to highlight a node, a window, or the button or click to focus on the Help window, hover over a toolbar button (Windows only) and press F1.

TABLE 1-1: THE HELP AND DOCUMENTATION TOOLBARS

BUTTON	NAME	DESCRIPTION
		Click the **Home** button or select the COMSOL Documentation top node in the table of contents tree to return to the COMSOL Documentation window home page. Not available when showing context help in the Help window.
	Show Table of Contents/ Show Topic	When a help topic page is shown in the Help window, click **Show Table of Contents** to open a tree-based menu of the COMSOL documentation. When you select a node in the table of contents tree the corresponding topic is shown on the topic page. Alternatively, click again (**Show Topic**) to return to the topic page.
	Search	On the Help window, click to open the search engine to look for contents in the COMSOL documentation. Search results are shown sorted by product. On the Documentation window, enter search terms in the field and choose the **Search scope** — **All documents**, **Selected only**, or **Application libraries**. See Searching Help and Documentation Content for more information about search terms you can use.
/	Back	Navigates backward to the previous page in the Help or Documentation window's browser history.
/	Forward	Navigates forward in the browser history, but only to the end of the current list.
	Sticky Help	On the Help window, click the **Sticky Help** button to lock the current help window (the icon is highlighted), which can be useful to keep some help topic or model instruction active, or to release the window and view topic-based (context) help when a node or window is clicked.
	Show in External Browser	On the Help window, click the **Show in External Browser** button to show the current help topic in an external web browser.
Previous Section		Navigates backward to the previous section in the documentation.
Next Section		Navigates forward to the next section in the documentation.

CHANGING THE DEFAULT DOCUMENTATION AND HELP SETTINGS

To edit the following settings, open The Preferences Dialog Box and click **Help**.

Locate the **Target** area and choose **Documentation window** (the default) from the **Show documentation in** list to show the help contents in the **Documentation** window that is included in the COMSOL Desktop environment (default), or select **External browser** to display the help contents in a separate web browser. For further details on how to access and use the documentation, see the section The Documentation Window below. Similarly, using the **Show help in** list, you can choose between **Help window** (default) and **External browser** to show topic-based help in the **Help** window inside the COMSOL Desktop or in an external web browser, respectively. When showing topic-based help in an external browser, you need to press F1, choose **Help>Help**, or select the **Help** context menu item (when applicable) to trigger an update of the browser's contents. The **PDF-file target** setting controls what happens when you click a PDF link on the COMSOL Documentation entry page. Choose **In place** to display PDF documents using the native browser's PDF display settings, or choose **New window** to launch them in the default system application for PDF-files. On Windows, the native browser is always Microsoft Edge or Internet Explorer.

In the **Source** area, set the **Location** to **Local** to display help using locally installed help files or to **Online** to access help from the COMSOL website. For the **Local** option, edit the **Documentation root directory** file path as required. The default file paths are based on the platform:

- On Windows `C:\Program Files\COMSOL\COMSOL55\Multiphysics\doc`, or generically `COMSOL55\doc`.
- For macOS and Linux, under the main COMSOL installation directory: `COMSOL55/Multiphysics/doc`.

Proxy Server Settings

If you connect to the internet through a web proxy, you can use the controls in the **Proxy server settings** area to specify the proxy server settings to use when communicating with the COMSOL website for displaying online help in integrated mode as well as for performing updates of the COMSOL Application Library and the COMSOL Part Library (see The Application Library Update Window and The Part Library Update Window for further details about these services).

The **Configuration** list has the following options:

- **No proxy server**: Connect to the update server directly, bypassing any proxies. This is the default setting.
- **Use system settings**: Use the system-wide proxy server settings defined on your computer.
- **Manual**: Choose this alternative if you want to specify a proxy server by entering the name (or IP address) and port number in the **Server** and **Port number** fields. The default port number is 443, which is the default for HTTP secure (HTTPS). If the proxy server requires authentication, you are asked to provide username and password the first time in each COMSOL session you access documentation or update the COMSOL Application Library or Part Library.

Selecting a Web Browser

In the **General** section of the **Preferences** dialog, under **Web browser** (Windows and Linux only), you can choose which browser the COMSOL Multiphysics software should use to show pages on the COMSOL website and documentation when using the web browser mode. The following settings are available:

- On Windows: Choose the **Program** setting **System default** to use the default system web browser. Alternatively, choose **Custom** and then give the path to an **Executable** location for a different browser installed on your computer.
- On Linux: Type the path to the web browser directly in the **Executable** field, or click the **Browse** button and then point to the executable file on the file system.

On macOS, this setting is not available, and COMSOL Multiphysics always uses the system's default web browser.

Win	To open the **Documentation** window: • Press Ctrl+F1. • From the **File** menu, select **Help>Documentation**.

Mac Linux	To open the **Documentation** window: • Press Ctrl+F1 • In the main toolbar, click the **Documentation** ( ) button. • From the **Help** menu, select **Documentation**.

In the Documentation window (or a browser window, depending on your preference settings), you can navigate to PDF or HTML versions of the documentation (availability is based on your license), as well as search all the documentation, save or open PDFs, or view the HTML content in this window. There are different ways to access the same information using either the left-hand side (Figure 1-1) or right-hand side (Figure 1-2) of the window.

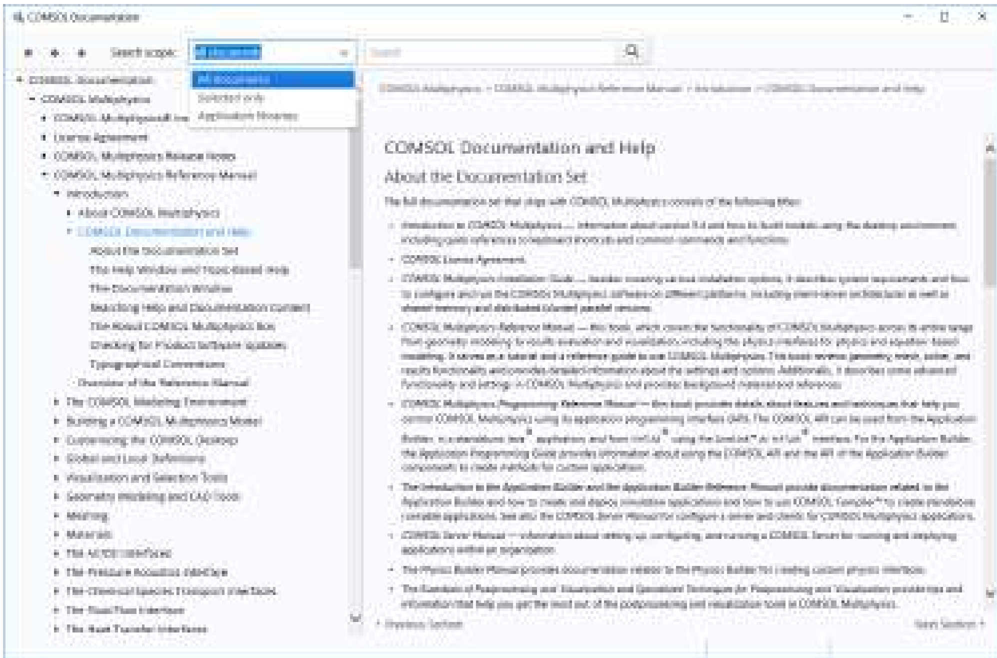


Figure 1-1: Based on your license, links to the HTML versions of the product documentation are accessed and can be browsed in the tree. When you click a topic in the tree the information displays to the right. You can also adjust the search scope.

Figure 1-2: Based on your license, links to PDF and HTML versions of the product documentation are accessed from this window. When you click HTML it jumps to the first page of the documentation for that product; when you click PDF you can Open or Save a full PDF version of that document.

- The Help Window and Topic-Based Help
- Table 1-1 for a list of the Documentation toolbar buttons.

Searching Help and Documentation Content

After you open The Help Window and Topic-Based Help, click the Search button ( ) to open the search engine and search the HTML content. Search results are shown sorted by product. You can also search in the contents of The Documentation Window.

SEARCHING THE DOCUMENTATION

On the **Documentation** window, you can adjust the **Search scope** (see Figure 1-1). Enter a search term in the **Search expression** field and then select **All documents**, **Selected only**, or **Application libraries** from the list to narrow or expand the search scope as needed. For **Selected only** (Figure 1-3), first click a branch in the tree (for example, **COMSOL Multiphysics Reference Manual**) and then the search includes all the documents below the selected node until the beginning of the next branch. In this example it searches until the end of the Troubleshooting License Errors section.

> **!** The first search can take a couple of minutes while the search index is generated.

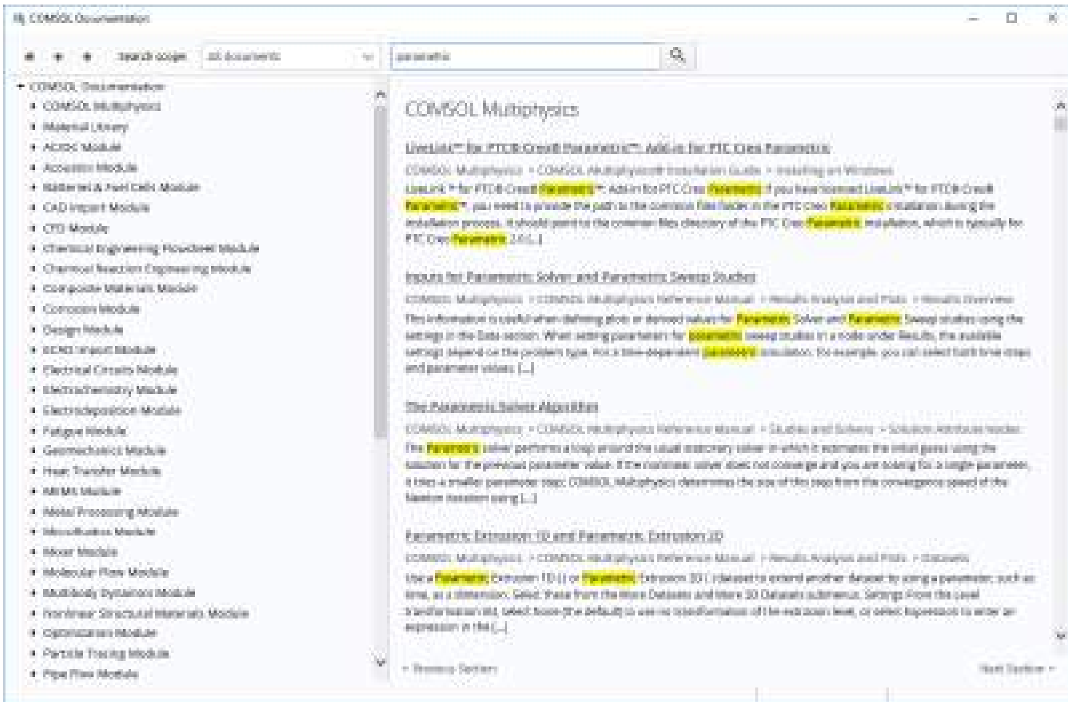


Figure 1-3: When searching in the Documentation window, choose a Search scope to search only a selected portion of the documentation, all the documentation, or only the Application Libraries.

SEARCH PARAMETERS FOR HELP AND THE DOCUMENTATION

Some examples of search parameters you can use:

TABLE 1-2: SEARCH PARAMETERS FOR THE COMSOL HELP SYSTEM

OPERATOR	EXAMPLE	SEARCH RESULT EXAMPLE
&&, AND	block && cone block AND cone	Results include all instances of the words.
OR, ||	block OR cone block || cone	Results include any of the listed words.
+, –	+block –cone	Search for one term (+block) but not the other term (–cone).
“, ~	“plot line”~10	Search for the words enclosed in the quotation marks (plot line) within (~) a certain number of words (10) from each other.
~	excentric~	Search for something “almost” spelled in a particular way. For example, excentric. The results include eccentric cone.

TABLE 1-2: SEARCH PARAMETERS FOR THE COMSOL HELP SYSTEM

OPERATOR	EXAMPLE	SEARCH RESULT EXAMPLE
?	h?t	Use in a search query to mean exactly one character. For example, search for all instances of hat, hit, or hut where ? represents a, i, or u, or any other letter between h and t.
*	strain* strain*d	The asterisk (*) is a wildcard character. Search for any word that starts with "strain". Results include strain-based, strain-rate, or strain, for example. The asterisk represents any number of characters. If the asterisk is used in the middle of the word, it searches for one letter between "n" and "d". The result in this example is strained. You can use the asterisk before and after the word in order to search for it in the middle of a longer word or expression, For example, `*setdefaultgeometry*` finds `ModelUtil.setDefaultGeometryKernel`.
enclosed quotation marks " "	"time dependent study"	Use quotation marks around a text string to search for exactly that phrase; that is, to search for the words in the order given within the quotation marks.

The About COMSOL Multiphysics Box

Figure 1-4: The About COMSOL Multiphysics dialog box with the setting to show the Acknowledgments list.

To open the **About COMSOL Multiphysics** ( ) window:

- For Windows users, select it from the **File** menu.
- For macOS and Linux (cross platform) users, choose it from the **Help** menu.

In addition to copyright and patent information, the **About COMSOL Multiphysics** dialog box has the following information:

- The **Version number**

- The user or company **This product is licensed to**
- The **License number**

Select an option from the list below and then click **Show Information** to open a separate window of the same name containing this information:

- Select **Acknowledgments** to show information about third-party software components, including license notices required by the software component authors. Then click **Show Information**.
- Select **License agreement** to show the COMSOL Multiphysics software license agreement. Then click **Show Information**.
- Select **Licensed products** to show the licensed COMSOL products, including the number of used licenses and the total number of licenses for each product. Then click **Show Information**.
- Select **Patents** to show the patents that the COMSOL software products are protected by. Then click **Show Information**.
- Select **System information** to show a list of system properties, which can be useful for troubleshooting purposes, for example. Then click **Show Information**.
- Click **COMSOL Web Page** to open your web browser on the main COMSOL web page.

You can also get information about the licensed products from the **Licensed and Used Products in Session** window.

The Root Settings and Properties Windows

Checking for Product Software Updates

COMSOL provides product software updates that improve the software and correct any issues found.

To check if a product update is available, from the **File** menu select **Help>Check for Product Updates** ( ).

The program then checks if an update that is applicable, but not yet installed, is available from the COMSOL website.

If an update is available, an **Update** dialog box appears; click **Download** to download the update directly, or click **Browse Update** to open the COMSOL website where you can read about and download the update.

If no updates are available, the **Update** dialog box reports that your COMSOL installation is up to date. Open The Preferences Dialog Box and click **Updates** to select the **Check for updates at launch** check box to make the program check for updates each time you launch the COMSOL Multiphysics program.

Typographical Conventions

All documentation uses a set of consistent typographical conventions that make it easier to follow the discussion, understand what you can expect to see on the graphical user interface (GUI), and know which data must be entered into various data-entry fields.

In particular, these conventions are used throughout the documentation:

CONVENTION	EXAMPLE
text highlighted in blue	Click text highlighted in blue to go to other information in the PDF. When you are using the help desk in COMSOL, links to other modules, application examples, and documentation sets also work.
boldface font	A **boldface** font indicates that the given word(s) appear exactly that way on the COMSOL Desktop (or, for toolbar buttons, in the corresponding tooltip). For example, the **Model Builder** window is often referred to, and this is the window that contains the model tree. As another example, the instructions might say to click the **Zoom Extents** button ( ), and this means that when you hover over the button with your mouse, the same label displays on the COMSOL Desktop.
italic font	An *italic* font is the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of other documents in the COMSOL documentation set are also in *italic* font.
Forward arrow symbol **>**	The forward arrow symbol **>** means you select a series of menu items or nodes in a specific order. For example, **Component>Mesh** is equivalent to: Under the **Component** node, click the **Mesh** node.
`code` (monospace) font	A `code` (monospace) font means you make a keyboard entry in the COMSOL Desktop. You might see an instruction such as “Enter (or type) 1.25 in the **Current density** field.” The monospace font is also used to indicate programming code and variable names.
Italic *`code`* (monospace) font	An italic *`code`* (monospace) font indicates user inputs and parts of names that can vary or be defined by the user.
Arrow brackets <> following the `code` (monospace) or *`code`* (italic) fonts	The arrow brackets included in, for example, programming examples (after a monospace `code` or an italic *`code`* font) mean that the content in the string can be freely chosen or entered by the user, such as a feature Name or Label. For example, `model.geom(<label>)`, where `<label>` is the geometry’s label. When the string is predefined by COMSOL Multiphysics, no bracket is used and this indicates that this is a finite set, such as a feature type.

KEY TO THE GRAPHICS

Throughout the documentation, icons are used to help organize the information. These icons vary in importance, but it is recommended that you read these text boxes.

ICON	NAME	DESCRIPTION
	Caution	A Caution icon indicates that the user should proceed carefully and consider the next steps. It might mean that an action is required, or if the instructions are not followed, that there will be problems with the model solution.
	Important	An Important icon indicates that the information provided is key to the model building, design, or solution. The information is of higher importance than a note or tip, and the user should endeavor to follow the instructions.
	Note	A Note icon indicates that the information can be of use to the user. It is recommended that the user reads the text.
	Tip	A Tip icon is used to provide information, reminders, shortcuts, suggestions for improving model design, and other information that might be useful.
	See Also	The See Also icon indicates that other useful information is located in the named section. If you are working online, click the hyperlink to go to the information directly. When the link is outside of the current PDF document, the text indicates this, for example, “See The Laminar Flow Interface in the *COMSOL Multiphysics Reference Manual*.” Note that if you are in the online help, the link works.

ICON	NAME	DESCRIPTION
	An example from the Application Libraries	The icon is used in the documentation to indicate examples that demostrate the use of some functionality. In some cases, an example is only available if you have a license for a specific module. The Application Library path describes how to find the actual file in COMSOL Multiphysics, for example: If you have the RF Module, see *Radar Cross Section*: Application Library path **RF_Module/ Scattering_and_RCS/radar_cross_section**
Space Dimension		Another set of icons is used in the Model Builder — the component space dimension is indicated by 1D axial symmetry , 2D , 2D axial symmetry , and 3D icons. The 1D and 0D icons are not used but the space dimension is indicated. These icons are also used in the documentation to list the differences to a physics interface, node, or theory section, which are based on space dimension.
Win	Windows	This icon means that the information is specific to a Microsoft Windows operating system.
Mac	macOS	This icon means that the information is specific to a macOS operating system. This may also be referred to as cross-platform when describing how to access a feature or menu on the COMSOL Desktop.
Linux	Linux	This icon means that the information is specific to a Linux operating system. This may also be referred to as cross-platform when describing how to access a feature or menu on the COMSOL Desktop.

Overview of the Reference Manual

This *COMSOL Multiphysics Reference Manual* provides comprehensive information about all modeling steps using the COMSOL Multiphysics software. See the individual module manuals for information specific to a specialized module (see The COMSOL Multiphysics Modules and Interfacing Options for a link to the COMSOL website).

As detailed in the section COMSOL Documentation and Help this information can also be searched from the **Help** system in COMSOL Multiphysics.

TABLE OF CONTENTS, GLOSSARY, AND INDEX

To help you navigate through this guide, see the Contents, Glossary, and Index.

ENVIRONMENT

The COMSOL Modeling Environment chapter provides an overview of the COMSOL modeling environment as controlled by the COMSOL Desktop and the tools and windows it provides in the Windows version as well as the cross-platform version. Topics include The COMSOL Desktop, The Application Libraries Window, The Physics Interfaces, Creating a New Model with the Model Wizard, and a key to the icons including links in the Toolbars and Keyboard Shortcuts section.

MODELING

Building a COMSOL Multiphysics Model explains a range of methods and topics including information about the following: details about an introduction to The Model Builder, The Component Node, The Physics Nodes, Selecting Physics Interfaces, Analyzing Model Convergence and Accuracy, Specifying Model Equation Settings, Boundary Conditions, Using Units, Numerical Stabilization, and much more.

CUSTOMIZING THE COMSOL DESKTOP

In the Customizing the COMSOL Desktop chapter, the settings are described related to Customizing a Model, changing Preferences Settings, and details about the Showing More Options.

DEFINITIONS

The Global and Local Definitions chapter describes the global and local (component) definitions features. Depending on the geometric scope, you add the nodes described in this section to either the Global Definitions node or under the Definitions node for a particular component. Topics include Operators, Functions, and Constants, Predefined and Built-In Variables, Mass Properties, Functions, Nonlocal Couplings and Coupling Operators, Coordinate Systems, Identity and Contact Pairs, Probes, and Infinite Elements, Perfectly Matched Layers, and Absorbing Layers.

VISUALIZATION AND SELECTION

The Visualization and Selection Tools chapter describes the tools used to visualize and control how you view models and select parts of the model geometry in the Graphics window and the Settings windows. Important topics include Working with Geometric Entities, Creating Named Selections, and User-Defined Views.

GEOMETRY

The Geometry Modeling and CAD Tools chapter covers geometry modeling in 1D, 2D, and 3D with examples of solid modeling, boundary modeling, Boolean operators, and other CAD tools in COMSOL. In addition, it shows how to use the tools for exploring geometric properties, such as volumes and surfaces. There is also information about using external CAD data. Topics include Creating a Geometry for Analysis, Working with Geometry Sequences, Geometric Primitives, Geometry Operations, and Virtual Geometry and Mesh Control Operations.

MESH

The Meshing chapter summarizes how to create and control your mesh for 1D, 2D, and 3D geometries in the COMSOL Multiphysics software. It also explains these topics, which include: Creating a Mesh for Analysis, Meshing Techniques, Meshing Operations and Attributes, and Importing and Exporting Meshes.

MATERIAL

The Materials chapter introduces you to the material databases included with the COMSOL products. Topics include a Materials Overview, Working with Materials, Material Properties Reference, User-Defined Materials and Libraries, Using Functions in Materials, and Module-Specific Material Databases.

AC/DC

The AC/DC Interfaces chapter explains the physics interfaces available for modeling electromagnetics, which you find under the AC/DC branch ( ) when adding a physics interface. It also contains sections about general fundamentals and theory for electric fields.

ACOUSTICS

The Pressure Acoustics Interface chapter describes how to use the Pressure Acoustics, Frequency Domain interface, found under the Acoustics branch ( ) when adding a physics interface, for modeling and simulation of acoustics and vibrations.

CHEMICAL SPECIES TRANSPORT

The Chemical Species Transport Interfaces chapter explains how to use the Transport of Diluted Species interface, found under the Chemical Species Transport branch ( ) when adding a physics interface, to model and simulate mass transfer by diffusion and convection based on Fick's law of diffusion.

FLUID FLOW

The Fluid Flow Interface chapter explains how to use the Laminar Flow interface, found under the Fluid Flow>Single-Phase Flow branch ( ) when adding a physics interface, to model and simulate fluid mechanics for laminar, incompressible fluids.

HEAT TRANSFER

The Heat Transfer Interfaces chapter describes the different types of Heat Transfer interfaces (Heat Transfer in Solids and Heat Transfer in Fluids), and the Joule Heating interface, all found under the Heat Transfer branch ( ) when adding a physics interface.

SOLID MECHANICS

The Solid Mechanics chapter explains how to use the Solid Mechanics interface, found under the Structural Mechanics branch ( ) when adding a physics interface, to simulate and analyze applications involving solid mechanics. The physics interface is used for stress analysis and general solid mechanics simulation.

EQUATION-BASED MODELING

The Equation-Based Modeling chapter describes the use of the mathematics interfaces, found under the Mathematics branch (Δu) when adding a physics interface, which are used for equation-based modeling. With those interfaces you can solve various types of PDEs using different formulations. You can also solve ODEs and other global equations, add events and curvilinear coordinates, compute sensitivities, and add moving interfaces and deforming meshes.

SENSITIVITY ANALYSIS

The Sensitivity Analysis chapter describes how to perform sensitivity analysis using the Sensitivity interface, found under the Mathematics>Optimization and Sensitivity ( ) branch when adding a physics interface.

DEFORMED MESHES

The Deformed Geometry and Moving Mesh chapter explains how to use the modeling physics interfaces that control mesh deformation. These are found under the **Mathematics>Deformed Mesh** ( ) branch when adding a physics interface. It also contains fundamentals about deformed meshes and information about the Eulerian and Lagrangian formulations of the physics, the frame types that support these formulations, and the arbitrary Lagrangian-Eulerian (ALE) method.

STUDIES AND SOLVERS

The Studies and Solvers chapter lists the various types of solvers and studies in the COMSOL Multiphysics software and explains the study steps and solver configurations. It also describes the major solvers and settings as well as batch jobs, parametric sweeps, and cluster computing. See also the *Optimization Module Manual* for other supplementary information.

RESULTS AND VISUALIZATION

The Results Analysis and Plots chapter helps you analyze results in COMSOL Multiphysics and describes numerous result-evaluation and visualization tools, including advanced graphics, data display, and export functions. Topics include Results Overview, Datasets, Plot Groups and Plots, Derived Values, Evaluation Groups, and Tables, Exporting Data and Images, Reports, and Printing and Capturing Screenshots.

RUNNING COMSOL MULTIPHYSICS

Running COMSOL Multiphysics is an overview of the different ways that you can run the COMSOL Multiphysics software in addition to running the COMSOL Desktop on a dedicated computer, including client-server and distributed-memory architectures and cloud-based computing. This chapter also includes information about how to compile COMSOL applications from the command line using COMSOL Compiler.

2

The COMSOL Modeling Environment

The COMSOL Desktop® provides a complete and integrated modeling environment for creating, analyzing, and visualizing multiphysics models. This chapter provides an overview of the COMSOL Multiphysics® modeling environment as controlled by the COMSOL Desktop and the tools and windows it provides.

In this chapter:

- The COMSOL Desktop
- The Application Libraries Window
- The Physics Interfaces
- Creating a New Model
- Toolbars and Keyboard Shortcuts

The COMSOL Desktop

This section is an overview of the major components in the COMSOL Multiphysics environment. These components are integrated into the *COMSOL Desktop*, which you can personalize to your own modeling needs and preferences. Primarily consisting of the *Model Builder* nodes, *Settings* windows, and *Graphics* windows, other dockable windows can be opened, closed, and organized according to the modeling settings you need to access and the GUI configuration you want to work in. You can save these configurations, and the last opened configuration is always displayed when you open COMSOL again.

- Creating a New Model
- Building a COMSOL Multiphysics Model
- Customizing the COMSOL Desktop
- The Model Builder

Mac

Linux

The COMSOL Desktop in the cross-platform version, primarily for the Linux and macOS operating systems, looks slightly different than for the Windows operating system (shown in Figure 2-1). The primary difference is that the Main Menu and Main Toolbar are used instead of ribbons. Otherwise, the default windows (Model Builder, Graphics, Settings, Log, Progress, and Messages) are in the same location on the default desktop layout. See The COMSOL Desktop Menus and Toolbars for more details.

You can also launch the cross-platform version on Windows using `comsolxpl.exe`.

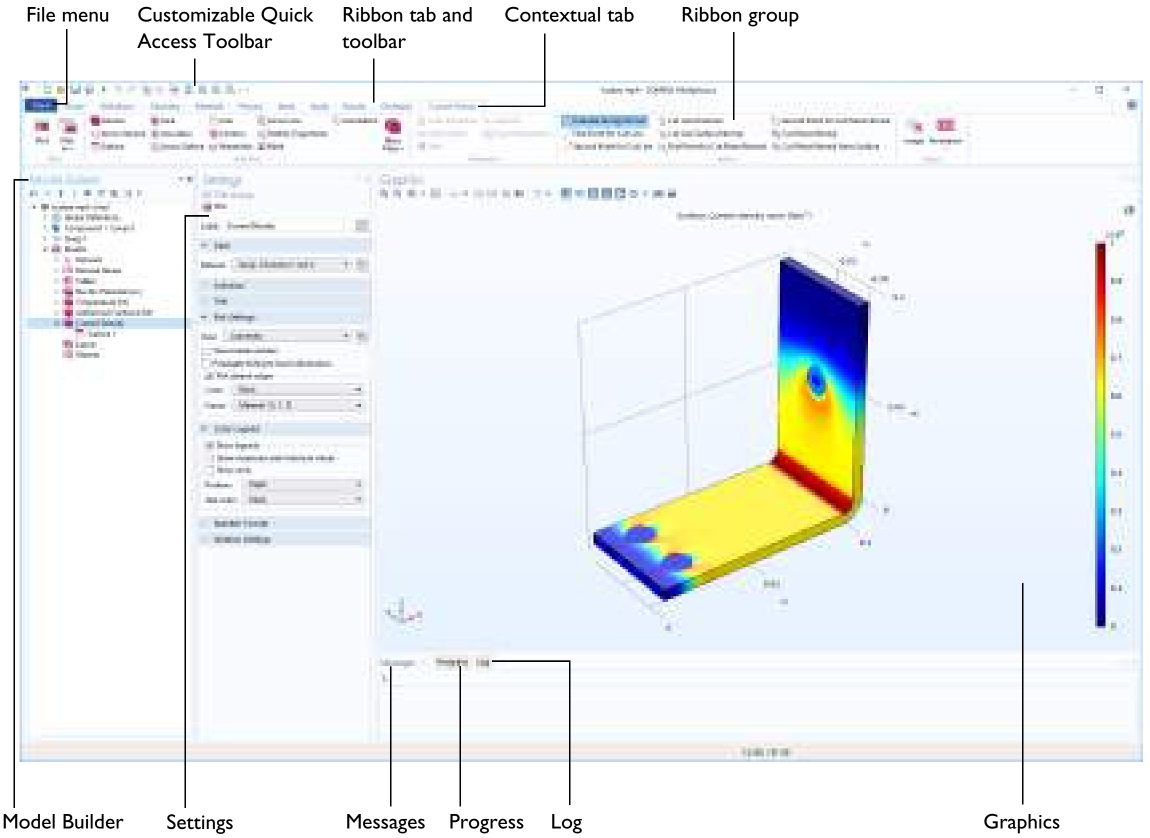


Figure 2-1: The default COMSOL Desktop with its major windows in a widescreen layout. The ribbon tabs and groups are available for Windows users. For macOS and Linux users the layout is similar but you access some options from the main menu or contextual toolbars.

Win	A *ribbon tab*, *ribbon group*, or *modal ribbon tab*, are available in the Windows version. In general, these are referred to as *toolbars*. See Figure 2-1 for an example of the Windows **Home** toolbar. Also see Figure 2-2 for an example of how the ribbon changes when a window is resized.

Mac Linux	The *Model Toolbar* and *Contextual Toolbar* are available in the cross-platform version, primarily for macOS and Linux users. See Figure 2-12 for an example of these toolbars.

ABOUT CHANGES TO THE RIBBON DISPLAY (WINDOWS USERS)

When the complete COMSOL Desktop is resized, the toolbar collapses and the buttons are grouped into menus. In Figure 2-2, all the groups in the **Home** toolbar are collapsed into menus. As the window is widened, the ribbon groups expand again to include the options as buttons or other submenus.

Figure 2-2: When the COMSOL Desktop is resized, the ribbon toolbar buttons are grouped together with the ribbon tab group name. In this example for the Home toolbar, all the buttons are available from a menu, such as Definitions, Geometry, Material, Physics, and so forth (top). As the window is widened, the menus expand accordingly (bottom).

OPENING THE APPLICATION BUILDER FROM THE COMSOL DESKTOP

When you are on the COMSOL Desktop you can toggle between the Application Builder and COMSOL Multiphysics. In the **Home** toolbar click **Application Builder** to open the Application Editor and modify the user interface of the application and to create and edit code for the application. You can also press Ctrl+Shift+A.

Conversely, when you are in the Application Builder, you can click **Model Builder** in the **Home** toolbar to return to COMSOL Multiphysics. You can also press Ctrl+Shift+M.

OVERVIEW

The rest of this section introduces you to the features of the COMSOL Desktop, explains some basic navigation, and provides you with an overview of the windows, toolbars, and menus available. In this chapter you will also learn about the model file formats, the options to save files, and the units systems available for modeling.

- Basic Navigation
- Adjusting Window Location and Size on the Desktop
- The COMSOL Desktop Windows
- The COMSOL Desktop Menus and Toolbars
- Windows Toolbars and Menus
- Cross Platform (macOS and Linux) Toolbars and Menus
- Features Available on Toolbars and From Menus
- The Messages Window
- About the COMSOL Model File Formats
- Saving COMSOL Files
- Saving and Opening Recovery Files
- The Root Settings and Properties Windows
- Unit Systems

After this introductory overview, The Application Libraries Window section explains how to work with the application libraries included with the COMSOL Multiphysics products. The Physics Interfaces section lists the interfaces available with a basic COMSOL Multiphysics license. This prepares you to start creating a new model.

The next section, Creating a New Model, shows you how to use the Model Wizard to begin building a new model by choosing a physics interface and study combination.

The last section, Toolbars and Keyboard Shortcuts, is a quick reference to all the features found on the toolbars. It includes links to the information contained throughout this reference manual.

Basic Navigation

Basic navigation on the COMSOL Desktop extensively involves the nodes in the Model Builder as well as moving between windows and sections on **Settings** windows.

WORKING WITH NODES IN THE MODEL BUILDER

The following methods are available to select nodes, expand and collapse branches, open the **Settings** window, or move up and down the nodes in the model tree:

- Click a node in the Model Builder to highlight it and to open the associated **Settings** window. See Settings and Properties Windows for Feature Nodes. You can also adjust how you are Displaying Node Names, Tags, and Types in the Model Builder.
- Once a node is highlighted, there are many things you can do; for example, you can copy, duplicate, delete, and move most nodes around. See Copying, Pasting, and Duplicating Nodes, Moving Nodes in the Model Builder, and Clearing Sequences and Deleting Sequences or Nodes.
- Right-click a node to open a context menu. See Opening Context Menus and Adding Nodes.
- Group related nodes together, for better overview and structure for the model tree. See Custom Grouping of Nodes.
- When a node is highlighted, use the up arrow key on the keyboard to move to the node above; to move to the node below, use the down arrow key.
- To expand a branch to display all nodes in the branch, click the small left-pointing white triangle next to the branch icon in the model tree, or press the right arrow key. To collapse a branch to display only the main branch node, click the small downward-right pointing black triangle next to the branch icon in the model tree, or press the left arrow key. See The Model Builder Toolbar for information about how to collapse or expand all branches.
- A highlighted node is also dynamic and its appearance can change based on where in the modeling process you are. See Dynamic Nodes in the Model Builder for a list of these visual cues.

The COMSOL Desktop Menus and Toolbars

MOVING BETWEEN WINDOWS AND SECTIONS ON THE COMSOL DESKTOP

Keyboard shortcuts are quick ways to navigate between the windows on the COMSOL Desktop and to switch focus between windows and **Settings** window sections:

- Press Ctrl+Tab to switch focus to the next window on the desktop.
- Press Ctrl+Shift+Tab to switch focus to the previous window in the desktop.
- Press Ctrl+Alt+left arrow to switch focus to the **Model Builder** window.
- Press Ctrl+Alt+right arrow to switch focus to the **Settings** window.

- Press Ctrl+Alt+up arrow to switch focus to the previous section in the **Settings** window.
- Press Ctrl+Alt+down arrow to switch focus to the next section in the **Settings** window.

The section Keyboard Shortcuts lists additional shortcuts for all operating systems.

- The COMSOL Desktop
- The Model Builder
- Creating a New Model

Adjusting Window Location and Size on the Desktop

MOVING AND RESIZING THE WINDOW

- To move a window, click-and-drag the window tab (the tab is where the window name displays, **Model Builder** for example) to where you want it.
- To resize a window, hover your mouse over the window borders until a double arrow displays. Click-and-drag the borders between windows until the layout appears as you want it to be.

At any time, in the **Home** toolbar, **Layout** group, click the **Reset desktop** button.

CLOSING A WINDOW

Not all windows in the COMSOL Desktop are closable. Windows that you can close have an **X** to the right of the window tab. Click that **X**, right-click the window and choose **Close**, or press Ctrl+F4 (Command+W on macOS). You can reopen a closed window by choosing it from the **Windows** menu (on the **Home** toolbar in the Windows® version), for the **Properties** and **Statistics** window, by right-clicking a node and choosing **Properties** or **Statistics** (for mesh statistics), respectively.

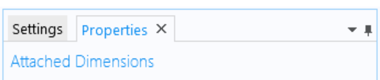


Figure 2-3: Click the X to close the Properties window. The Settings window cannot be closed.

FLOATING/DETACHING A WINDOW

Win

To detach a window to move and resize it, right-click the window tab and select **Float**. Right-click the window and choose **Dock** to return it to its default location on the Desktop.

Mac

Linux

To detach a window to move and resize it, right-click the window tab and select **Detached**. Right-click and select the option again to dock it to the COMSOL Desktop, or drag and drop it back to where you want it.

HIDING OR PINNING A WINDOW TO THE SIDE OF THE DESKTOP (WINDOWS USERS)

To hide a window, right-click the window and select **Hide**. The window is minimized along the side of the Desktop (see Figure 2-4). Hover over the name to view a hidden/minimized window. To restore a hidden window, either right-click the window, or from the list, select **Float** or **Dock**.

Pinning a window performs the same action as hiding it. Click the **Toggle hide** button in the top-right corner of any window to hide and pin it to the side of the COMSOL Desktop. To return the window to its unhidden state, hover over the window name to open it, then click the **Toggle hide** button (now laying on its side, see Figure 2-4) to restore the window to its default location.

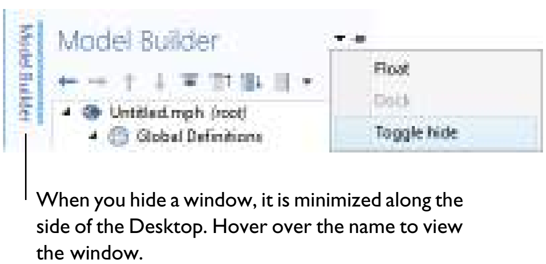


Figure 2-4: A hidden window is minimized along the side of the Desktop. Hover over the window name to view it. You can then choose to Float or Dock the window (either right-click the window or choose options from the menu), or click the Toggle hide icon to restore it to the default location on the Desktop.

USING THE POSITION GUIDES (WINDOWS USERS)

When customizing your COMSOL Desktop, or when you want to return a floating window to the Desktop (dock it), there are several visual guides available to assist.

Click and hold the mouse on a window to reposition or dock it on the Desktop. This displays the positioning guides (Figure 2-5 and Figure 2-6). Drag the window over any of the guides to highlight the area where the window is to be placed on the desktop (Figure 2-6). The center guide has five options. There are two vertical positioning guides, one on the left and one on the right of the Desktop and two horizontal positioning guides, one on the top and one on the bottom of the Desktop.

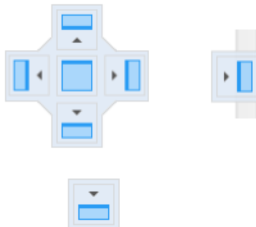


Figure 2-5: Examples of the positioning guides.

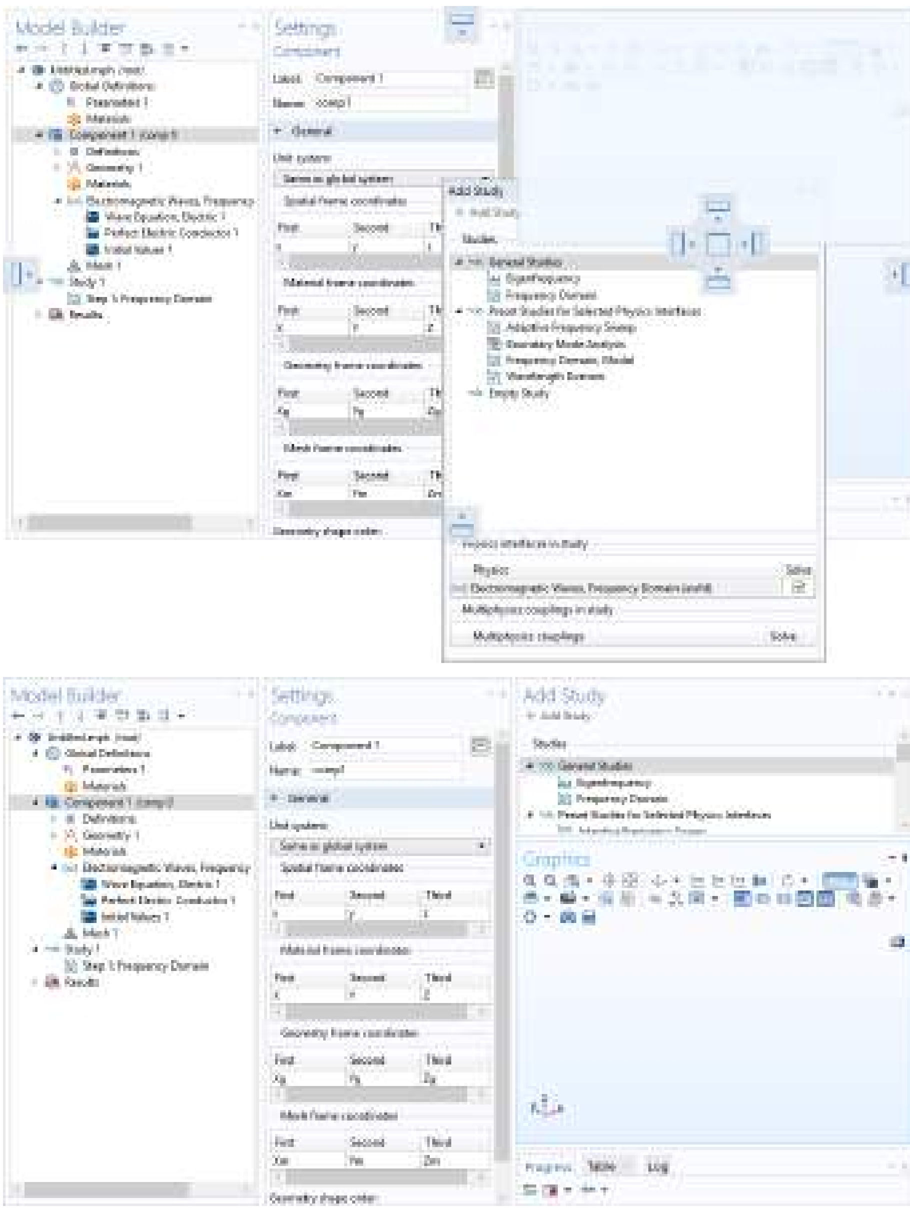


Figure 2-6: The positioning guides display (top image) when you click and hold the mouse pointer on a window. Drag the window over any of the guides to see the highlighted light blue area, which indicates the destination for the window. Release the mouse button and the window drops into place (bottom image).

RESIZABLE TABLES AND TEXT AREAS (WINDOWS USERS)

Some tables and text areas are resizable so that you can drag the area to extend it if it contains a lot of text. A border that you can drag to resize a table or text area is indicated by a thicker line:

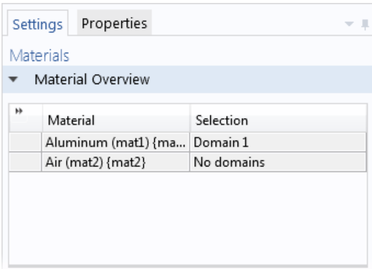


Figure 2-7: You can click and drag the thicker bottom border to resize the Material Overview table.

You can also click the **>>** button in the top-left corner to expand the table to show its contents. If you click at the border between two columns in a table, you can then drag to resize the column to the left of the border, or double-click to resize that column to fit its contents.

SELECTING, COPYING, AND PASTING IN TABLES

In tables of parameters or variables, for example, you can select a row by clicking in one of the cells, and you can select multiple rows by Shift-clicking to select a range of rows or Ctrl-clicking to select a deselect a single row. Pressing Ctrl+A when you are editing a cell selects all text in the cell. If you are not editing a cell, the pressing Ctrl+A select the entire table. You can right-click selected rows in a table and choose **Select All**.

You can also right-click to **Cut**, **Copy**, and **Paste** the selected table rows. Alternatively, you can use the corresponding keyboard shortcuts Ctrl+X, Ctrl+C, and Ctrl+V, respectively.

MOVING, MINIMIZING, AND MAXIMIZING WINDOWS (MACOS AND LINUX)

- Right-click the window tab and select **Move>View** (to move a separate window). Move the mouse to where you want the window to display and left-click to confirm the move.
- Right-click the window tab and select **Move>Tab Group** (to move several tabbed windows) from the list. Move the mouse to where you want to the group of windows to display and left-click to confirm the move.
- To resize a window, hover the mouse over the left, right, top, or bottom boundaries of the window until a double arrow displays. Drag the mouse to resize the window. Or right-click the window tab and select **Size>Left**, **Right**, **Top**, or **Bottom**. A blue line highlights the choice; drag to resize.
- To maximize and restore a window's original position, double-click a window tab to maximize it; double-click again to restore it.
- Click the **Minimize** or **Maximize** button in the top-right corner or right-click the window tab and select **Minimize** or **Maximize** from the list.

At any time, click the **Reset desktop** button in the main toolbar. The section Keyboard Shortcuts has additional shortcuts for all operating systems.

VERTICAL OR HORIZONTAL WINDOW ORIENTATION (MACOS AND LINUX)

After a window is minimized along the side of the COMSOL Desktop, you also have the option to change the window **Orientation** to **Vertical** (the default) or **Horizontal** when you click the window icon (see Figure 2-8).

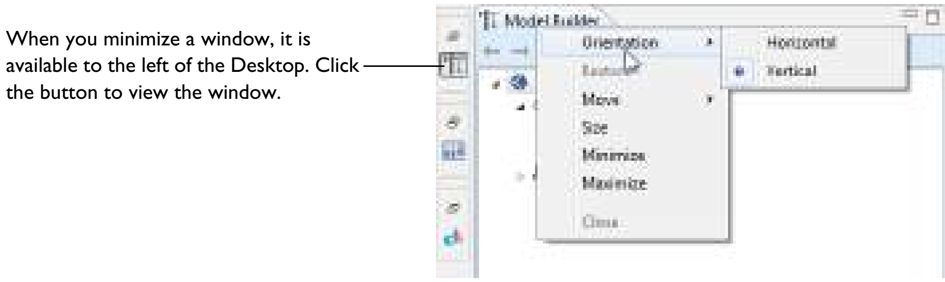


Figure 2-8: A minimized window is accessible to the left of the Desktop. Click the window icon to view it. You can then right-click the window to Move, Size, Minimize or Maximize the window. You can also change the Orientation of a minimized window to be Horizontal or Vertical when you click and view it on the Desktop in its minimized state.

- The COMSOL Desktop
- The Model Builder
- Creating a New Model

The COMSOL Desktop Windows

The COMSOL Desktop windows, including those shown in Figure 2-1, are integral to building your model. The windows listed in Table 2-1 are described throughout the documentation and the table includes links to this information. You can open windows that are currently not open in the COMSOL Desktop by choosing them from the **Windows** menu. The **Model Builder**, **Settings**, **Graphics**, and **Statistics** (for mesh statistics) windows cannot be closed and are therefore not available in the **Windows** menu.

TABLE 2-1: COMSOL DESKTOP WINDOWS

WINDOW NAME AND LINK	DESCRIPTION
BUILDING A MODEL	
The Model Wizard	Start building a model by choosing the Component space dimension, physics interfaces, and study.
The New window	Open the New window to begin modeling using the Model Wizard to start with a Blank Model. See Open a New Window to Begin Modeling.
The Model Builder	Control the modeling procedure using the model tree. This window has all the functionality and operations for building and solving models and displaying the results.
The Graphics Window	This window is a graphical view of the geometry, mesh, and results of the model. It also has useful tools to change the view and select multiple geometric entities, for example.
The Material Browser Window	Browse the material libraries and load materials into your models.
The Add Material Window	Add predefined materials.
Part Libraries	The Part Libraries contain collections of geometry parts, which serve as more advanced geometric primitives specially created for an application area.
The Add Physics Window	Add physics interfaces.
The Add Multiphysics Window	Add applicable multiphysics couplings.

TABLE 2-1: COMSOL DESKTOP WINDOWS

WINDOW NAME AND LINK	DESCRIPTION
The Settings and Properties windows	When a node is clicked in the Model Builder a corresponding Settings window opens with the same name as the node. It is a window with settings that define operations and properties specific to that node. The Properties window is accessed from the context menu and displays other node properties. See Settings and Properties Windows for Feature Nodes.
The Selection List Window	Choose objects, for example, while working with complex geometries and when you need to easily locate a geometric entity that is not easily viewed.
The Message window for geometry measurements	A tool used to measure geometry objects and entities. See Measuring Geometry Objects.
The Find Results window	Displays search results from searches performed using the Find tool. See Searching and Finding Text.
The Recovery Files window	Displays any existing recovery files and provides tools for opening, saving, and deleting recovery files. See Saving and Opening Recovery Files.
The Add-in Libraries Window	Add an available add-in in the Model Builder from the list of add-in in this window.
RESULTS AND ANALYSIS	
The Add Study Window	Add a study or studies to models.
The Plot Windows	Plot windows are also graphics windows. These plot windows display convergence results and monitor probe values while solving, for example.
The Table window	Displays the results from integral and variable evaluations defined in Derived Values nodes or by Probes and stored in Table nodes. It also displays results from nodes under an Evaluation Group node. See The Table Window and Tables Node.
The Messages Window	Contains information useful after an operation is performed.
The Progress Window	Displays the progress of the solver or mesher during the process, including a progress bar and progress information for each solver or mesher.
The Log Window	Contains information from previous solver runs, including convergence information, solution time, and memory use.
The Debug Log window	Contains debug information for model methods. See the Application Builder documentation for more information about debugging methods.
The External Process Window	Follow external processes (such as distributed batch jobs) that have been started. The window updates when you are attached to the external processes.
The Mesh Statistics Window	This window includes information about the minimum and average mesh element quality and a mesh element quality histogram, which shows the relative frequency of mesh elements with different quality values.
Comparing Models and Applications	This window includes the resulting differences from a comparison of models or applications.
APPLICATION EXAMPLES	
The Application Libraries Window	Displays all the models and applications included with an installation. The folders contain models and applications specific to the installed module.
The Application Library Update Window	A service that provides new and updated models and applications for each of the application libraries of the COMSOL products that your license includes.
HELP AND DOCUMENTATION	
The Help Window and Topic-Based Help	Provides access to context help in the COMSOL Desktop.

TABLE 2-1: COMSOL DESKTOP WINDOWS

WINDOW NAME AND LINK	DESCRIPTION
The Documentation Window	Navigate to PDF or HTML versions of the documentation (availability is based on your license), as well as Search, Bookmark, Print Topics, and Link with Contents.
The Root Settings and Properties Windows	The root node is the topmost level of the Model Builder tree. When you click this node, the root node's Settings window opens and includes detailed information about the model file.

- Creating a New Model
- The COMSOL Desktop
- COMSOL Documentation and Help
- Toolbars and Keyboard Shortcuts

The COMSOL Desktop Menus and Toolbars

The menus and toolbars available from the COMSOL Desktop vary slightly between operating systems. However, the variations are subtle and the overall functionality remains the same.

The sections Windows Toolbars and Menus and Cross Platform (macOS and Linux) Toolbars and Menus show examples of the main terms and locations of the toolbars and menus.

The Model Builder toolbar is the same for all platforms and is described in this section.

The Features Available on Toolbars and From Menus section details the available features and functions.

THE MODEL BUILDER TOOLBAR

The Model Builder toolbar is the same for all operating systems. It is located at the top of the window as shown in Figure 2-9. The actions listed in Table 2-2 are used to navigate the Model Builder tree.

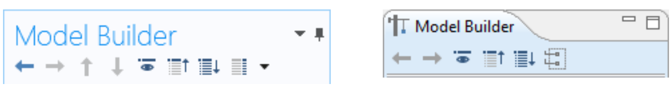


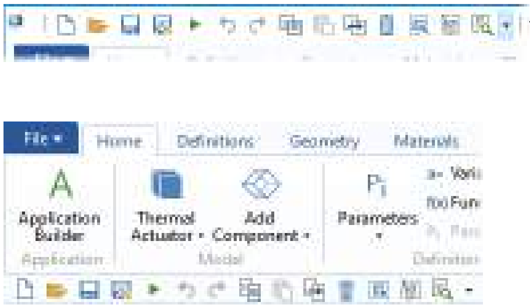
Figure 2-9: The Model Builder toolbar for Windows (left) and macOS and Linux (right).

- The COMSOL Desktop
- Creating a New Model
- The Toolbars and Keyboard Shortcuts section has detailed information about the contextual toolbars available on the COMSOL Desktop.

Windows Toolbars and Menus

The available ribbon toolbar options are dynamic, based on where in the model you are working and what is logically available for a specific task. When a blank model is created, only the default tabs are included (Model, Definitions, Study, and Results). The Physics, Geometry, and Mesh tabs are added once a model and physics interface are added to the Model Wizard, as shown in Figure 2-10.

The top of the COMSOL Desktop includes a customizable Quick Access Toolbar. Underneath this are ribbon tabs and ribbon groups, which together, are referred to as *toolbars*. The **Home** toolbar is a collection of frequently used features from all the other toolbars. For documentation purposes, a toolbar uses the same name as the tab. For example, the **Home** toolbar, **Physics** toolbar, **Geometry** toolbar, or **Study** toolbar. See The Model Builder Toolbar and Features Available on Toolbars and From Menus for a detailed list of all the features available.



The Quick Access Toolbar displays above the ribbon by default. Click the small arrow at the end to customize the toolbar.

Figure 2-10: The Quick Access Toolbar can be positioned above or below the ribbons. You can also customize the toolbar to include or exclude a variety of buttons.

CUSTOMIZE THE QUICK ACCESS TOOLBAR

The Quick Access Toolbar has several default buttons that can be displayed above or below the ribbon. Click the small arrow at the end of the toolbar to open the **Customize the Quick Access Toolbar** list. You can either edit which of the default buttons display directly from the list, or click **More Commands** to **Add** and **Remove** (or double-click to add or remove) the buttons as detailed in the section Features Available on Toolbars and From Menus. This can also be done in The Preferences Dialog Box in the **Quick Access Toolbar** section.

KEYBOARD SHORTCUTS FOR THE QUICK ACCESS TOOLBAR

You can use numeric keyboard shortcuts for the buttons on the Quick Access Toolbar. To activate those keyboard shortcuts, press the Alt key. The keyboard shortcuts (1, 2, 3, and so on) then appear underneath the Quick Access Toolbar (see the following screenshot).

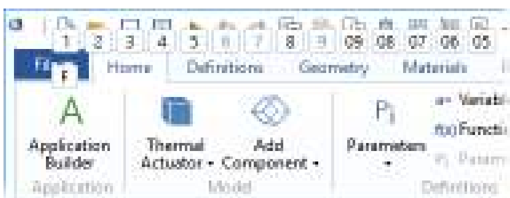


Figure 2-11: Pressing the Alt key displays the keyboard shortcuts for the Quick Access Toolbar.

DISPLAY THE QUICK ACCESS TOOLBAR ABOVE OR BELOW THE RIBBON

Right-click a ribbon to select **Show Quick Access Toolbar Above the Ribbon** or **Show Quick Access Toolbar Below the Ribbon**. These options are also available from the Customize Quick Access Toolbar menu. See Figure 2-10.

Select **Minimize the Ribbon**. To restore the ribbon, right-click anywhere in the top of the window and click **Minimize the Ribbon** to deactivate it (remove the check mark).

MINIMIZE (HIDE) THE RIBBON

Right-click anywhere on a ribbon and choose **Minimize the Ribbon** to hide the ribbon on the Desktop. To access the ribbon features, click the ribbon tab name (for example, **Model**, **Definitions**, or **Study**). The ribbon features are then available. To restore the ribbon to the top of the Desktop, right-click in the tab name area and click to remove the check mark next to **Minimize the Ribbon**.

Cross Platform (macOS and Linux) Toolbars and Menus

For cross-platform users (primarily macOS and Linux), the **Main Toolbar** is similar to the Quick Access Toolbar for Windows. In addition, there is a **Model Toolbar** and a variety of **Contextual Toolbars** available. These are a mixture of drop-down menus and buttons for frequently used actions. For documentation purposes, a toolbar uses the same name as the contextual toolbar. For example, the **Physics** toolbar, **Geometry** toolbar, or **Study** toolbar. See The Model Builder Toolbar and Features Available on Toolbars and From Menus for a detailed list of all the features available.

The Contextual Toolbar changes when you click a **Definitions**, **Geometry**, **Mesh**, **Study**, or **Results** node in the **Model Builder**. The Model Toolbar and Contextual Toolbar are similar to the ribbon toolbars for a Windows operating system.

> You can also launch the cross-platform version on Windows using `comsolxpl.exe`.

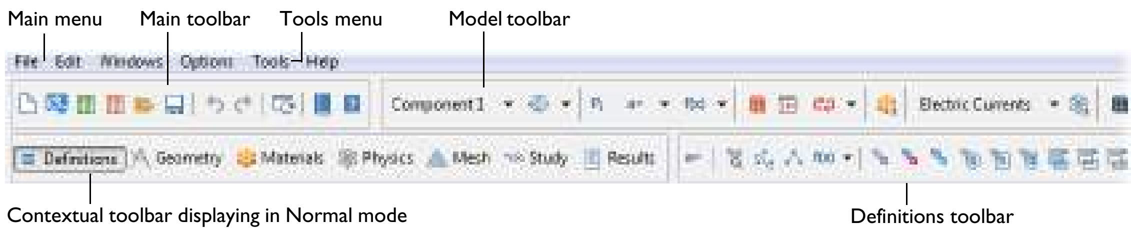


Figure 2-12: The menu and toolbar options for cross-platform users (usually macOS and Linux operating systems). Only part of the Model Toolbar and Contextual Toolbars are shown. When one of the buttons is clicked on this toolbar, the associated toolbar opens, in this example for the Definitions node. This toolbar also opens when the Definitions node is clicked in the Model Builder.

DISPLAY OR HIDE THE TOOLBARS FROM THE TOOLS MENU

From the **Tools** menu, you can choose to display or hide each toolbar. Select **Main Toolbar**, **Model Toolbar**, or **Contextual Toolbar** to turn that toolbar on or off in the COMSOL Desktop. For the **Toolbar Button Label**, you can also choose to **Show Icon Only** or **Show Icon and Text**. When **Show Icon and Text** is on it adjusts what is available on the toolbar as some buttons in the Model Toolbar display the text, while others have the label when you hover over the button. Finally, choose the **Toolbar Display Mode** as **Normal** or **Compact**. **Compact** compresses some buttons in the Contextual Toolbar and Model Toolbar under menus.

> If you are using a **Regular Screen Layout** and want to view all available buttons, the optimal settings are to set the **Toolbar Button Label** to **Show Icon Only** and the **Toolbar Display Mode** to **Compact**.

OTHER USEFUL FUNCTIONS AVAILABLE FROM THE WINDOWS MENU

From the **Windows** menu there are also other useful functions:

- Open a variety of useful windows. See The COMSOL Desktop Windows for a list and links to applicable sections.
- From the **Model Builder Node Label** submenu, choose a way to label the nodes in the Model Builder. See Displaying Node Names, Tags, and Types in the Model Builder.
- From the **Desktop Layout** submenu, choose a **Widescreen** or **Regular Screen** layout, or **Reset the Desktop**. See Customizing the Desktop Layout.

Features Available on Toolbars and From Menus

The features listed in Table 2-2 are often accessed from multiple locations. In general, the button or menu option is located as follows, with some minor differences between Windows and the cross-platform (macOS and Linux) systems and as described in Windows Toolbars and Menus and Cross Platform (macOS and Linux) Toolbars and Menus.

- The **File** menu. See Figure 2-10 (Windows) and Figure 2-12 (macOS and Linux).
- The **Model Builder** toolbar. See Figure 2-9.
- The **Quick Access Toolbar** (Windows only; see Figure 2-10). Customize the toolbar to access some of the buttons listed in the table.
- **Main Menu** and a **Main Toolbar** (macOS and Linux, see Figure 2-12).
- Additional menus along the top of the Desktop: **Edit**, **Windows**, **Options**. **Tools**, and **Help** (macOS and Linux, see Figure 2-12).

TABLE 2-2: FEATURES AVAILABLE ON VARIOUS TOOLBARS AND FROM MENUS

ICON	NAME	DESCRIPTION OR LINK TO MORE INFORMATION
Creating Models		
	New (Ctrl+N)	Open the New window to begin modeling using the Model Wizard or start with a Blank Model. See Creating a New Model. For all users, this is available from the **File** menu. It is also available on the Quick Access Toolbar (Windows users) or the Main Toolbar (cross-platform users). Additional options are available when check boxes are enabled on the Preferences dialog box under Physics Builder.
	Blank Model	Start a new blank model without any settings. This command is available after choose **File>New**. It is also available on the Quick Access Toolbar (Windows users).
Working in the Application Builder		
	Application Builder (Ctrl+Shift+A)	Toggle between the Application Builder and COMSOL Multiphysics Model Builder windows. For Windows users this is available in the **Home** toolbar and the **Developer** toolbar.
	Data Access	Add data and properties that can be modified from a running application. For Windows users this is available in the **Developer** toolbar. See Data Access in the *Application Builder Reference Manual*.
	Record Method	Record changes to the embedded model to a new method. For Windows users this is available in the **Developer** toolbar. See Recording Code in the *Application Builder Reference Manual*.
	Test Application (Ctrl+F8)	Launch the application in a separate window. For Windows users this is available in the **Developer** toolbar and on the Quick Access Toolbar. See Testing the Application in the *Application Builder Reference Manual*.
	Run Application	Run an application as a standalone application using a custom user interface. For all users, this is available from the **File** menu. It is also available on the Quick Access Toolbar (Windows users).
	Compiler	Compile an application to run separately from COMSOL Multiphysics or COMSOL Server as a standalone executable application. Requires a license for COMSOL Compiler.
Opening and Saving Files		
	Open (Ctrl+O)	Open an existing file located on the computer. For all users, this is available from the **File** menu. It is also available on the Quick Access Toolbar (Windows users) or the Main Toolbar (cross-platform users).

TABLE 2-2: FEATURES AVAILABLE ON VARIOUS TOOLBARS AND FROM MENUS

ICON	NAME	DESCRIPTION OR LINK TO MORE INFORMATION
	Recent files	From the **File** menu, select a recent file to open. For Windows users, the file is selected from the **Recent** submenu. For cross-platform users, the most recent files are listed at the bottom of the list. Windows users can also customize the Quick Access Toolbar to access this button.
	Find	Displays search results from searches performed using the Find tool. Press Ctrl+F. Windows users can also customize the Quick Access Toolbar to access this button. See Searching and Finding Text.
	Application Libraries	Open The Application Libraries Window. For Windows users, this is available in the **Home** toolbar's **Windows** menu or from the **File** menu. You can also customize the Quick Access Toolbar and then click the button. For cross-platform users, this is available from the **File** menu.
	Open Recovery File	COMSOL Multiphysics can store recovery files each time you start a solver. This is a preference setting that is initially active by default. For all users, this is available from the **File** menu. It is also available on a customized Quick Access Toolbar (Windows users). See Saving and Opening Recovery Files.
	Save (Ctrl+S)	Save the current file. For all users, this is available from the **File** menu. It is also available on the Quick Access Toolbar (Windows users) or the Main Toolbar (cross-platform users). See Saving COMSOL Files.
	Save As	Choose to save in one of the COMSOL file formats. The **Save As** window opens, and from the **Save as type** list select: **COMSOL Application (*.mph)** (the default), **Model file for Java (*.java)**, **Model file for MATLAB (*.m)**, or **Model File for Visual Basic (*.vba)**. For all users, this is available from the **File** menu. It is also available on the Quick Access Toolbar (Windows users). See About the COMSOL Model File Formats.
	Revert to Saved	Opens the last saved version of the file and reinitializes the GUI. For all users, this is available from the **File** menu. It is also available on a customized Quick Access Toolbar (Windows users) or the Main Toolbar (cross-platform users). See Reverting to the Last Saved File.
	Compact History	The files for Java and for MATLAB contain the entire model history, including settings that are no longer part of the model or application. For all users, this is available from the **File** menu. It is also available on a customized Quick Access Toolbar (Windows users). See Compacting the History.
	Run Application	Run an application created using the Application Builder. For all users, this is available from the **File** menu. It is also available on the Quick Access Toolbar (Windows users) or the Main Toolbar (cross-platform users).
COMSOL Multiphysics Server		
	Connect to Server	To connect to a server from the COMSOL Desktop. For all users, this is available from the **File>COMSOL Multiphysics Server** menu. It is also available on a customized Quick Access Toolbar (Windows users). See Connecting to a COMSOL Multiphysics Server.
	Disconnect from Server	To close the connection to the server or MATLAB. For all users, this is available from the **File>COMSOL Multiphysics Server** menu. It is also available on a customized Quick Access Toolbar (Windows users). See Disconnecting from a COMSOL Multiphysics Server.

TABLE 2-2: FEATURES AVAILABLE ON VARIOUS TOOLBARS AND FROM MENUS

ICON	NAME	DESCRIPTION OR LINK TO MORE INFORMATION
	Import Application from Server	To import a particular application when working with MATLAB, Excel, or the COMSOL API. For all users, this is available from the **File>COMSOL Multiphysics Server** menu. It is also available on a customized Quick Access Toolbar (Windows users). See Working with MATLAB, Excel, or the COMSOL API.
	Remove Applications from Server	To delete applications (remove them from the server) that you have created using `ModelUtil`. For all users, this is available from the **File>COMSOL Multiphysics Server** menu. It is also available on a customized Quick Access Toolbar (Windows users). See Working with MATLAB, Excel, or the COMSOL API.
Model Builder Toolbar		
	Previous Node (Alt+Left) Next Node (Alt+Right)	Navigate back to the node previously selected or to the next node in the sequence. See also Keyboard Shortcuts.
	Show More Options	Click to select options to display from the **Show More Options** dialog box. See Showing More Options.
	Collapse All Expand All	Click to collapse or expand all nodes in the model tree, except the top nodes on the main branch.
	Model Builder Node Label	Choose to display any combination of Name, Tag, or Type. See Displaying Node Names, Tags, and Types in the Model Builder and Settings and Properties Windows for Feature Nodes.
Undo, Redo, Copy, Paste, Duplicate, and Delete		
	Undo (Ctrl+Z) Redo (Ctrl+Y)	Undo and Redo the last operation for some operations (such as adding, disabling, moving, and deleting nodes in the Model Builder) as well as changing values in the Settings window. For Windows users, this is available on the Quick Access Toolbar. For cross-platform users, this is available in the Main Toolbar or from the **Edit** menu. See Undoing and Redoing Operations.
	Copy	Copy, paste, and duplicate some features. Also right-click a node to select one of these options from the context menu. For Windows users, this is available on the Quick Access Toolbar. For cross-platform users, this is available in the Main Toolbar or from the **Edit** menu. See Copying, Pasting, and Duplicating Nodes.
	Paste	
	Duplicate	
	Delete (Del)	Delete some nodes while building a model, or delete a selected geometry object. Also press the Del key or right-click a node to select this option from the context menu. For Windows users, this is available on the Quick Access Toolbar. For cross-platform users, this is available in the Main Toolbar or from the **Edit** menu. See Clearing Sequences and Deleting Sequences or Nodes.

TABLE 2-2: FEATURES AVAILABLE ON VARIOUS TOOLBARS AND FROM MENUS

ICON	NAME	DESCRIPTION OR LINK TO MORE INFORMATION
	Select All	To select all or clear the selection of all geometric entities in the Graphics window, click the **Select All** or **Clear Selection** buttons, respectively. For Windows users, this is available on the Quick Access Toolbar. For cross-platform users, this is available from the **Edit** menu. See Selecting and Clearing Selection of Geometric Entities.
	Clear Selection	
Other		
	Reset Desktop	Set the COMSOL Desktop back to widescreen or regular screen, or reset it to default settings. For Windows users, this is available in the **Home** toolbar, in the **Layout** menu. You can also customize the Quick Access Toolbar and then click the button. For cross-platform users, this is available in the Main Toolbar or from the **Windows>Desktop Layout** menu. See Customizing the Desktop Layout.
	Licensed and Used Products	Open the Licensed and Used Products window to list or block the products your license includes. See Checking and Controlling Products and Licenses Used. For Windows users, this is available from the **File** menu. You can also customize the Quick Access Toolbar and then click the button. For cross-platform users, this is available from the **Options** menu.
	Preferences	To make changes to how items are displayed or available throughout the COMSOL Multiphysics software. See Preferences Settings. For Windows users, this is available from the **File** menu. You can also customize the Quick Access Toolbar and then click the button. For cross-platform users, this is available from the **Options** menu.
	Measure	Measure geometric properties such as volumes (see Measuring Geometry Objects). Available in the **Geometry** toolbar. Also, for Windows users, you can customize the Quick Access Toolbar and then click the button.
	Group (Ctrl+G)	Group related and similar nodes under a Group node. Available in several parts of the model tree (see Custom Grouping of Nodes). Also, for Windows users, you can customize the Quick Access Toolbar and then click the button.
	Ungroup (Ctrl+Shift+G)	Ungroup nodes that are grouped under a **Group** node. Available in several parts of the model tree (see Custom Grouping of Nodes). Also, for Windows users, you can customize the Quick Access Toolbar and then click the button.
Help and Documentation		
	Help (F1)	Open the context help. See COMSOL Documentation and Help. For Windows users, this is available from the **File>Help** menu or in the upper-right corner of the Desktop. For cross-platform users, this is available in the Main Toolbar or from the **Help** menu.
	Documentation (Ctrl+F1)	Open the Documentation. For Windows users, this is available from the **File>Help** menu. For cross-platform users, this is available in the Main Toolbar or from the **Help** menu. See COMSOL Documentation and Help.
	Application Gallery	Go to the online Application Gallery on the COMSOL website. For Windows users, this is available from the **File>Help** menu. For cross-platform users, this is available from the **Help** menu.

TABLE 2-2: FEATURES AVAILABLE ON VARIOUS TOOLBARS AND FROM MENUS

ICON	NAME	DESCRIPTION OR LINK TO MORE INFORMATION
	Support Center	Go to the online Support Center on the COMSOL website. For Windows users, this is available from the **File>Help** menu. For cross-platform users, this is available from the **Help** menu.
	Training	Go to the Training page on the COMSOL website. For Windows users, this is available from the **File>Help** menu. For cross-platform users, this is available from the **Help** menu.
	Check for Product Updates	For Windows users, this is available from the **File>Help** menu. For cross-platform users, this is available from the **Help** menu. See Checking for Product Software Updates.
	Update COMSOL Application Libraries	For Windows users, this is available from the **File>Help** menu. For cross-platform users, this is available from the **Help** menu. See The Application Library Update Window
	Update COMSOL Part Libraries	For Windows users, this is available from the **File>Help** menu. For cross-platform users, this is available from the **Help** menu. See The Part Library Update Window
	About COMSOL Multiphysics	For Windows users, this is available from the **File>Help** menu. For cross-platform users, this is available from the **Help** menu. See The About COMSOL Multiphysics Box.

The Messages Window

The **Messages** window ( ) displays by default and contains information useful to you after an operation is performed.

The information in this window includes:

- Details about opening and saving model files such as MPH-files.
- Information about geometry objects imported from CAD files.
- On the **Mesh** and **Geometry** toolbars, click the **Measure** ( ) button to view information about:
  - The geometry finalization (forming a union or an assembly) and about the number of geometric entities (domains, boundaries, and so on) in the finalized geometry.
  - The number of mesh elements and degrees of freedom in the model.
- Solution times.
- Error messages. The messages are in chronological order and can be scrolled through.

By default, the messages are preceded by a timestamp, providing the current date and time for the message. To turn off the timestamps, open the **Preferences** dialog box and clear the **Add timestamps to messages** check box on the **General** page under **Log and messages**.

To open the **Messages** window:

- From the **Home** toolbar (Windows users) select **Windows>Messages**.

- From the main menu (macOS and Linux users) select **Windows>Messages**.
- To clear the window of all messages, click the **Clear** button ( ).

- Meshing
- Geometry Modeling and CAD Tools
- Studies and Solvers

About the COMSOL Model File Formats

Below find a list of the COMSOL file formats: MPH-files for applications and models, model files for Java, model files for MATLAB, and model files for VBA.

THE ROOT NODE

When you first open or create a new model, the *root node* ( ) is the topmost level of the tree. By default, unnamed files are called `Untitled.mph`. The filename changes when the file is saved for the first time, but the root node *Name* does not change for this top level node. See The Root Settings and Properties Windows for details about the settings available when this node is clicked.

COMSOL MPH-FILES

The default standard file format with the extension `.mph`. The files contain binary and text data. The mesh and solution data are stored as binary data, while all other information is stored as plain text.

You can quickly save and load MPH-files. All models and applications in the COMSOL Application Libraries in the modules are saved as MPH-files.

The MPH-files in the COMSOL Application Libraries can have two formats:

- *Solved MPH-files* include all meshes and solutions. In the **Application Libraries** window these files appear with the icon . If the MPH-file's size exceeds 25MB, a tooltip with the text "Large file" and the file size appears when you position the cursor at the node in the **Application Libraries** tree.
- *Compact MPH-files* include all settings but have no built meshes and solution data to save space (a few compact MPH-files have no solutions for other reasons). You can open these to study the settings and to mesh and re-solve it. It is also possible to download the full versions — with meshes and solutions — of most of these through Application Library Update (see The Application Library Update Window). In the **Application Libraries** window these appear with the icon . If you position the cursor at a compact file in the Application Libraries window, a **No solutions stored** message appears. If a full MPH-file is available for download, the corresponding node's context menu includes a **Download File With Solution** icon ( ).

File Locking

Only one user can open and edit an MPH-file at the same time. If you try to open an MPH-file that is already open in another user's COMSOL Desktop, that MPH-file is locked, and you get an option to open the MPH-file in a read-only mode (click **Open As Read-Only**). That means that you can edit the model but you cannot save it unless you save the MPH-file under another name. When an MPH-file is locked, COMSOL creates a separate lock file with the same filename as the MPH-file plus the extension `.lock`, stored in the same directory as the locked MPH-file. If a lock file remains after all COMSOL Desktop sessions have ended (which can happen if the

COMSOL Desktop session is ended in a nonstandard way), you can reset the lock when trying to open the file the next time by clicking **Reset Lock and Open**.

Linux and macOS do not support operating system locking of files. On those platforms, locking is supported to help users avoid editing the same COMSOL Multiphysics model file, but it is possible to ignore the file locking and delete the lock files.

MODEL FILES FOR JAVA

Editable script files that contain sequences of COMSOL commands as Java code (see the *COMSOL Multiphysics Programming Reference Manual* for more information about these commands). You can compile these Java files and run them as separate applications. Edit the files in a text editor to add additional commands. For parts of the model that appears under a component (such as the geometry, physics, and mesh), the default format is to include the component level in the Java code. If you want do not want to include the component level, clear the Use component syntax check box on the **Methods** page in the **Preferences** dialog box.

When saving a Model Java-file history for running a method call in the COMSOL Desktop, it contains the history produced while running the method call and not the method itself.

MODEL FILES FOR MATLAB

Model files for MATLAB are editable script files (M-files), similar to the model files for Java, for use with MATLAB. A model file for MATLAB contains a sequence of COMSOL commands as an M-file. You can run these model files in MATLAB like any other M-file scripts. You can also edit the files in a text editor to include additional COMSOL commands or general MATLAB commands.

Running model files in the M-file format requires LiveLink™ *for* MATLAB®.

MODEL FILES FOR VBA

Model files for VBA are editable script files (VBA-files), similar to the model files for Java, for use with VBA (Visual Basic for Applications) in Microsoft Excel®. A model file for VBA contains a sequence of COMSOL commands as a VBA-file (extension .vba). You can use these files from Excel® to access settings and data in COMSOL models.

Using model files in the VBA format requires LiveLink™ *for* Excel®.

- The Application Libraries Window
- Saving COMSOL Files
- Reverting to the Last Saved File
- Printing and Capturing Screenshots
- Saving and Opening Recovery Files

Saving COMSOL Files

The following options are selected from different menus and toolbars as described in The COMSOL Desktop Menus and Toolbars.

SAVING A NEW MODEL OR APPLICATION

If this is the first time saving a model or application, or if you want to update the file and keep the current name and format, in general, these are the ways to save a model:

- Click the **Save** button ( ) on the Quick Access Toolbar or Main Toolbar.
- Press Ctrl+S.
- Select **File>Save**.

If you save a model that was previously saved in an earlier version of COMSOL Multiphysics, you get a question about if you want to continue to save the model file, thereby overwriting the original file with a new file that has been converted so that it can only be opened in version 5.4. Select the **Do not show this message again** check box if desired. There is also a preference setting on the **Files** page in the **Preferences** dialog box. Under **Saving COMSOL application files**, clear the **Warn before overwriting a file saved by an older version of COMSOL** check box (selected by default) to turn off this question.

CREATING A COPY USING SAVE AS

If the model has been saved before and you want to create a copy you can choose to save in one of the COMSOL file formats (see COMSOL MPH-Files, Model Files for Java, and Model Files for MATLAB).

Select **File>Save As**. The **Save As** window opens, and from the **Save as type** list select **COMSOL Application (*.mph)** (the default), **Model file for Java (*.java)**, **Model file for MATLAB (*.m)**, or **COMSOL File for VBA (*.vba)**.

In all cases, navigate to the location where you want to save the model, enter a **File name**, and then click **Save**.

You can add the author to the header of model files for Java and for MATLAB that are saved. Open The Preferences Dialog Box and under **General>History export**, select the **Include author** check box.

REVERTING TO THE LAST SAVED FILE

To open the last saved version of the file and reinitialize the GUI, select **File>Revert to Saved** ( ). For Windows users, you can also customize the Quick Access Toolbar and then click the **Revert to Saved** button.

COMPACTING THE HISTORY

The COMSOL files for Java and for MATLAB contain the entire history of the model, including settings that are no longer part of it. To compact the history so that the files only include the settings that are part of the current model, select **File>Compact History**. For Windows users, you can also customize the Quick Access Toolbar and then click the **Compact History** ( ) button.

Compacting the history works best if you make sure that the geometry is built before running **File>Compact History**.

WHEN SAVING, OPTIMIZING FOR SPEED OR FILE SIZE

You can choose to save COMSOL files (MPH-files) that are optimized for speed (the default, using uncompressed files that are faster to save) or optimized for file size (using compressed files). In the **Preferences** dialog box (see

Preferences Settings), click **Files** and choose **Speed** or **File size** from the **Optimize for** list under **Saving COMSOL application files**.

- About the COMSOL Model File Formats
- Windows Toolbars and Menus
- The Root Settings and Properties Windows
- Printing and Capturing Screenshots

Saving and Opening Recovery Files

The COMSOL Multiphysics software can store recovery files each time you start a solver. This setting is initially active by default. If the server has lost contact with the client, a recovery file is also saved or kept.

If any recovery files exist, the **Recovery Files** window is open when you start the COMSOL Desktop. You can also open the **Recovery Files** window from the **Windows** menu.

The update of the recovery file occurs at the following events:

- After completing the solution for each time step specified as the output times in the **Times** field for a time-dependent simulation.
- After completing each parameter step in a parametric simulation.
- After completing each successful Newton iteration for a nonlinear stationary simulation.

The recovery files are COMSOL MPH-files that represent the state at the time that they were saved. They make it possible to recover from a solver error, which can be especially useful for long time-dependent or parametric runs.

You can control the use of recovery files using the buttons at the top of the Recovery Files window:

- Click the **Open** button ( ) to open a selected recovery file. Opening the recovery files this way ensures that the recovery file can be deleted without compromising the open model. It does not remove the recovery file when the open model is saved though
- Click the **Save and Open** ( ) button saves the recovery file to a Model MPH-file and open it. It also removes the recovery file.
- Click the **Save As** button ( ) to save the recovery file to a Model MPH-file and remove the recovery file.
- Click the **Delete** button ( ) or press Delete to remove the selected recovery files.

The COMSOL Multiphysics software keeps track of the computed time steps or parameter steps in the recovery file, so right-click the **Study** node and select **Continue** ( ) to continue the computation from the point where it was stored in the recovery file. If you are solving a stationary nonparametric problem, the last converged Newton iteration is stored in the recovery file; selecting **Continue** causes the software to resume solving from this stored state.

You can make changes to these default settings in The Preferences Dialog Box in the **Files** section.

- The **Save recovery file** check box is selected by default to save recovery files to disk during the solution process for time-dependent, parametric, and nonlinear solvers.
- The **Check for recovery files at launch** check box is selected by default so that the **Recovery Files** window opens at launch when there are any recovery files.
- In the **Folder for recovery files** field, you can specify a different folder from the default to, for example, use a folder on the server where there is more disk space for storing large recovery files. Click **Browse** to browse to a recovery file folder.

- In the **Folder for temporary files** field you can specify a different folder than the default to, for example, use a folder where there is more disk space for storing large temporary files. Click **Browse** to browse to a folder for temporary files.
- If you run the COMSOL Multiphysics software in a client-server configuration, you can specify a **Folder for temporary files on client** and a **Folder for temporary files on server**.

Studies and Solvers

The Root Settings and Properties Windows

The root node is the topmost level of the Model Builder tree and the Explorer tree in the Application Builder. When you click this node, the **Settings** window for the **root** node (, or in the Application Builder) opens and includes detailed information about the model file. To open the corresponding **Properties** window, right-click the **root** node and choose **Properties** from the context menu.

ROOT SETTINGS WINDOW

Some of the fields can be edited directly in this window, while others display system information that cannot be changed, or information that changes as updates are made throughout the model (for example, adding a node that requires an additional module license).

- **Protection**: Click **Set Password** next to **Editing not protected** or **Running not protected** to enter a password in the **Protect Edit with Password** or **Protect Running with Password** dialog boxes. To change the password, click Change Password to enter the previous password and a new password in the **Protect Edit with Password** or **Protect Running with Password** dialog boxes. Note that, for the password that protects editing, lost passwords cannot be recovered.
- **Used Products**: The information included here is based on the purchased license or modules. See Checking and Controlling Products and Licenses Used. Also see The About COMSOL Multiphysics Box.
- **Presentation**: In this section you can specify a title and a description of the model. By default, these texts are used on the title page of a report; see The Title Page. Under **Computation time**, you can enter an expected computation time in the **Expected** field. Also, the time after **Last** is the last measured computation time (if available). To illustrate the model you can also set an image as a thumbnail that displays in this section, when opening a file in the **Application Libraries** window, and as the default report title page image. See the section Setting and Clearing the Thumbnail Image for details about how to do this.
- **Unit System**: The default unit system is SI units. Or select any other option from the list. See Unit Systems and Setting the Unit System for Models to change the setting globally or locally.
- **Graphics**: For specifying the color theme used for selection colors, use the **Color theme** list, where you can select from all available color themes. The default is to use the default color theme — for example, **Default from preferences (Default)**. You specify the default color theme in the **Preferences** dialog box, in the **Color theme** list on the **Graphics and Plot Windows** page. See Selection Colors for more information about selection colors.

  For the **Font** settings, the default is to use the font family that is set in the **Preferences** dialog box — on the **Graphics and Plot Windows** page under **Default font** — with a default font size. Depending on the operating system and the installed fonts on the computer, you can select from a number of other font families from the **Family** list. The default font is indicated in the default settings for **Family** (**Default from preferences (Vera)**, for example). For the font size, the **Default size** setting in the **Size** list uses the font size that is used in the graphics and plots, which is a system-dependent value. You can also choose another font size between 6 and 24 points or type a font size in the **Size** combined field and list. The font and the font size affect text in the Graphics window and other plot windows in the COMSOL Desktop and in Graphics form objects in the Application Builder. See Changing the Font for Plot Labels and Titles to make global changes.

- **Applications** (this section is only available from the Application Builder window): Select the **Ask to save application when closing** check box to ask users if they want to save changes in an application when closing it. Also, from the **When starting with COMSOL Multiphysics** list, you can control the behavior when a user starts COMSOL Multiphysics with the option to open an application or when a user double-clicks an MPH-file in Windows: Select **Edit application** (the default) to open the COMSOL Desktop for editing the application, or select **Run application** to launch and run the application directly.

  Select the **Ignore license errors during launch** check box to make it possible to start the application even if not all required licenses are available. It is, however, still not possible to use products when the license is not available, so available API methods will typically have to be used in the application to limit the application's functionality depending on what licenses are available.

ROOT PROPERTIES WINDOW

To access the **Properties** window, right-click the **root** node and choose **Properties** from the context menu.

> You can change a filename by saving the file, but the root node *Name* (`root`) cannot be changed for this top level node. This is different than for other nodes in the tree, where the name can be edited. See Displaying Node Names, Tags, and Types in the Model Builder for information. Also see Settings and Properties Windows for Feature Nodes.

- **File**: The file location where a file is saved. This field cannot be edited, but is automatically updated when the file is saved to a new path. Also see Documentation and Application Libraries Root Directories.
- **Version**: Includes the version and build of the COMSOL Multiphysics instance that you are running. This information is system generated.
- **Created** and **Last modified**: These sections cannot be edited and are automatically generated based on the computer system or network time and date settings.
- **Saved with license**: The license number of the installed software that the model or application was saved with is included here. Also see The About COMSOL Multiphysics Box.
- **Application version**: Enter a tracking version number for the model or application (for example, `Internal Draft V1`, `Sales Demonstration V2`, or `Version B`).
- **Search path**: Enter a file path to set a search path for external files. This path corresponds to `model.modelPath(<path>)` in the COMSOL Multiphysics API.

Unit Systems

The COMSOL Multiphysics software supports the following unit systems:

METRIC UNIT SYSTEMS

- SI units, the International System of Units (SI, Système International d'Unités). This is the default unit system (sometimes also called MKS). For a list of SI units in COMSOL Multiphysics, see SI Base, Derived, and Other Units.
- CGSA units. The CGS system uses centimeter, gram, and second as basic units of length, mass, and time, respectively. The remaining basic units are identical to the SI units. The CGS unit system gives nice values for small lengths, masses, forces, pressures, and energies when working on a microscale and with weak electromagnetic forces. The derived units of force, pressure, and energy have well-known and widely used names:

dyne, barye, and erg, respectively. CGSA adds *ampere* as the basic unit for electric current. For a list of CGSA units, see Special CGSA Units.

- Electromagnetic units (EMU). This system is based on Ampère's law, which defines the unit of electric current once you select an appropriate value for the constant C. When dealing exclusively with magnetic effects, it is convenient to set $C = 1$. If CGS units are used for the remaining basic dimensions, the current unit is called an *abampere*, and the corresponding coherent unit system is called electromagnetic units. Unique names for derived units have been introduced by prefixing the SI name with *ab-*. For a list of EMU units, see Special EMU Units.
- Electrostatic units (ESU). Based on Coulomb's law for the force between point charges, ESU uses a unit of charge called the *statcoulomb* with CGS units for length, mass, and time. From there, the *statampere*, or *franklin*, and other derived units of the electrostatic unit system follow. For a list of ESU units, see Special ESU Units.
- MPa units. For stationary structural mechanics, where the density does not appear in the equations, it can be convenient to use a system where newton and megapascal (hence the name "MPa system") are naturally derived units of force and pressure, respectively. Keeping the SI unit for time, the basic units of length and mass become millimeter and tonne. Except for the force and pressure units, other derived units are nameless. For a list of MPa units, see Special MPa Units.

ENGLISH UNIT SYSTEMS

- Foot-pound-second unit system (FPS units). The original foot-pound-second system seems to be the absolute system using the pound as a unit of mass. This version of the FPS system is in agreement with the IEEE standard (the pound is a unit of mass and not of force). The natural derived unit of force is the *poundal*. For a list of FPS units, see Special FPS Units.
- British engineering units. An alternative to the standard FPS system is the British engineering unit system (also called gravitational foot-pound-second system or foot-slug-second system). Here, the pound force is the natural unit of force, which causes the introduction of the mass unit *slug* such that a pound force is a slug-foot per second squared. For a list of British engineering units, see Special British Engineering Units.
- Inch-pound-second unit system (IPS units). It is possible to define varieties of the FPS and British engineering systems based on the inch instead of the foot as basic unit of length. This gives rise to two distinct inch-pound-second systems: the *absolute IPS system* (just called IPS) and the *gravitational IPS system*. For a list of IPS units, see Special IPS Units.
- Gravitational IPS units. This alternative IPS unit system considers the pound a unit of weight rather than a unit of mass. For a list of Gravitational IPS units, see Special Gravitational IPS Units.

OTHER

- None. No units appear in the settings, which can be useful in nondimensionalized (de-dimensionalized or dimensionless) models.

- Using Units
- Setting the Unit System for Models

Searching and Finding Text

Press Ctrl+F to open a **Find** tool that you can use to search for variables or text in all of the model or, for application development, only in methods. In the **Find** tool, click **All** to search the entire model, including user interface components, variable definitions, model entity tags, identifiers, and labels. You can specify to search using an **Exact match**, a **Regular expression**, or a **Case sensitive** search by selecting the corresponding check boxes. Windows users can also customize the Quick Access Toolbar to access this button ( ).

Click **Methods** to find and optionally replace a text string in methods developed for an application. See the Application Builder documentation for details.

Click **Advanced** to access some advanced search tools. See Advanced Search Options below.

Click the **Find** button to launch the search. The search results for each search appears in a separate **Find Results** window, where each occurrence of the search string appears in a row. Double-click the row to open the node or method and highlight the search result in the Settings window or method where it occurs. The **Node** column lists the node where the search string appears; the **Type** column lists the type of the search results, such as **Setting**, **Description**, or **Method**); and the **Text** column shows the text in which the search string appears.

ADVANCED SEARCH OPTIONS

Click **Advanced** to enable a more specific search method where individual types of candidates can be searched for. In the **Find** field, type a search query in the same way as on the **All** page using an **Exact match**, a **Regular expression**, or a **Case sensitive** search, if desired. In the **Filter** section, below the **Find** field, use the **Include** list to specify what to include: **Node tags**, **Node labels**, **Node names**, **Node types**, **Descriptions**, and **Settings**. The **Include** list specifies what you search for, so selecting **Node tags**, for example, only matches the query in the **Find** field against node tags. The option **Descriptions** searches for any fixed text in the COMSOL Desktop but not text that can be changed such as lists. The **Settings** option searches for values of a setting (for example, text in a text field or in a list). In addition, it is possible to add a name query in the **Name filter** field. Any nonempty entry here will only include settings with a specific name, which is the description of the setting. This query is also affected by the settings for the **Exact match**, **Regular expression**, and **Case sensitive** check boxes. Select the **Include API names in search** check box to allow the search filter to match against the API names. It is only applicable when searching for **Node types** and **Settings** and checks the API names of the following search candidates:

- The query in the **Name filter** field is matched against the API name of a setting (for example, the **Axis type** list in a **Block** geometry feature has `axistype` as its API name).
- The query in the **Find** field is matched against the API value of list settings (for example, the option **x-axis** in the **Axis type** list has the API value `x`).
- A search for the value of a check box (typically `on` or `off`) is ignored in normal searches but is active when including API names. The **Find** field can then match values that are on or off for most check boxes.
- For the **Node types** option in the **Include** list, which otherwise searches for the descriptive string of the type but also includes the API name of the type (for example, the **Work Plane** geometry feature has the API type `WorkPlane`).

ADDITIONAL SEARCH TOOLS FOR THE PHYSICS BUILDER

When searching in the Physics Builder, you can also control the scope of the search in the **Find** tool. In addition to physics builder files, it can also search in linked files. A linked file can either be a file in an **Import** node under the **External Resources** branch (downward search), or a file listed in the **External Resources** links in the table of the root node's Setting window (upward search). The search can look for files in the upward direction by selecting the **Follow link sources** check box, and it can look for files in the downward direction by selecting the **Follow imports** check box. A **Depth** value can be used to limit the search. Under the **Links** tab there is a specialized search that finds links to a selected node. It supports the **Follow link sources** option, and only need to search upward one level. It is not applicable for nodes that cannot be a target of a link.

The Application Libraries Window

The **Application Libraries** window (Figure 2-13) contains models and applications that you can use for a variety of purposes: for learning how to build COMSOL models, as starting points for your own models and applications, and as demonstrations of specific functionality. Each add-on module includes its own application library with information about how to use the module within its application areas. Each file includes full documentation and a brief description, including the solution times and information about the computer used for solving the model.

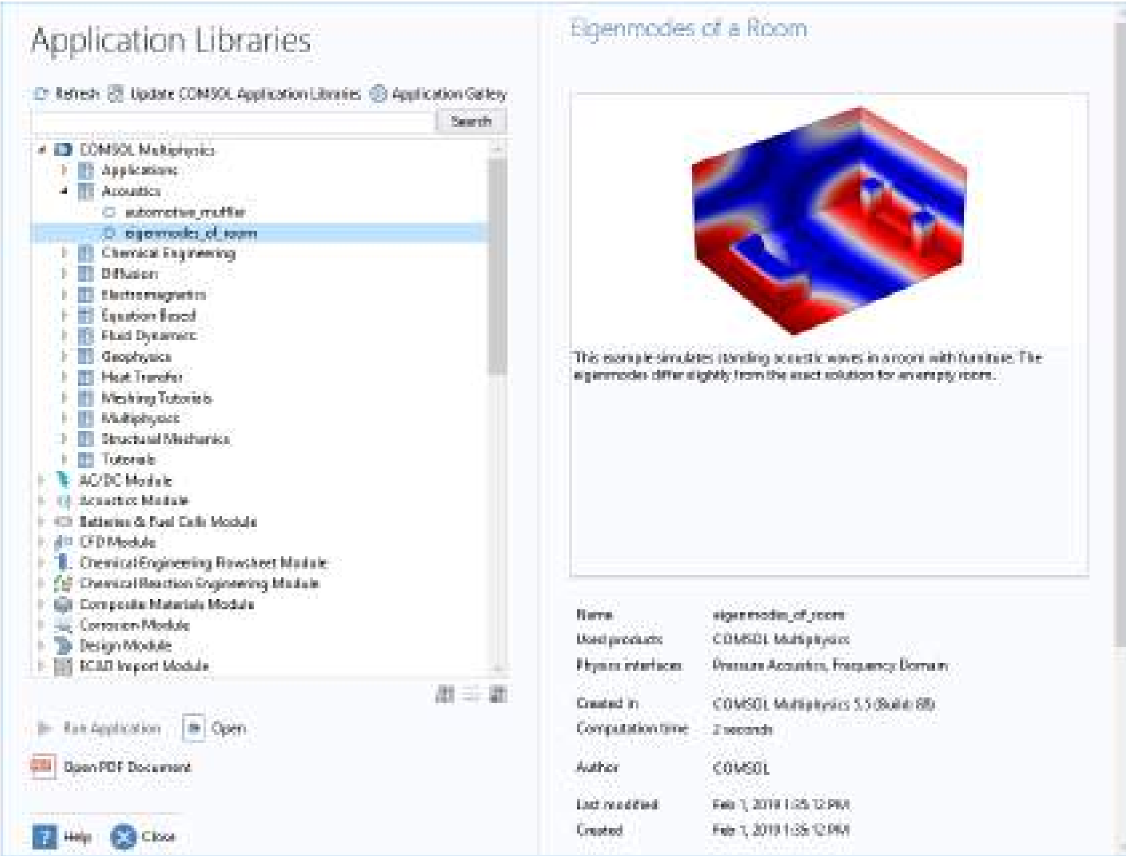


Figure 2-13: The Application Libraries window with Eigenmodes of a Room highlighted in the tree. Specific information about the model or application is displayed to the right, including its name and the computation time.

Win

To open the **Application Libraries** window ( ):

- From the **Home** toolbar, **Windows** menu, select **Application Libraries**.
- You can also customize the Quick Access Toolbar and then click the **Application Libraries** button on the toolbar. See Windows Toolbars and Menus.
- From the **File** menu select **Application Libraries**.

Mac

Linux

To open the **Application Libraries** window ( ):

- In the Main toolbar, click the **Application Libraries** button.
- Select **Windows>Application Libraries**.

Browse through the **Application Libraries** tree to see what is available. Click to highlight the file in the tree and read information about it to the right, or search for a specific application. The information for each application includes:

- The COMSOL products used.
- The physics interfaces used.
- The version that the application was created in.
- The computation time. If you hover over the computation time, a tooltip displays information about the computer used for the computation (CPU, clock rate, and number of cores) and, if applicable, the solution times for each study step.
- The dates and times for when the application was created and last modified.

When you browse the tree, you may notice that three different icons are used for the model and application nodes. Their respective look and significance are as follows:

- (*solved*) — The file is complete with built meshes and solutions.
- (*compact*) — The file contains no stored meshes or solutions. A solved version is available for download via Application Library Update (except in a few cases where the file is a template related to another, regular, model).
- (*preview*) — The file is a preview of an application or a model file, containing only what is needed to represent the model in this particular context, including the model description and information about used products and physics interfaces and the computation time. Solved and compact versions are available for download. Model documentation will be available if you have specified the use of online help (see The Help Window and Topic-Based Help for details on how to do this).

The following sections describe what is available and what you can do from the **Application Libraries** window:

- The Applications Folders
- Running or Opening a Model or Application and Its Documentation
- Downloading MPH-Files With or Without Solutions
- Searching the Application Libraries
- The Application Library Update Window

You can also set the root directory and create and remove a user-defined library as described next.

APPLICATION LIBRARY PREFERENCES

The following settings can be modified using the buttons at the bottom of the **Application Libraries** tree on the **Application Libraries** page in The Preferences Dialog Box and — if the **Allow managing libraries in the Application Libraries window** check box on that page is selected (the default) — also in the **Application Libraries** window itself.

Add User Application Library

Click the **Add User Application Library** button ( ) to add customized folders. In the **Add User Application Library** dialog box, navigate to a location on your computer and select an existing directory or click **Make New Folder** to create a custom folder. Click **OK** to save the changes and exit, or **Cancel** to exit without saving.

It is not possible to add an application library identical to, containing, or being contained in, an already used application library.

Optionally, you can replace the standard folder icon ( ) with custom icons of your choice that reflect the content of your library folders. To use a custom icon for a folder, create a PNG-file with an image size of 16-by-16 pixels and save it in the folder under the name `folder.png`.

Set the COMSOL Application Libraries Root

Click the **Set COMSOL Application Libraries Root Directory** button ( ) to edit or set the root folder. This redirects the COMSOL software to a different folder where customized applications can be stored.

In the **Set COMSOL Application Libraries Root Directory** dialog box, navigate to the new root folder location or click **Make New Folder**. Click **OK** to save the changes and exit, or **Cancel** to exit without saving.

Remove Selected Library

This button is enabled after a user application library folder has been created. Select a user application library root folder in the **Application libraries** tree and then the click the **Remove Selected** ( ) button to remove the library from the tree.

The Applications Folders

In the **Applications** library folders you find runnable applications with custom user interfaces tailored with the Application Builder to simplify solving a specific problem using COMSOL Multiphysics. To run an application, click the ( ) **Run Application** button. If, instead, you want to explore how the application is constructed, click ( ) **Open**. For applications in this folder, clicking the ( ) **Open PDF document** button launches the PDF document that you can access from the running application. If no such document is available, this button is not activated. For further details, see the next section.

Running or Opening a Model or Application and Its Documentation

RUNNING AN APPLICATION

You can run an application built with the Application Builder by clicking **Run application**. This button is not activated for models.

OPENING A MODEL OR APPLICATION

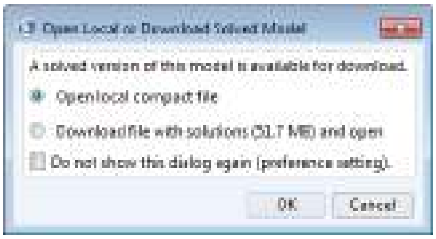
Once you have located the file you want to open — for example, you used a search and it is successful (see Searching the Application Libraries), or you browsed the **Application Libraries** tree — then to open the file:

- Double-click the name in the tree.
- Select the name, then click **Open**.
- Right-click the name, then from the context menu select **Open**.

> It is possible to open and postprocess models that include functionality that you have blocked or that your license does not include. Nodes with functionality that requires a license for a product that is blocked or not available get a **License Error** subnode ( ), where you find information about the missing but required product license. Unless you disable or remove such nodes, it is not possible to re-solve such models.

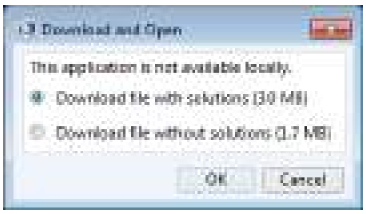
If the node represents a solved model (●) or a compact (○) or solved application, the file will open directly. If, instead, the file is a compact model MPH-file or a preview file (◌) and you are connected to the internet, you will be presented with the applicable download options:

- For a compact file, choose between opening the local model MPH-file and downloading the solved version before opening.



  If you select **Do not show this dialog again (preference setting)**, the local file will open directly the next time you open a compact model. If you change your mind later, you can control the behavior using the **Ask about download before opening compact application file check box** on the **Application Libraries** page of the **Preferences** dialog

- For a preview file, choose between downloading the file with or without solutions, that is between the solved and the compact MPH-file.



The approximate download sizes are displayed next to the options. For models, the numbers shown do not include the documentation, which is downloaded together with the MPH-file if you have chosen to install documentation.

If you have no internet connection or if the Application Library Update server is unavailable for some other reason, these dialogs will not appear and the associated context-menu options will not be shown. You can entirely disable availability checks on the **Preferences** dialog's **Application Libraries** page by clearing **Check Application Library Update availability on first access**. (If you already opened the **Application Libraries** window, this setting will take effect in the next session.)

OPENING A PDF DOCUMENT

To read the documentation in PDF format, including step-by-step instructions:

- Click to highlight the name in the tree, then click **Open PDF Document**.
- Right-click the name, then from the context menu select **Open PDF Document**.

You can enable the **Open PDF Document** button for a user-library model or application by placing a PDF-file with the same name as the MPH-file in the same folder.

PDF and HTML documentation is available also for preview nodes if online help is activated; see The Help Window and Topic-Based Help.

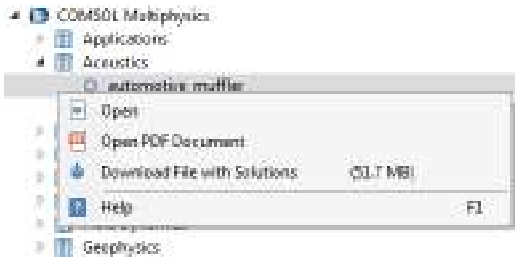
OPENING THE APPLICATION GALLERY

The Application Gallery, which is a part of the COMSOL website, provide access to a large number of models and applications, and you can download the MPH-files, PDF documentation, and other related files to extend the application libraries. Click the **Application Gallery** button ( ) above the application libraries tree to open the Application Gallery in a web browser.

Downloading MPH-Files With or Without Solutions

REPLACING COMPACT FILES

To replace compact MPH-files, you can download the files complete with solutions via **Application Library Update** (see The Application Library Update Window). Alternatively, right-click a compact node ( ) in the **Application Libraries** tree and choose **Download File with Solutions**.

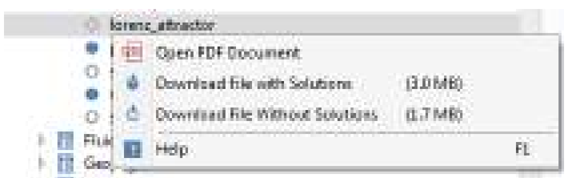


You can also generate the complete models by building the mesh sequences and computing the studies.

> The procedure for restoring the solutions can involve other steps, such as adjusting physics interface settings. See the individual documentation for details if the simple approach does not work.

REPLACING PREVIEW FILES

To replace a preview file, use **Application Library Update** or right-click the node in the **Application Libraries** tree and choose between **Download File with Solutions** and **Download File Without Solutions**.



Searching the Application Libraries

In the **Application Libraries** window, you can **Search** the application libraries to find any files using a specific feature. For example, enter all or part of the name, a physics interface name, a feature name, a feature tag or name prefixed by '@', or any other phrase or words or and click **Search**.

By default, the search includes all words in the **Search** field (with space signifying logical AND).

COMSOL files are named using an underscore between words (for example, `effective_diffusivity`) because the filename is also the name of the corresponding Model MPH-file. The underscore is required to form a valid filename, so it is recommended that you, if you are not sure of the full name, enter only the first word (rather than multiple words separated by spaces) in the **Search** field when searching for a model or application name.

The search in the **Application Libraries** window does not include the models' PDF documents. To search for text in the model documentation, use the COMSOL Documentation window and select **Application libraries** from the **Search scope** list to limit the search for the application libraries (see Searching the Documentation).

SEARCH PARAMETERS FOR MODELS AND APPLICATIONS

- To search for models and applications by filename only, use the prefix "@name:", for example `@name:busbar`. You can also use the wildcard character '*' at the beginning and the end of the search expression, for example `@name:fluid*` or `@name:*electr*`.
- To search for models and applications whose computation time (as displayed in the **Application Libraries** window) falls within a specific range, use the prefix "`@time:`". For example, the search expression `@time:>=1[h] <=2[h]` returns all models and applications with a computation time between 1 and 2 hours. The supported relational operators are `<` (the default if no operator is given), `>`, `<=`, `>=`, and `==`. Elapsed times can be expressed using the supported time units (see the section SI Base, Derived, and Other Units in The COMSOL Modeling Environment chapter for details). If no unit is specified, the expression entered is assumed to be given in seconds.
- If you enter more than one search term separated by spaces (in a search that is not restricted to filename using the "`@name`" prefix), the search finds files where all of the search terms appear.
- Limit the search to tags (identifiers) with the prefix "`@tag:`". For example, `@tag:genext` finds all files using a General Extrusion node, and `@tag:ehs` finds all files with an Electron Heat Source node.
- Limit the search to node labels (excluding trailing digits and tags enclosed in parentheses) with the prefix "`@label:`". To search for labels that include spaces, you need to enclose the label text within quotes. For example, `@label:"point evaluation"` finds files containing Derived Values nodes of Point Evaluation type with the default label base (as shown immediately below the window label in the **Settings** window).
- Limit the search to type names with the prefix "`@type:`". For example, `@type:segregated` finds all files using a segregated solver, `@type:slider` finds all applications containing a Slider form object, and `@type:bodyload` finds all files with a Body Load node.To find out the type names, consult the *COMSOL Programming Reference Manual* or, for application form objects, the *COMSOL Multiphysics Application Programming Guide*. Alternatively, you can save an application as a Model File for Java or Model File for MATLAB and look up the second argument for the `create(…)` method for an object of the type of node you want to search for.
- To search for a specific physics interface, use the scoping syntax `@physics:`*`id`*, where *`id`* is a default physics-interface identifier (forming the base of the tags shown in the **Model Builder** window if you have selected **Model Tree Node Text>Tag** in the window's toolbar). For example, enter `@physics:ec` to find all models that use the Electric Currents interface. Similarly, enter `@geom:if` to search for models that include an If statement (node) in the geometry sequence. Table 2-3 lists the supported model-object context scopes.
- The model-context search scopes `@cpl`, `@export`, `@func`, `@multiphysics`, `@numerical`, `@probe`, and `@selection` used without an identifier return all models that contain *any* node of the corresponding type; see Table 2-3.

- In addition to the model-object scopes, the following custom keywords are available:
  - `@keyword:tutorial` — finds introductory and tutorial models;
  - `@keyword:verification` — finds models classified as verification models or benchmarks;
  - `@keyword:industrial` — finds models classified as industrial applications.
- To find all models and applications that are locally available in solved or compact form, or only as previews, use the search strings `$solved`, `$compact`, and `$preview` respectively

TABLE 2-3: MODEL-OBJECT SEARCH SCOPES.

SCOPE	DESCRIPTION	EXAMPLES
`@cpl`	Search for component-coupling nodes.	`@cpl:genext`, `@cpl`
`@dataset`	Search for dataset nodes.	`@dataset:cpt`
`@export`	Search for export nodes.	`@export:anim`, `@export`
`@func`	Search for function nodes.	`@func:wv`, `@func`
`@geom`	Search for geometry-feature nodes.	`@geom:c`
`@mesh`	Search for mesh-feature nodes.	`@mesh:swe`
`@multiphysics`	Search for multiphysics-coupling nodes.	`@multiphysics:emh`, `@multiphysics`
`@numerical`	Search for derived-values nodes.	`@numerical:min`, `@numerical`
`@physics`	Search for physics-interface nodes.	`@physics:c`
`@probe`	Search for probe nodes.	`@probe:bnd`, `@probe`
`@result`	Search for plot-feature nodes.	`@result:slc`
`@selection`	Search for selection nodes.	`@selection:box`, `@selection`
`@sol`	Search for solver nodes.	`@sol:se`
`@study`	Search for study nodes.	`@study:param`

When a search result is presented, hover over a top-level node in the tree to see the number of matching files under the corresponding folder. If the search does not return any results, the **Application Libraries** window contains the message **No Results Found**. Click the **Refresh** button ( ) under the tree to return to the root **Application Libraries** folder list.

The Application Library Update Window

Application Library Update is a service that provides new and updated models and applications for the application libraries of the COMSOL products that your license includes. The text below describes how to use the Application Library Update service.

Using the COMSOL **Application Library Update** service requires internet access. For a default installation, you also need to run COMSOL as an administrator. See Proxy Server Settings section below for instructions on how you can modify your installation to avoid this restriction.

APPLICATION LIBRARY UPDATE

Open the **Application Library Update** window by clicking **Update COMSOL Application Libraries** in the **Application Libraries** window or by going to the **File>Help** menu (Windows users) or the **Help** menu (macOS and Linux users) and choosing **Update COMSOL Application Libraries** ( ).

When the **Application Library Update** window opens, choose if you want to download files containing solutions (**Solved**) or more compact versions of the models and applications (**Compact**), then click **Find Applications** to check if all the models and applications you have are up to date or if there are any updated or new ones. If the message **Your Application Libraries are up to date** displays, no updated or new models or applications are available.

If the library is not up to date, browse the list that appears with a description and image. Choose which ones to download by selecting or clearing the check boxes next to the thumbnail images. By default all check boxes are selected; by clicking **Clear selection** and **Select all** you can change the global selection state.

> Note that if you chose to download solved models and applications and then choose **Select all**, the download size can be on the order of 10s of GB if you have license for several products. Therefore, it may be wiser to choose the compact option and then download solved versions of the examples you are interested in directly from the **Application Libraries** window.

Click the **Download** button to download the selected models and applications. The download time depends on the size of the files, which is listed for each file, and the bandwidth of the internet connection.

> Also see the *Introduction to Application Builder*.

Proxy Server Settings

If you connect to the internet through a web proxy, you can specify the required settings in the **Help** section of the Preferences dialog box; for details, see Proxy Server Settings.

DESTINATION DIRECTORIES FOR LIBRARY UPDATES

To edit these settings under **Destination directories for library updates**, open The Preferences Dialog Box and go to the **Updates** section.

The **Destinations** list provides two options for specifying which application, documentation, and part directories are synchronized with the COMSOL Multiphysics server when you launch an Application Library Update or Part Library Update (see the next section) request:

- **Current directories** (default): Synchronize with application MPH-files under the COMSOL Application Libraries root set on the **Preferences** dialog's **Application Libraries** page, with documentation files under the directory specified in the **Documentation root directory** field on the **Preferences** dialog's **General** page, and with part MPH-files under the COMSOL Part Libraries root set on the **Preferences** dialog's **Part Libraries** page.
- **Specify custom directories**: Choosing this option lets you specify COMSOL Application Libraries, documentation, and COMSOL Part Libraries root directories separate from those of your current COMSOL Desktop environment.

By default, the COMSOL Application Libraries, COMSOL Part Libraries, and documentation root directories are located directly under the COMSOL installation root directory, in `applications/`, `parts/`, and `doc/`, respectively. This typically implies that special permissions are required for saving downloaded files, and it can therefore be beneficial to move or copy the directories to a different location. The settings referred to in this section are provided to let you customize Application Library Update to the IT environment of your organization.

The Part Library Update Window

Part Library Update is a service that provides new and updated geometry parts for the part libraries of the COMSOL products that your license includes. The text below describes how to use the Part Library Update service.

Using the COMSOL **Part Library Update** service requires internet access. For a default installation, you also need to run COMSOL as an administrator. If you connect to the internet through a proxy server, see the Proxy Server Settings section for the relevant settings.

PART LIBRARY UPDATE

Open the **Part Library Update** window by clicking **Update COMSOL Part Libraries** in the **Part Libraries** window or by going to the **File>Help** menu (Windows users) or the **Help** menu (macOS and Linux users) and choosing **Update COMSOL Part Libraries** ( ).

If the message **Your Part Libraries are up to date** displays when the **Part Library Update** window opens, no updated or new geometry parts are available. If the library is not up to date, browse the list that appears with a description and image. Choose which ones to download by selecting or clearing the check boxes next to the thumbnail images. By default all check boxes are selected; by clicking **Clear selected** and **Select all** you can change the global selection state.

Click the **Download** button to download the selected geometry parts. The download time depends on the size of the files, which is listed for each geometry part, and the bandwidth of the internet connection.

The Physics Interfaces

This section is an overview of the core physics interfaces included with a COMSOL Multiphysics license. If your license include add-on modules, there are additional physics interfaces described in the individual documentation for each module.

- Building Models in the Model Builder
- Adding and Inserting Physics Interfaces
- Modeling Guidelines
- Creating a New Model

Introduction to the Physics Interfaces

Solving PDEs generally means you must take the time to set up the underlying equations, material properties, and boundary conditions for a given problem. COMSOL Multiphysics, however, relieves you of much of this work. The software provides a number of *physics interfaces* that consist of nodes and settings that set up the equations and variables for specific areas of physics. An extensive set of physics-dependent variables makes it easy to visualize and evaluate the important physical quantities using conventional terminology and notation.

Suites of physics interfaces that are optimized for specific disciplines together with specialized application libraries are available in a group of optional products. See The COMSOL Multiphysics Modules and Interfacing Options.

A complement to the interfaces for physics, special interfaces for equation-based modeling simplify the setup of PDEs for modeling that does not explicitly refer to any particular application field. In addition, other interfaces supplement the physics with special functionality such as the Sensitivity and Moving Mesh user interfaces.

- Physics Groups and Subgroups
- Physics Interface Guide
- Selecting Physics Interfaces

Physics Groups and Subgroups

The Select Physics page in the **Model Wizard**, as well as The Add Physics Window, contain groups and subgroups of physics and mathematics interfaces (some items only display if a license includes the corresponding add-on modules).

RECENTLY USED ( )

This group contains the most recently used physics interfaces for easy access.

AC/DC ( )

This group contains physics interfaces for low-frequency electromagnetics such as electrostatics, electric currents and magnetic fields together with multiphysics couplings to heat transfer, structural mechanics and charged particle dynamics.

ACOUSTICS ( )

This group contains acoustics-based physics interfaces. Except for the Pressure Acoustics group, these subgroups require additional licenses:

- Elastic Waves
- Acoustic-Structure Interaction
- Aeroacoustics
- Thermoviscous Acoustics
- Ultrasound
- Geometrical Acoustics
- Pipe Acoustics

CHEMICAL SPECIES TRANSPORT ( )

This group contains chemical species transport physics interfaces used, for example, for convection and diffusion, solving for the species concentrations and for chemical reactions. These subgroups are available depending on the specific license:

- Moisture Transport
- Reacting Flow (including an additional Turbulent Flow subgroup)
- Reacting Flow in Porous Media
- Rotating Machinery, Rotating Flow

ELECTROCHEMISTRY ( )

This group contains electrochemistry physics interfaces for modeling electrochemical components such as batteries and fuel cells. This group and its subgroups are only available with additional licenses:

- Battery Interfaces
- Corrosion, Deformed Geometry
- Electrodeposition, Deformed Geometry

FLUID FLOW ( )

This group contains fluid flow physics interfaces such as laminar single-phase flow and, with add-on modules, multiphase flow and turbulent flow. These subgroups are available based on the license:

- Single-Phase Flow (including additional Turbulent Flow and Rotating Machinery, Fluid Flow subgroups)
- Thin-Film Flow
- Multiphase Flow (including additional Bubbly Flow; Mixture Model; Euler-Euler Model; Two-Phase Flow, Level Set; Two-Phase Flow, Phase Field; Two-Phase Flow, Moving Mesh; Three-Phase Flow, Phase Field; and Rotating Machinery, Multiphase Flow subgroups)
- Porous Media and Subsurface Flow
- Nonisothermal Flow (including additional Turbulent Flow and Rotating Machinery, Nonisothermal Flow subgroups).
- High Mach Number Flow (including an additional Turbulent Flow subgroup)
- Rarefied Flow
- Particle Tracing
- Fluid-Structure Interaction

HEAT TRANSFER ( )

This group contains physics interfaces for heat transfer in solids, fluids, pipes, and in porous media. Other physics interfaces are available for bioheat transfer and for heat and moisture transport. There are also multiphysics interfaces for thermal multiphysics applications such as Joule heating. These subgroups are available depending on the specific license:

- Heat and Moisture Transport
- Thin Structures (for heat transfer in thin shells, thin films, and fractures)
- Conjugate Heat Transfer (including an additional Turbulent Flow subgroup)
- Radiation
- Electromagnetic Heating
- Metal Processing

OPTICS ( )

This group contains physics interfaces for electromagnetic wave propagation in linear and nonlinear optical media for accurate component simulation and design optimization. This group and the subgroups are only available with additional licenses

- Ray Optics
- Wave Optics

PLASMA ( )

This group contains physics interfaces for plasma modeling. This group and the Equilibrium Discharges and Species Transport subgroups require the Plasma Module.

RADIO FREQUENCY ( )

This group contains physics interfaces for high-frequency electromagnetic field simulations solving the full Maxwell equations. This group is only available with the RF Module.

SEMICONDUCTOR ( )

This group contains physics interfaces that solves Poisson's equation for the electric potential and the drift-diffusion equations for electrons and holes in a semiconductor material: Semiconductor interfaces and interfaces for solving the Schrödinger and Schrödinger-Poisson equations.This group is only available with the Semiconductor Module.

STRUCTURAL MECHANICS ( )

This group contains structural mechanics physics interfaces for example to study displacements and stresses in solids and for multibody dynamics, fatigue, thermal stress, piezoelectricity, and other structural multiphysics couplings. Depending on licenses, the following subgroups can be available:

- Rotordynamics
- Thermal Stress
- Piezoresistivity

MATHEMATICS (Δu)

This group contains mathematics interfaces for solving PDEs, ODEs, and DAEs, for optimization (which requires the Optimization Module) and sensitivity analysis, and for modeling moving meshes and parameterized geometry. These subgroups are available:

- PDE Interfaces (including a Lower Dimensions subgroup)
- ODE and DAE Interfaces

- Optimization and Sensitivity
- Classical PDEs
- Moving Interface (available with either the CFD Module or Microfluidics Module)
- Deformed Mesh
- Wall Distance
- Mathematical Particle Tracing (available with the Particle Tracing Module)
- Curvilinear Coordinates

For a list of all the physics interfaces under each group and subgroup, see the individual documentation included with each add-on module. Or search the online Help for the key words *Physics Interface Guide*.

- Creating a New Model
- Physics Interface Guide
- Selecting Physics Interfaces

Adding and Inserting Physics Interfaces

THE ADD PHYSICS WINDOW

The **Add Physics** window is similar to the **Select Physics** window accessed through The Model Wizard. It has the same physics interfaces available. This window is a quick way to add physics interfaces to models.

To open the **Add Physics** window:

Win

- Right-click a **Component** node and choose **Add Physics**.
- From the **Home** toolbar, click **Add Physics**
- Select **Windows>Add Physics**.
- From the **Physics** toolbar, click **Add Physics** .

Mac

Linux

- Right-click a **Component** node and choose **Add Physics**.
- In the **Model Toolbar**, click **Add Physics** .
- In the **Physics Contextual Toolbar**, click **Add Physics** .
- Select **Windows>Add Physics**.

The **Add Physics** toolbar button is a toggle button: Click it again to close the **Add Physics** window.

TO ADD PHYSICS INTERFACES TO A MODEL COMPONENT

1. In the **Add Physics** window, either enter a **Search** term or navigate the tree to locate the physics interface to be added to the **Component**.

   The tree organizes the available physics interfaces by application areas such as fluid flow, heat transfer, and structural mechanics. The physics interfaces found in the modules your license supports display in the different application areas. In some cases, licensing of a module adds physics interfaces to these application areas as well as attributes to existing physics interfaces, which are enhanced with additional functionality. The **Recently Used** branch lists the last five physics interfaces used in recent modeling sessions. You can also enter a text string in the search field and click the **Search** button to list all the interfaces with the search term.

> The contents of the **Add Physics** window depends on the space dimension of the active model component. If there are no **Component** nodes in the model, the list of physics interfaces is empty.

2 Once a physics interface is clicked, review and optionally modify any dependent variable names in the **Dependent Variables** section and, for some physics interfaces, specify the number of dependent variables. For PDE interfaces under **Mathematics**, you can also click the **Units** tab to specify units for the equation by defining a unit through a physical quantity for the dependent variable and the source term. See Units for more information about these settings.

   If you select multiple physics interfaces, the **Dependent Variables** section is not available.

3 When there is already a physics interface added to the Component, the existing **Studies** are listed under **Physics interfaces in study**. By default, the studies appear with a check mark ( ) in the **Solve** column, which indicates that the study solves for the dependent variables in the physics interface. Click in the column to clear the check mark and exclude the physics interface from that study.

4 Click the **Add to Component** or **Add to Selection** buttons.

   - If you click **Add to Component**, the physics interfaces are added to the **Model Builder** and become active in the entire model component's geometry by default. You can also press Enter to add the physics components to the component.
   - If you click **Add to Selection**, the physics interfaces are added to the **Model Builder** with a selection taken from the selected geometric entities in the **Graphics** window. This is a method called *preselection*.

COPYING, PASTING, AND INSERTING PHYSICS INTERFACES

You can copy and paste physics interfaces within a COMSOL Multiphysics session and also between COMSOL Multiphysics sessions, as long as the copied physics interface information remains in the clipboard.

To copy a physics interface with all its subnodes, right-click the physics interface node and choose **Copy** (or, for Windows users, click **Copy** on the Quick Access Toolbar). You can then paste it in the same or a new COMSOL Desktop session by right-clicking a compatible **Component** node and choosing, for example, **Paste Solid Mechanics** (or, for Windows users, click, for example, **Paste Solid Mechanics** on the Quick Access Toolbar).

It is also possible to insert physics interfaces from another COMSOL Multiphysics model into a model component. To do so, right-click a **Component** node and choose **Insert Physics from Model**. An **Insert Physics** dialog box appears where you can browse or type the path and name of the COMSOL Multiphysics model file from which you want to insert components in the **Model** field. Select one or more of the physics interfaces in the model from the **Physics** list and then click **OK** to insert them into the current model. The list includes the space dimension for the physics interface so that you can select compatible physics interfaces.

> You can insert a physics from another space dimension if it is applicable in the component you insert it into. Most physics applicable in 2D axial symmetry is applicable in 3D, so an interface specified in a 2D axisymmetric component can therefore be inserted in a 3D component.

Some aspects when inserting or pasting a physics interface into an existing model component:

- All selections of a physics interface and its physics features depend on the geometry they belong to. When you insert or paste a physics interface into another component and geometry, there is a high risk of thats selections change or become empty.
- Any references to named selections will be broken, and no selection nodes are included automatically in the insertion process.
- All use of global parameters, functions, variables, materials, multiphysics couplings, and coordinate systems will be invalid after the insertion process unless an item with the same name exists in the target component. No attempt is made to automatically include any item in the insertion process. Materials should be of minor importance if the physics interfaces consistently use the option **From material** in all their material property settings.
- Some physics features are not applicable in all dimensions, so these will simply be ignored. You will be notified about this in the message dialog box.

Altogether, these aspects may cause an insert or paste operation to be incomplete to some degree. In some cases, the difference is reported in a message dialog box after the insertion process has finished. Click **Cancel** in this dialog box to revert the insertion process.

- Creating a New Model
- Physics Interface Guide

Physics Interface Guide

The table lists the physics and mathematics interfaces in COMSOL Multiphysics and the availability for 1D, 1D axisymmetric, 2D, 2D axisymmetric, and 3D geometries.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
AC/DC				
Electric Currents		ec	All dimensions	Stationary
Electrostatics		es	All dimensions	Stationary; Time Dependent
Magnetic Fields		mf	2D, 2D axisymmetric	Stationary; Frequency Domain
Acoustics				
Pressure Acoustics				
Pressure Acoustics, Frequency Domain		acpr	All dimensions	Eigenfrequency; Frequency Domain
Chemical Species Transport				
Transport of Diluted Species		tds	All dimensions	Stationary; Time Dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Fluid Flow				
Single-Phase Flow				
Laminar Flow		spf	3D, 2D, 2D axisymmetric	Stationary; Time Dependent
Heat Transfer				
Heat Transfer in Solids		ht	All dimensions	Stationary; Time Dependent
Heat Transfer in Fluids		ht	All dimensions	Stationary; Time Dependent
Electromagnetic Heating				
Joule Heating[1],		—	All dimensions	Stationary; Time Dependent
Structural Mechanics				
Solid Mechanics		solid	3D, 2D, 2D axisymmetric	Stationary; Eigenfrequency; Time Dependent
Mathematics				
Wall Distance		wd	All dimensions	Stationary; Time Dependent
Curvilinear Coordinates		cc	All dimensions	Stationary; Eigenvalue
PDE Interfaces				
Coefficient Form PDE		c	All dimensions	Stationary; Time Dependent; Eigenvalue
General Form PDE		g	All dimensions	Stationary; Time Dependent; Eigenvalue
Wave Form PDE		wahw	All dimensions	Time Dependent
Weak Form PDE		w	All dimensions	Stationary; Time Dependent; Eigenvalue
PDE, Boundary Elements		pdebe	3D and 2D	Stationary
Lower Dimensions				
Coefficient Form Boundary PDE		cb	All dimensions	Stationary; Time Dependent; Eigenvalue
Coefficient Form Edge PDE		ce	3D	Stationary; Time Dependent; Eigenvalue

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Coefficient Form Point PDE	Δu	cp	3D, 2D, 2D axisymmetric	Stationary; Time Dependent; Eigenvalue
General Form Boundary PDE	Δu	gb	All dimensions	Stationary; Time Dependent; Eigenvalue
General Form Edge PDE	Δu	ge	3D	Stationary; Time Dependent; Eigenvalue
General Form Point PDE	Δu	gp	3D, 2D, 2D axisymmetric	Stationary; Time Dependent; Eigenvalue
Weak Form Boundary PDE	$\int du$	wb	all dimensions	Stationary; Time Dependent; Eigenvalue
Weak Form Edge PDE	$\int du$	we	3D	Stationary; Time Dependent; Eigenvalue
Weak Form Point PDE	$\int du$	wp	3D, 2D, 2D axisymmetric	Stationary; Time Dependent; Eigenvalue
$\frac{d}{dt}$ **ODE and DAE Interfaces**				
Global ODEs and DAEs	$\frac{d}{dt}$	ge	All dimensions	Stationary; Eigenfrequency; Time Dependent; Frequency Domain; Eigenvalue
Domain ODEs and DAEs	$\frac{d}{dt}$	dode	All dimensions	Stationary; Time Dependent; Eigenvalue
Events		ev	All dimensions	Time Dependent
Boundary ODEs and DAEs	$\frac{d}{dt}$	bode	All dimensions	Stationary; Time dependent; Eigenvalue
Edge ODEs and DAEs	$\frac{d}{dt}$	eode	3D	Stationary; Time dependent; Eigenvalue
Point ODEs and DAEs	$\frac{d}{dt}$	pode	3D, 2D, 2D axisymmetric	Stationary; Time dependent; Eigenvalue
Optimization and Sensitivity				
Optimization Requires the Optimization Module		opt	All dimensions	Stationary; Eigenfrequency; Time Dependent; Frequency Domain; Eigenvalue
Sensitivity		sens	All dimensions	Stationary; Eigenfrequency; Frequency Domain; Eigenvalue; Time Dependent (available with the Optimization Module)
∇^2 **Classical PDEs**				
Laplace Equation	∇^2	lpeq	1D, 2D, 3D	Stationary
Poisson's Equation	∇^2	poeq	1D, 2D, 3D	Stationary
Wave Equation	∇^2	waeq	1D, 2D, 3D	Time Dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Helmholtz Equation		hzeq	1D, 2D, 3D	Stationary
Heat Equation		hteq	1D, 2D, 3D	Stationary; Time Dependent
Convection-Diffusion Equation		cdeq	1D, 2D, 3D	Stationary; Time Dependent
Stabilized Convection-Diffusion Equation		scdeq	1D, 2D, 3D	Stationary; Time Dependent
Deformed Mesh				
Deformed Geometry		dg	All dimensions	Stationary; Time Dependent; Frequency Domain; Eigenvalue
Moving Mesh		ale	All dimensions	Stationary; Time Dependent; Frequency Domain; Eigenvalue
[1] This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.				

Common Physics Interface and Feature Settings and Nodes

There are several common settings and sections available for the physics interfaces and feature nodes (Table 2-4). Some of these sections also have similar settings or are implemented in the same way no matter the physics interface or feature being used. There are also some physics feature nodes (Table 2-5) that display in COMSOL Multiphysics.

In each module's documentation, only unique or extra information is included; standard information and procedures are centralized in this manual.

> Table 2-4 has links to common sections and Table 2-5 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

SHOW MORE PHYSICS OPTIONS

To display additional sections and options for the physics interfaces (and other parts of the model tree), click the **Show More Options** button ( ) on the **Model Builder** and then select the applicable options in the **Show More Options** dialog box that opens.

After selecting some options, sections display on the **Settings** window when a node is clicked, or additional nodes are made available from the **Physics** toolbar or context menu. Selecting **Advanced Physics Options** either adds an **Advanced** settings section or enables nodes in the context menu or **Physics** toolbar. In many cases these options are described in the individual documentation.

For more information about the Show options, see Showing More Options and The Model Builder.

Common Physics Settings Sections

TABLE 2-4: COMMON PHYSICS SETTINGS SECTIONS

SECTION	CROSS REFERENCE AND NOTES
Advanced Settings — Pseudo time stepping	Pseudo Time Stepping and Pseudo Time Stepping for Laminar Flow Models
Advanced Settings — Frames	See Frames.
Advanced	This section can display after selecting Advanced Physics Options. The Advanced section is often unique to a physics interface or feature node.
Anisotropic materials	For some User defined parameters, the option to choose Isotropic, Diagonal, Symmetric, or Full displays. See Modeling Anisotropic Materials for information.
Consistent Stabilization	See Stabilization.
Constraint Settings	Constraint Reaction Terms, Weak Constraints, and Symmetric and Nonsymmetric Constraints
Coordinate System Selection	Coordinate Systems Selection of the coordinate system is standard in most cases. Extra information is included in the documentation as applicable.
Dependent Variables	Predefined and Built-In Variables This section is unique for each physics interface, although some interfaces have the same dependent variables.
Discretization	Settings for the Discretization Sections
Discretization — Frames	See Frames.
Equation	Physics Nodes — Equation Section The equation that displays is unique for each interface and feature node, but how to access it is centrally documented.
Frames (Advanced Settings — Frames and Discretization — Frames)	Handling Frames in Heat Transfer and About Frames
Geometric entity selections	Working with Geometric Entities Selection of geometric entities (Domains, Boundaries, Edges, and Points) is standard in most cases. Extra information is included in the documentation as applicable.
Inconsistent Stabilization	See Stabilization.
Settings	Predefined and Built-In Variables Displaying Node Names, Tags, and Types in the Model Builder There is a unique Name for each physics interface.
Material Type	About Using Materials in COMSOL Multiphysics The Settings Window for Material Selection of material type is standard in most cases. Extra information is included in the documentation as applicable.

TABLE 2-4: COMMON PHYSICS SETTINGS SECTIONS

SECTION	CROSS REFERENCE AND NOTES
Model Inputs	About Model Inputs and Model Inputs and Multiphysics Couplings Selection of Model Inputs is standard in most cases. Extra information is included in the documentation as applicable. To define the absolute pressure for heat transfer, see the settings for the Fluid node. To define the absolute pressure for a fluid flow interface, see the settings for the Fluid Properties node (described for the Laminar Flow interface). If you have a license for a nonisothermal flow interface, see that documentation for further information.
Override and Contribution	Physics Exclusive and Contributing Node Types Physics Node Status
Pair Selection	Identity and Contact Pairs Continuity on Interior Boundaries Selection of pairs is standard in most cases. Extra information is included in the documentation as applicable. Contact pair modeling requires the Structural Mechanics Module or MEMS Module. Details about this pair type can be found in the respective user's guide.
Stabilization — Consistent and Inconsistent	Numerical Stabilization, Numerical Stability — Stabilization Techniques for Fluid Flow and Heat Transfer Consistent and Inconsistent Stabilization Methods

COMMON FEATURE NODES

TABLE 2-5: COMMON FEATURE NODES

FEATURE NODE	CROSS REFERENCE AND NOTES
Auxiliary Dependent Variable	Auxiliary Dependent Variable
Axial Symmetry	See Symmetry.
Continuity	Continuity on Interior Boundaries and Identity and Contact Pairs. This is standard in many cases. When it is not, the node is documented for the physics interface.
Discretization	Discretization (Node)
Equation View	Equation View The Equation View node's contents is unique for each physics and mathematics interface and feature node, but it is centrally documented.
Excluded Edges, Excluded Points, and Excluded Surfaces	Excluded Points, Excluded Edges, Excluded Surfaces
Global Constraint	Global Constraint. Also see the Constraint Settings section.
Global Equations	Global Equations
Harmonic Perturbation	Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis
Initial Values	Physics Interface Default Nodes, Specifying Initial Values, and Dependent Variables This is unique for each physics interface.
Periodic Condition and Destination Selection	Periodic Condition and Destination Selection Periodic Boundary Conditions Periodic Condition is standard in many cases. When it is not, the node is documented for the physics interface.
Pointwise Constraint	Pointwise Constraint. Also see the Constraint Settings section.

TABLE 2-5: COMMON FEATURE NODES

FEATURE NODE	CROSS REFERENCE AND NOTES
Symmetry	Using Symmetries and Physics Interface Axial Symmetry Node. There is also information for the Solid Mechanics interface about Axial Symmetry. This is standard in many cases. When it is not, the node is documented for the physics interface.
Weak Constraint	Weak Constraint. Also see the Constraint Settings section.
Weak Contribution	Weak Contribution (ODEs and DAEs) and Weak Contribution (PDEs and Physics)
Weak Contribution on Mesh Boundaries	Weak Contribution on Mesh Boundaries

- Creating a New Model
- The Add Physics Window
- Showing More Options
- Selecting Physics Interfaces

Creating a New Model

This section describes how to create a new model using The Model Wizard or to begin with a blank model. First you need to Open a New Window to Begin Modeling. It is also useful to have a basic model added to the Model Builder; then you can experiment with The Model Builder, which is described in the Building a COMSOL Multiphysics Model chapter.

Open a New Window to Begin Modeling

To open a **New** window:

- On the **Quick Access Toolbar** (Windows users) or the **Model Toolbar** (macOS and Linux users), click the **New** button ( ).
- Press Ctrl+N.
- Select **File>New**.

After the **New** window opens to the **Model** page, select an option:

- Click the **Model Wizard** button ( ) to open the **Select Space Dimension** window. Go to The Model Wizard section to continue.
- Click the **Blank Model** button ( ) to open COMSOL Multiphysics without any model set up in the Model Builder or return to the default COMSOL Desktop. You can then add components and physics interfaces to the model.

To enable the **Physics Builder** choose **Preferences** ( ) from the **File** menu (Windows users) or from the **Options** menu (macOS and Linux users). Click **Physics Builder** and select the **Enable Physics Builder** check box.

After the applicable check box is selected, go to the **Model Wizard** and on the **New** page under **Physics**, click **Physics Builder** ( ). See the *Physics Builder Manual* for information.

The Model Wizard

The Model Wizard helps you build a model by choosing the space dimension, physics interfaces, and the study you want to use. In the Model Wizard you Select Space Dimension, Select Physics, Review Physics Interface, and finally Select Study.

SELECT SPACE DIMENSION

1 Open the Model Wizard (see Open a New Window to Begin Modeling).

2 On the **Select Space Dimension** page, click to choose the **Component** geometry dimension: **3D**, **2D axisymmetric**, **2D**, **1D axisymmetric**, **1D**, or **0D**.

Component Nodes by Space Dimension

The **Component** node has different icons based on space dimension **0D** ( ) (no space dimension), **1D** ( ), **1D axisymmetric** ( ), **2D** ( ), **2D axisymmetric** ( ), and **3D** ( ).

0D is used for interfaces modeling spatially homogeneous systems such as chemical reacting systems, electrical circuits, and general ODEs and DAEs. If you want to import a geometry, this is done in the Model Builder, but make sure you choose spatial dimensions that this geometry exists in. Remember, not all physics interfaces are available for all space dimensions.

Also add a **Component** node to the **Model Builder**:

- By right-clicking the **root** node ( ) and selecting it from the **Add Component** menu.
- In the **Home** toolbar, select an option from **Add Component** list.

SELECT PHYSICS

On the **Select Physics** window, there are different ways to select one or several physics interfaces to add to the model. There are also interfaces for PDEs, ODEs, and DAEs, and other mathematical interfaces for equation-based modeling under the **Mathematics** branch.

Once one or more physics interfaces are selected, there are several options to continue; see Figure 2-14. Click to select one of these buttons:

ICON AND NAME	ACTION
Add	Double-click to add the selected physics interface to the **Added physics interfaces** list, or right-click and select **Add physics** to add one or more selected physics interfaces. You can also add the selected physics by pressing Enter. Add as many physics interfaces as you want. You can use the **Review Physics Interface** page to edit the **Dependent Variables** as required. Click **Remove** as required to organize the physics interfaces in the list.
Space Dimension	Click to go back to the **Select Space Dimension** page.
Study	Click to choose the study for the model.
Done	Click to add the physics interface without a study.
Cancel	Click to return to the **New** page of the Model Wizard.
Help	Click to open the context-based **Help** window.

You can also add physics interfaces from the **Model Builder** and The Add Physics Window:

- In the **Home** toolbar, click the **Add Physics** button.
- By right-clicking the **Component** node and selecting **Add Physics**.

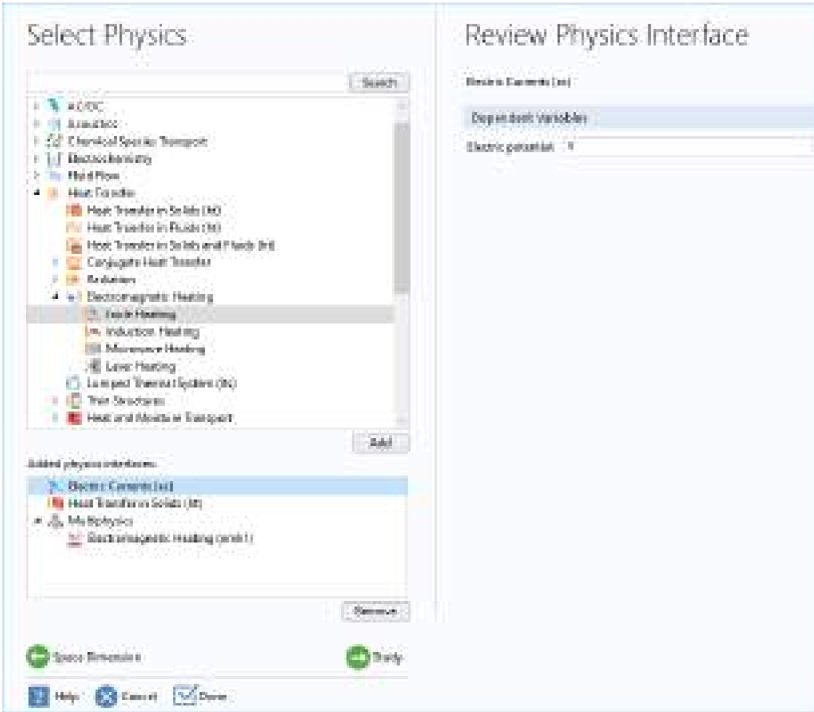


Figure 2-14: The Select Physics and Review Physics Interface windows in the Model Wizard.

REVIEW PHYSICS INTERFACE

Under **Added physics interfaces** (see Figure 2-14), click any interface to open the **Review Physics Interface** page. Here you can review and optionally modify any **Dependent Variables** names and, for some physics interfaces, specify the number of dependent variables. For other physics interfaces you can edit both the name of the field and the field components. Examples of fields with components are the displacement field in a Solid Mechanics interface and the Velocity field for a Laminar Flow interface.

To remove a physics interface already selected, highlight it in the list and click **Remove** under the table.

SELECT STUDY

On the **Select Study** page, click to select the type of study to perform. You can also double-click the study to select it and close the Model Wizard. The available options depend on the set of physics interfaces included in the model. Some study types are applicable to all physics interfaces for which you choose to solve, while others are not, but all are in some way available. The most applicable and common studies, such as **Stationary**, appear at the top of the list under **General Studies**. You can also select a study type from one of the following branches (see Figure 2-15):

- **General Studies** — General studies that are the most applicable for the selected physics interfaces.
- **Preset Studies for Selected Physics Interfaces** — Studies applicable to each physics interface that you have chosen to include. These preset studies appear under subnodes for each physics interface or under **Suggested by Some Physics Interfaces**.
- **Preset Studies for Selected Multiphysics** — Studies applicable to the multiphysics couplings that you have chosen to solve for.
- **More Studies** — Any fundamental study types (**Stationary**, **Time Dependent**, **Eigenfrequency**, **Eigenvalue** and **Frequency Domain**) that are not applicable to any of the physics interfaces being solved for. Also, this branch can contain some advanced composite study types.

- **Preset Studies for Some Physics Interfaces** — The study types recognized by some, but not all, of the physics interfaces being solved for.
- **Empty Study** — An empty study that will not perform any type of study unless you add some study steps to it.

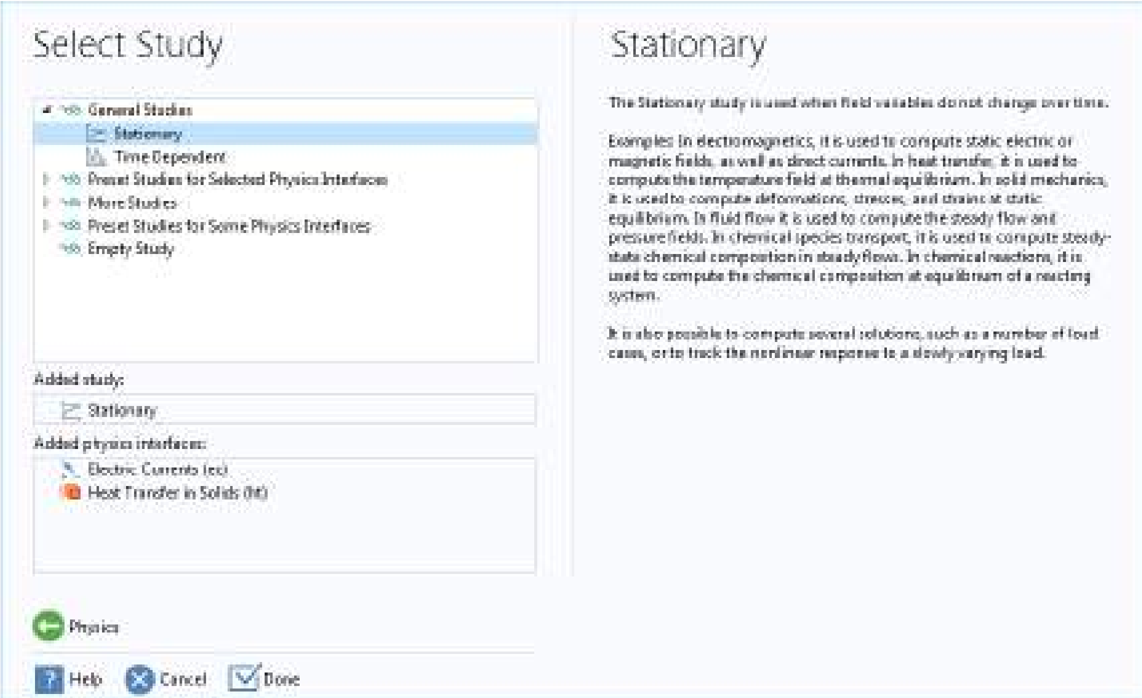


Figure 2-15: The Select Study page in the Model Wizard.

Once a study is highlighted, information about it displays to the right of the window and it is included under **Added study**. There are several options to continue.

ICON AND NAME	ACTION
Physics	Click to go back to the **Select Physics** and **Review Physics Interface** window.
Done	Click to exit the Model Wizard.
Cancel	Click to return to the **New** page of the Model Wizard.
Help	Click to open the context-based **Help** window.

Also add physics interfaces from the **Model Builder** and The Add Physics Window:

- In the **Home** toolbar, click the **Add Physics** button.
- By right-clicking the **Component** node and selecting **Add Physics**.

After clicking **Done**, the **Model Builder** window displays a model tree with a set of default nodes in the **Component** branches: **Definitions**, **Geometry**, **Materials**, **Mesh**, and nodes based on the physics interfaces selected (see Figure 2-16). The **Component** nodes and branches form the sequence of operations that define the model.

Figure 2-16: After clicking Done, a 3D Component with Electric Currents, Heat Transfer in Solids interfaces, and a Stationary study is added to the Model Builder.

Also add a study from the **Model Builder** and The Add Study Window:

- By right-clicking the **root** node and selecting **Add Study**.
- In the **Home** toolbar, click the **Add Study** button.

- Building a COMSOL Multiphysics Model
- The Component Node
- The Add Physics Window
- The Add Study Window
- The Model Builder

Toolbars and Keyboard Shortcuts

The toolbars and context menus in COMSOL Multiphysics are based on the stage of modeling. This section is a single resource for each of the ribbon and contextual toolbars available on the COMSOL Desktop. There are also several Keyboard Shortcuts that are useful for navigating during the modeling process. The following sections have a table where there are links to more information about the available items in the ribbon toolbar or contextual toolbar.

- Home Toolbar
- Definitions Toolbar
- Geometry Toolbar
- Sketch Toolbar
- Work Plane Modal Toolbar
- Materials Toolbar
- Physics Toolbar
- 0D Component Toolbar
- Mesh Toolbar
- Study Toolbar
- Results Toolbar
- Plot Group Contextual Toolbar
- Report Group Contextual Toolbar
- Template Group Contextual Toolbar
- View Toolbar
- Developer Toolbar

About Changes to the Ribbon Display (Windows Users)

Home Toolbar

The **Home** ribbon toolbar (Windows) and the **Model Toolbar** (macOS and Linux) contains many of the common features and actions required to build and analyze a model.

For step-by-step instructions and general documentation descriptions, this is the **Home** toolbar.

TABLE 2-6: THE HOME TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Application			
A	Application Builder (Ctrl+Shift+A)	Toggle between the Application Builder and Model Builder windows.	
Model			
	Component	Select any existing model component to move to that component. The name and icon indicates the current component and can therefore vary.	The Component Node
	Add Component	Add 3D, 2D, 2D Axisymmetric, 1D, 1D Axisymmetric, and 0D model components.	The Component Node

TABLE 2-6: THE HOME TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Definitions			
Pi	Parameters	Add globally available Parameters to your model.	Global Definitions, Geometry, Mesh, and Materials and Parameters
a=	Variables	Choose from Global Variables and Local Variables.	Global Definitions, Geometry, Mesh, and Materials, Predefined and Built-In Variables
f(x)	Functions	Choose from a list of all available Functions	Functions
Pi	Parameter Case	Add a Parameter Case subnode to a Parameters node.	Parameter Cases
Geometry			
	Build All	Build all features in the current geometry.	Editing and Building Geometry Nodes The Geometry Node
	Import	Import the geometry from a COMSOL Multiphysics file or CAD file.	Import
	LiveLink	Choose to connect to a CAD software using a LiveLink connection that is included in your license.	See the documentation for the CAD LiveLink products.
Materials			
	Add Material	Open the Add Material window to add materials to components or selections.	The Add Material Window
Physics			
—	Various — the physics interface name	When physics interfaces are added, these are listed here and you can click to take you to the node in the Model Builder.	Physics Interface Guide
	Select Physics Interface	For a blank model this is available.	Selecting Physics Interfaces
	Add Physics	Open the Add Physics window to add physics interfaces to the current component.	The Add Physics Window
Mesh			
	Build Mesh	Build the current mesh.	Adding, Editing, and Building Meshing Sequences
	Select Mesh[1]	Available for a blank model.	Mesh Elements for 1D, 2D, and 3D Geometries
	Mesh[1]	When meshes are added, these are listed here and you can click to take you to the node in the Model Builder.	Creating a Mesh for Analysis
Study			
	Compute[1]	Compute the selected study.	Computing a Solution

TABLE 2-6: THE HOME TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
	Select Study[1]	Available for a blank model.	Open a New Window to Begin Modeling and The Add Study Window
	Study[1]	When studies are added, these are listed here and you can click to take you to the node in the Model Builder.	Introduction to Solvers and Studies
	Add Study[1]	Open the Add Study window to add a study to the current Component.	The Add Study Window
Results			
	Select Plot Group[1]	Available for a blank model.	About Plot Groups
—	Various —the name of the plot group[1]	Once results are available or plot groups are added, these are listed here and you can click to take you to the node in the Model Builder. It also opens a new Plot Group contextual toolbar for the plot group.	Plot Groups and Plots
	Add Plot Group[1]	3D Plot Group, 2D Plot Group, 1D Plot Group, Polar Plot Group, or Smith Plot Group.	About Plot Groups
Layout			
	Reset Desktop[2]	Reset the COMSOL Desktop to its default settings and choose Widescreen Layout or Regular Screen Layout.	Customizing the Desktop Layout
	Windows	Choose any of the available windows that you can add to the COMSOL Desktop. The Windows menu contains the windows on the following rows in this table.	
	Add Physics[2]	Open the Add Physics window to add physics interfaces to the current component.	Creating a New Model
	Add Multiphysics[2]	Open the Add Multiphysics window to add multiphysics couplings to the current component.	The Add Multiphysics Window
	Add Study[2]	Open the Add Study window to add a study to the current Component.	The Model Wizard
	Add Material from Library[2]	Open the Add Material window to add materials to components or selections from the material libraries.	The Add Material Window
	Material Browser[2]	Open the Material Browser where you can access and edit material libraries.	The Material Browser Window
	Application Libraries[2]	Open the Application Libraries window.	The Application Libraries Window
	Part Libraries[2]	Open the Part Libraries window where you can access collections of geometry parts.	Part Libraries
	Selection List[2]	Choose objects, for example, while working with complex geometries and when you need to easily locate a geometric entity that is not easily viewed.	The Selection List Window
	Properties[2]	The Properties window is accessed from the context menu and displays other node properties.	Settings and Properties Windows for Feature Nodes

TABLE 2-6: THE HOME TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
	Messages[2]	Contains information useful after an operation is performed.	The Messages Window
	Debug Log	Displays debug information for debugging code in methods.	See the *Application Builder Reference Manual*.
	Table[2]	Displays the results from integral and variable evaluations defined in Derived Values nodes or by Probes and stored in Table nodes.	The Table Window and Tables Node
	External Process[2]	Follow external processes (such as distributed batch jobs) that have been started. The window updates when you are attached to the external processes.	The External Process Window
	Recovery Files[2]	Manage and open recovery files.	Saving and Opening Recovery Files
	Comparison Results[2]	Comparison of Physics Builder files.	See the *Physics Builder Manual*.
	Physics Builder Manager[2]	Manage testing, compilation, and comparison of your Physics Builder files.	See the *Physics Builder Manual*.

[1] For cross-platform users (macOS and Linux), the combination of buttons that display is dependent on the toolbar setting. For example, some buttons may not be visible if the Tools>Toolbar Display Mode is set to Normal and the Tools>Toolbar Button Label is set to Show Icon and Text. It also depends on whether you are using a widescreen or regular screen layout.

[2] For cross-platform users (macOS and Linux), these options are available from other menus or toolbars. See Cross Platform (macOS and Linux) Toolbars and Menus.

Definitions Toolbar

The **Definitions** ribbon toolbar (Windows) and the **Definitions** contextual toolbar (macOS and Linux) contain many of the common features and actions required to work with features found under the Definitions node in the Model Builder.

For step-by-step instructions and general documentation descriptions, this is the **Definitions** toolbar.

TABLE 2-7: THE DEFINITIONS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Variables			
a=	Local Variables	Add a Variables node under Definitions.	Global Definitions, Geometry, Mesh, and Materials, Predefined and Built-In Variables
	Variable Utilities	Add a variable utility such as a matrix variable under Definitions.	Variable Utilities
Functions			
	Analytic	Add an Analytic function node to define an analytic function.	Analytic

TABLE 2-7: THE DEFINITIONS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
	Interpolation	Add an Interpolation function node to define an interpolation function.	Interpolation
	Piecewise	Add a Piecewise function node to define a piecewise function.	Piecewise
f(x)	More Functions	All available Functions (except Analytic, Interpolation, and Piecewise).	Functions
Selections			
	Explicit	Add an Explicit node under Definitions.	Creating Named Selections and Explicit
	Complement	Add a Complement node under Definitions.	Union, Intersection, Difference, and Complement
	Adjacent	Add an Adjacent node under Definitions.	Adjacent
	Ball/Disk	Add a Ball (3D) or Disk node under Definitions.	Ball, Disk
	Box	Add a Box node under Definitions.	Box
	Cylinder	Add a Cylinder node under Definitions.	Cylinder
	Union	Add a Union node under Definitions.	Union, Intersection, Difference, and Complement
	Intersection	Add an Intersection node under Definitions.	
	Difference	Add a Difference node under Definitions.	
	Colors	Add, reset, or remove colors for selections.	Selection Colors
Probes			
	Update Probes	Update all probes.	Probes
	Probes	Select an option from the list to add a node under Definitions Table 5-18.	
Physics Variables			
	Mass Properties	Add a Mass Properties node to the current model component.	Mass Properties
	Mass Contributions	Add a Mass Contributions node to the current model component (under **More Physics Variables**).	Mass Contributions
	Participation Factors	Add a Participation Factors node to the current model component (under **More Physics Variables**).	Participation Factors
	Response Spectrum	Add a Response Spectrum node to the current model component (under **More Physics Variables**).	Response Spectrum

TABLE 2-7: THE DEFINITIONS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Coupling			
	Nonlocal Couplings	Select an option from the list to add a node under Definitions.	Nonlocal Couplings and Coupling Operators
	Pairs	Select an option from the list to add a node under Definitions.	About Identity and Contact Pairs
Coordinate Systems			
	Coordinate Systems[1]	Select an option from the list to add a node under Definitions Table 5-17.	Coordinate Systems
	Perfectly Matched Layer[1]	Add a Perfectly Matched Layers node under Definitions.	Perfectly Matched Layer
	Infinite Element Domain[1]	Add an Infinite Element Domain node under Definitions.	Infinite Element Domain
	Absorbing Layer[1]	Add an Absorbing Layer node under Definitions.	Absorbing Layer
Deformed Mesh			
	Moving Mesh	Add moving mesh nodes under Definitions.	Moving Mesh Features
	Deformed Geometry	Add deformed geometry nodes under Definitions.	Deformed Geometry Features
Optimization			
	Optimization	Add optimization nodes.	See the documentation for the Optimization Module.
View			
	View[1]	Add a View node to the current Component to control the view and lighting in the Graphics window. 3D and 2D.	View (1D and 2D), View (3D), Axis (2D and 2D Axisymmetric) and Axis (1D and 1D Axisymmetric)

[1] For cross-platform users (macOS and Linux), the combination of buttons that display is dependent on the toolbar setting. For example, some buttons may not be visible if the Tools>Toolbar Display Mode is set to Normal and the Tools>Toolbar Button Label is set to Show Icon and Text. It also depends on whether you are using a Widescreen or Regular Screen Layout.

Geometry Toolbar

Once a geometry is added to the model, the **Geometry** ribbon toolbar (Windows) and the **Geometry** contextual toolbar (macOS and Linux) contains many of the common features and actions required to create and build a geometry.

For cross-platform users, some options listed in Table 2-8 are available from other toolbars and menus. Table 2-9 lists the geometry drawing tools available on the toolbars in 1D and 2D, as well as on the Work Plane toolbar for 3D models. See Cross Platform (macOS and Linux) Toolbars and Menus.

For step-by-step instructions and general documentation descriptions, this is the **Geometry** toolbar.

TABLE 2-8: THE GEOMETRY TOOLBARS

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS
Build		
	Build All[2]	Build all features in the current geometry. See Editing and Building Geometry Nodes.
Import/Export		
	Import[2]	Import the geometry from a COMSOL Multiphysics file or CAD file.
	Insert Sequence	Insert a geometry sequence from a COMSOL Multiphysics file into the current geometry. See Inserting a Sequence.
	Export	Export the current geometry as a COMSOL binary or text file or to a CAD file format. See Exporting a Geometry.
Cleanup		
	Defeaturing and Repair[1,3]	Choose from Cap Faces, Delete Fillets, Delete Short Edges, Delete Sliver Faces, Delete Small Faces, Delete Spikes, Delete Faces, Detach Faces, Knit to Solid, and Repair.
	Remove Details	Remove small details from 3D geometries. See Remove Details.
	Virtual Operations[1]	Virtual geometry operations and mesh control operations. See Virtual Geometry Operations and Virtual Geometry and Mesh Control Operations.
Sketch (2D)		
	Sketch	Toggle the sketch visualization on or off. See The Sketch Visualization.
Primitives (3D)		
	Block	Add a Block to the current geometry.
	Cone	Add a Cone to the current geometry.
	Cylinder	Add a Cylinder to the current geometry.
	Sphere	Add a Sphere to the current geometry.
	Torus	Add a Torus to the current geometry.
	Helix	Add a Helix to the current geometry.
	More Primitives	Add all other 3D primitives as in Table 7-3 and Geometric Primitives. Also see the geometry parts available in the Part Libraries.
Primitives (2D)		
	Circle	Add a Circle to the current geometry.
	Ellipse	Add an Ellipse to the current geometry.
	Rectangle	Add a Rectangle to the current geometry.
	Square	Add a Square to the current geometry.
	Polygon	Add a Polygon to the current geometry.

TABLE 2-8: THE GEOMETRY TOOLBARS

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS
	Point	Add a Point to the current geometry.
	More Primitives	Add all other 2D primitives as in Table 7-3 and Geometric Primitives. Also see The 1D Geometry Toolbar — Drawing Tools (Table 2-9) and the geometry parts available in the Part Libraries.
Primitives (1D)		
	Interval	Draw an interval (line) in the Graphics window. For 1D models, first click the Interval button, then click the starting point and endpoint in the Graphics window. Also see Geometric Primitives and The 1D Geometry Toolbar — Drawing Tools (Table 2-9).
	Point	Add a point to the Graphics window. Use this to draw a single point. First click the Point button, then click in the Graphics window (in 1D and 2D), or specify the point location in its Settings window. In 2D and 3D, this button is available on the More Primitives menu.
Work Plane		
	Select Work Plane	Once Work Planes are available or added, these are listed here and you can click to take you to the associated Plane Geometry node in the Model Builder. It also opens a new Work Plane contextual toolbar for the Work Plane. See Using Work Planes, Work Plane Modal Toolbar and Table 2-9.
	Work Plane	Add a Work Plane to the current geometry. See Using Work Planes, Work Plane Modal Toolbar and Table 2-9.
Operations		
	Extrude	Extrude planar faces of geometry objects or objects from a work plane to create 3D geometry objects.
	Revolve	Revolve planar faces of geometry objects or objects from a work plane about an axis to create 3D geometry objects.
	Sweep	Sweep faces along a spine curve to create a solid object.
	Loft	Requires the Design Module. Lofting planar sections along a path.
	Booleans and Partitions	Create a geometry object using Boolean operations: Union, Intersection, Difference, and Compose. Partition geometry objects into parts using other geometry objects as tool objects for the partition. Alternatively, in 3D, use a work plane to partition the geometry objects. Select Partition Objects or Partition Edges.
	Transforms	This menu has these operations available: Array, Copy, Mirror, Move, Rotate[1], Scale[1]
	Conversions[1]	See Table 7-8 for a list of these features. Also see Conversion Operations.
	Chamfer[1]	Chamfer corners in 2D geometry objects. Also in 3D with the Design Module.
	Fillet[1]	Fillet corners in 2D geometry objects. Also in 3D with the Design Module.
	Tangent	Add a line segment that is tangent to a given edge in 2D geometries.
	Delete[1]	Delete the selected geometry objects. See Clearing Sequences and Deleting Sequences or Nodes.

TABLE 2-8: THE GEOMETRY TOOLBARS

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS
	Edit Object	Adjust edges and vertices for 2D geometry objects or add or delete edges and vertices in the object.
	Cross Section	Create 2D cross sections from 3D geometries.
Other		
	Parts	3D, 2D, and 1D Parts, Load Part, and Part Instances. See Part Libraries, Using Geometry Parts, and Part Instance.
	Programming[1]	If + End If and Parameter Check. From the Add Before and Add After menus: If, Else If, Else, and End If. See If, Else If, Else, End If, and Parameter Check.
	Selections[1]	Create named selections of geometry objects or geometric entities in geometry objects. See Creating Named Selections in the Geometry Sequence.
	Colors[1]	Add coloring to highlight named selections in the Graphics window. See Selection Colors.
	Measure[1]	Measure the volume, area, perimeter, or other geometric properties of the selected geometric entities or objects. See Measuring Geometry Objects.
	Delete Sequence[1]	Delete a geometry sequence. See Clearing Sequences and Deleting Sequences or Nodes.

[1] For cross-platform users (macOS and Linux), the combination of buttons that display is dependent on the toolbar setting. For example, some buttons may not be visible if the Tools>Toolbar Display Mode is set to Normal and the Tools>Toolbar Button Label is set to Show Icon and Text. It also depends on whether you are using a Widescreen or Regular Screen Layout. It also depends if the button is available on the Work Plane toolbar, in which case it may be visible.

[2] For cross-platform users, this option is available from a different toolbar or menu. See Cross Platform (macOS and Linux) Toolbars and Menus.

[3] These features are available for the LiveLink and CAD products, and you must use the CAD kernel for the geometry representation.

GEOMETRY DRAWING TOOLBAR BUTTONS

In 1D, there are buttons for drawing Geometric Primitives by using the mouse. In 2D, the Sketch Toolbar provides tools for drawing 2D geometric primitives. In 3D, the buttons are available to create primitives, but you cannot draw these using the mouse unless you are using a Work Plane.

TABLE 2-9: THE 1D GEOMETRY TOOLBAR — DRAWING TOOLS

BUTTON OR MENU	NAME	DESCRIPTION OR ACTION
Draw Settings		
	Snap to Grid	Snap to the grid when drawing a geometry object in the Graphics window. By default, the mouse pointer snaps to the grid points. The active grid point for the snapping is indicated with a red circle. To disable snapping to the grid, click the Snap to Grid button.
	Snap to Geometry	Snap to the vertices of other geometry objects when drawing a geometry object in the Graphics window. By default, the mouse pointer snaps to the geometry vertices. The active point for the snapping is indicated with a red circle. To disable snapping to the geometry objects, click the Snap to Geometry button.

TABLE 2-9: THE 1D GEOMETRY TOOLBAR — DRAWING TOOLS

BUTTON OR MENU	NAME	DESCRIPTION OR ACTION
Draw		
	Interval	Draw an Interval (line) in the Graphics window. For 1D models, first click the Interval button, then click the starting point and endpoint in the Graphics window.
	Point	Add a Point to the Graphics window. Use this to draw a single point. First click the Point button, then click in the Graphics window (in 1D and 2D), or specify the point location in its Settings window. In 2D and 3D, this button is available on the More Primitives menu.

Sketch Toolbar

Use the Sketch toolbar to draw in the graphics window for a 2D geometry or part, or in a work plane for a 3D geometry. If you have the Design Module, the Sketch toolbar also contains buttons for creating geometric constraints and dimensions (see the Design Module documentation for more information)..

TABLE 2-10: THE SKETCH TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR ACTION
Sketch		
	Sketch	Toggle the sketch visualization on or off. See The Sketch Visualization.
Draw Settings		
	Snap to Grid	Snap vertices and edges to the grid when drawing or dragging a vertex in the Graphics window. The snapped cursor position is indicated with a blue circle.
	Snap to Geometry	Snap vertices and edges to vertices and edges when drawing or dragging a vertex in the Graphics window. The vertex or edge snapped to is highlighted in turquoise. There can also be a green line or circle segment between the snapped entities. If you have the Design Module and you have enabled **Use constraints and dimensions** in the Geometry node's settings: When snapping occurs, a constraint feature will be generated in some cases, so that the snapping will persist when you modify your geometry.
	Solid	Create a solid object (instead of a curve object) when drawing a closed curve in the Graphics window.
Draw		
	Polygon	Draw a sequence of line segments in the Graphics window. Click the points of the polygon. Then right-click anywhere to finish, or click another curve drawing button to create a Composite Curve.
	Circular Arc	Draw a sequence of circular arcs in the Graphics window in one of the following ways: • Click the **Circular Arc** button or choose the menu item **Start, Center, Angle**. Then click the starting point, the center, and the direction of the end angle. • Choose the **Start, Tangent, End** menu item. Then click the starting point, the direction of the tangent line at the starting point, and the endpoint. • Choose the **Start, Tangent, End** menu item. Then click the starting point, the direction of the tangent line at the starting point, and the endpoint.

TABLE 2-10: THE SKETCH TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR ACTION
	Interpolation Curve	Draw a cubic spline curve that interpolates given points in the Graphics window, in one of the following ways: • Click the **Interpolation Curve** button or choose the **Interpolation Points** menu item. Then click a sequence of points to interpolate. • Choose the **Start, Tangent, Other** menu item. Then click the starting point, the direction of the tangent line at the starting point, and the other interpolation points. Finally, right-click anywhere to finish, or click another curve drawing button to create a Composite Curve.
	Quadratic	Draw a sequence of quadratic Bézier curve segments in the Graphics window. Click the three control points to create one Bézier segment. A Quadratic Bézier node is then added. Optionally, click an even number of additional control points to create several Bézier segments. Finally, right-click anywhere to finish, or click another curve drawing button to create a Composite Curve that contains **Quadratic Bézier** nodes corresponding to each Bézier segment.
	Cubic	Draw a sequence of cubic Bézier curve segments in the Graphics window. Click the four control points to create one Bézier segment. A Cubic Bézier node is then added. Optionally, click additional control points to create several Bézier segments. Finally, right-click anywhere to finish, or click another curve drawing button to create a Composite Curve that contains **Cubic Bézier** nodes corresponding to each Bézier segment.
	Point	Draw a point in the Graphics window. After clicking the button, click the point in the Graphics window.
	Rectangle	Draw a rectangle or square in the Graphics window in one of the following ways: • Click the Rectangle button or choose the **Rectangle** menu item. Then click two opposite corners of the rectangle. • Choose the Square menu item. Then click one corner. Move the mouse to adjust the size of the square. Click to finish. • Choose the **Rectangle (Center)** menu item. Then click the center and one corner. • Choose the **Square (Center)** menu item. Then click the center. Move the mouse to adjust the size of the square. Click to finish.
	Circle	Draw a circle or ellipse in the Graphics window in one of the following ways: • Click the Circle button or choose the **Circle** menu item. Then click the center. Move the mouse to adjust the size of the circle. Click to finish. • Choose the Ellipse menu item. Then click the center. Move the mouse to adjust the axes of the ellipse. Click to finish. • Choose the **Circle (Corner)** menu item. Then click one corner of the circumscribed square of the circle. Move the mouse to adjust the size of the circumscribed square. Click to finish. • Choose the **Ellipse (Corner)** menu item. Then click one corner of the circumscribed rectangle of the ellipse. Move the mouse to adjust the size of the circumscribed rectangle. Click to finish.

TABLE 2-10: THE SKETCH TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR ACTION
Operations		
	Composite Curves	Create a Composite Curve feature that has a sequence of child features of type Polygon, Circular Arc, Interpolation Curve, Quadratic Bézier, and Cubic Bézier. First select at least one edge from each curve object to include in the Composite Curve in the Graphics window. These objects must be of one of the following types: Polygon, Circular Arc, Interpolation Curve, Quadratic Bézier, Cubic Bézier, or Composite Curve. The curve objects must form a single connected chain. Then click the **Compose Curves** button. The original curve features will be replaced with a **Composite Curve** feature or a **Polygon** feature in the Model Builder.
	Delete	Delete the selected vertices and edges. Select some vertices and edges in the Graphics window. Then click the **Delete** button or press the Del key. This will delete the selected entities using a combination of the following methods: • Removing geometry features from the Model Builder • Removing points from Polygon features • Splitting Polygon and Composite Curve features into several features • Adding Delete Entities features This operation does not delete selected vertices that are adjacent to an edge that is not selected. If you have the Design Module, you can also select symbols for constraint and dimension features and then delete them by clicking **Delete** or pressing Del.

Work Plane Modal Toolbar

The **Work Plane** modal toolbar is available after clicking Plane Geometry under **Geometry>Work Plane** in the Model Builder.

For step-by-step instructions and general documentation descriptions, this is the **Work Plane** modal toolbar.

The Work Plane toolbar is similar to the 2D Geometry Toolbar except that the **Virtual Operations** menu is not available and there is a **Close** button.

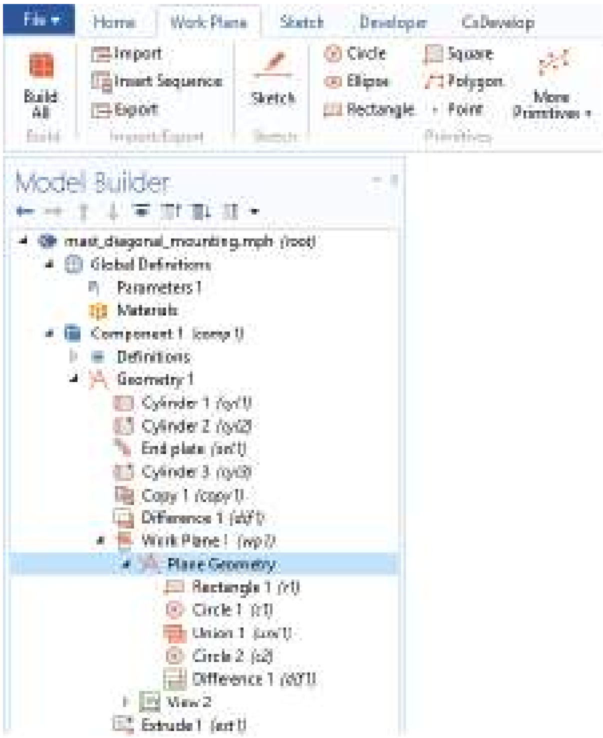


Figure 2-17: Click the Plane Geometry node to open the Work Plane toolbar.

Materials Toolbar

Once physics interfaces are added to the model, the **Materials** ribbon toolbar (Windows) and the **Materials** contextual toolbar (macOS and Linux) contains many of the common features and actions required to work with the features found under the **Materials** node in the Model Builder.

For step-by-step instructions and general documentation descriptions, this is the **Materials** toolbar.

TABLE 2-11: THE MATERIALS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Materials			
	Add Material	Open the Add Material window to add materials to components or selections.	The Add Material Window
	Blank Material	Add a blank material to your model.	Materials and The Settings Window for Material
	Browse Materials	Open the Material Browser where you can access and edit material libraries.	The Material Browser Window
	More Materials	Add a Material Link, a Material Switch, or an External Material.	Material Link, Switch for Materials, and Working with External Materials

TABLE 2-11: THE MATERIALS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Property Groups			
	Analytic	Add an Analytic function node to define an analytic function.	Analytic and Using Functions in Materials
	Interpolation	Add an Interpolation function node to define an interpolation function.	Interpolation and Using Functions in Materials
	Piecewise	Add a Piecewise function node to define a piecewise function.	Piecewise and Using Functions in Materials
	User-Defined Property Group	Add a user-defined property group to current material.	Property Groups
User-Defined Libraries			
	Add to Library	Add a material to a user-defined material library.	User-Defined Materials and Libraries

Physics Toolbar

Once physics interfaces are added to the model, the **Physics** ribbon toolbar (Windows) and the **Physics** contextual toolbar (macOS and Linux) contains many of the common features and actions required to add physics interfaces and features to the Model Builder.

For interfaces available in 0D, see 0D Component Toolbar.

For step-by-step instructions and general documentation descriptions, this is the **Physics** toolbar.

TABLE 2-12: THE PHYSICS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Physics			
—	Various - the physics interface name	Any physics interfaces added to the selected Component are listed. Click to go to the node in the Model Builder and open the Settings window. See Physics Interface Guide.	The Physics Interfaces
	Add Physics	Open the Add Physics window to add physics interfaces to the current component.	The Add Physics Window
	Add Multiphysics	Open the Add Multiphysics window to add applicable multiphysics couplings to the current component.	The Add Multiphysics Window
	Insert Physics from Model	Open the Insert Physics dialog box to insert physics interfaces in the current model component from another model file.	Adding and Inserting Physics Interfaces

TABLE 2-12: THE PHYSICS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Geometric Entity			
	Domains	See Table 3-3 for a list of all the icons by space dimension. Available physics features for the physics interface are listed. To add subnodes, however, you need to right-click the parent node. For example, to add a Destination Selection subnode, right-click Periodic Condition.	About Geometric Entities About Selecting Geometric Entities
	Boundaries		
	Pairs		
	Edges		
	Points		
	Global		
Contextual			
—	Attributes	Subnodes that can be added to a main (parent) node. 0D, 1D, 2D, and 3D	For example, Destination Selection (a subnode to Periodic Condition) or Damping (a subnode to Linear Elastic Material). When available, Harmonic Perturbation is added from this menu as an exclusive node. See Harmonic Perturbation — Exclusive and Contributing Nodes
	Load Group	After adding a Load Group to the Global Definitions branch you can activate it in one or more load cases.	Using Load Cases
	Constraint Group	After adding a Constraint Group to the Global Definitions branch you can activate it in one or more load cases.	
	Harmonic Perturbation	Click this to add Harmonic Perturbation as a contributing node.	Harmonic Perturbation — Exclusive and Contributing Nodes
Multiphysics			
	Multiphysics Couplings	This menu contains any physics features that provide multiphysics couplings likely to be used for the set of physics interfaces added to the Model Builder.	Multiphysics Modeling Workflow and The Multiphysics Branch

0D Component Toolbar

Once a 0D component interface is added to the model, a toolbar with the same name as the physics interface displays in the ribbon (Windows) and the contextual toolbar (macOS and Linux).

For step-by-step instructions and general documentation descriptions, the name of the toolbar is the same as the physics interface it is attached to.

These toolbars are documented for the interfaces in the applicable module documentation.

AVAILABLE WITH A COMSOL MULTIPHYSICS LICENSE:

- Global ODEs and DAEs. See The Global ODEs and DAEs Interface.
- Events. See The Events Interface.
- Sensitivity. See The Sensitivity Interface.

AVAILABLE WITH THE ADDITION OF VARIOUS LICENSES:

- Reaction Engineering
- Chemistry
- Optimization

Mesh Toolbar

Once a mesh is added to the model, the **Mesh** ribbon toolbar (Windows) and the **Mesh** contextual toolbar (macOS and Linux) contains many of the common features and actions required to work with meshes.

TABLE 2-13: THE MESH TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Build			
	Build Mesh[2]	Build the current mesh.	Adding, Editing, and Building Meshing Sequences
	Mesh (1, 2, 3,...)	Lists the meshes available in the model. Click a Mesh button to go to the node in the Model Builder.	Creating a Mesh for Analysis
	Add Mesh	Add a new mesh to the current model component.	Adding, Editing, and Building Meshing Sequences
Physics Controlled			
	Edit	Edit the physics-controlled sequence.	Physics-Controlled Mesh
	Reset	Reset the sequence to the physics-controlled settings.	
	Mesh Size (Normal is the default)	See Table 8-1 for a list of options.	The Mesh Toolbar (Predefined Mesh Element Sizes) Mesh Element Quality and Size
Generators			
	Free Tetrahedral	Generate unstructured tetrahedral mesh for 3D components.	Free Tetrahedral
	Swept	Generate swept mesh for 3D components.	Swept
	Boundary	The following are available from this menu: Free Triangular, Free Quad, Mapped, and Edge.	
	Boundary Layers	Generate boundary layer mesh.	Boundary Layers
Operations			
	Modify	Size: Distribution and Corner Refinement Elements: Convert and Refine Mesh: Reference and Scale	Meshing Operations and Attributes and Mesh Attributes

TABLE 2-13: THE MESH TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
	Copy	Copy Domain, Copy Edge, Copy Face, and Copy.	Meshing Operations and Attributes
Attributes			
	Mesh Size (Normal is the default)	See Table 8-1 for a list of options.	The Mesh Toolbar (Predefined Mesh Element Sizes) Mesh Element Quality and Size
	Distribution	Add a Distribution node under the selected node to specify an element distribution.	Mesh Attributes
	More Attributes	See Mesh Attributes.	
Import/Export			
	Import	Import mesh.	Importing Meshes
	Partition Entities	Create geometric entities by partitioning the mesh. Ball, Box, Cylinder, Logical Expression, and Detect Faces.	Using Operations on an Imported Mesh
	Create Entities	Create mesh vertices, edges, faces, and domains, and fill holes in imported meshes.	Using Operations on an Imported Mesh
	Delete Entities	Delete geometric entities.	Using Operations on an Imported Mesh
	Join Entities	Join adjacent geometric entities.	Using Operations on an Imported Mesh
	Export	Export mesh.	Exporting Meshes
	Create Part	Create a Mesh Part node, under which you import a mesh and prepare it for use in a component.	Using Mesh Parts
	Create Geometry from Mesh	Create a new model component with a geometry created from the mesh.	Creating Geometry from Mesh
Evaluate			
	Measure	Measure the volume, area, perimeter, or other geometric properties of the selected geometry objects.	Measuring Geometry Objects
	Statistics	Write information about number and quality of elements to the Messages window.	The Mesh Statistics Window
	Plot	Add a mesh plot.	Mesh (Plot)
Clear			
	Clear Mesh[1]	Clear the mesh.	Clearing Sequences and Deleting Sequences or Nodes
	Clear All Meshes[1]	Clear all meshes in the model.	

TABLE 2-13: THE MESH TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
	Delete Sequence[1]	Delete a meshing sequence.	Clearing Sequences and Deleting Sequences or Nodes

[1] For cross-platform users (macOS and Linux), the combination of buttons that display is dependent on the toolbar setting. For example, some buttons may not be visible if the Tools>Toolbar Display Mode is set to Normal and the Tools>Toolbar Button Label is set to Show Icon and Text. It also depends on whether you are using a Widescreen or Regular Screen Layout.

[2] For cross-platform users, this option is available from a different toolbar or menu. See Cross Platform (macOS and Linux) Toolbars and Menus.

Study Toolbar

The **Study** ribbon toolbar (Windows) and the **Study** contextual toolbar (macOS and Linux) contains many of the common features and actions required to work with studies and solvers.

For step-by-step instructions and general documentation descriptions, this is the **Study** toolbar.

TABLE 2-14: THE STUDY TOOLBAR

BUTTON OR MENU	NAME	OPTIONS DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Study			
	Compute[1]	Compute the selected study.	Computing a Solution
	Select Study[1]	See a list of all available studies in Table 20-2.	Study and Study Step Types
	Add Study[1]	Open the Add Study window to add a study to the current model component.	The Add Study Window
	Continue	Continue the computation of a solver sequence from the last computed feature.	The Progress Window
	Update Solution	Update the solution using the current values of parameters and user-defined variables.	Updating a Solution
	Get Initial Value	Generate a solution using the initial values of the dependent variables (without solving for the dependent variables).	Computing the Initial Values
Solver			
	Show Default Solver	Display the nodes in the solver sequences that are created by default.	Show Default Solver
Study Step			
	Study Steps	See lists of all available study types and study steps in Table 20-2 and Table 20-3.	The Add Study Window and The Relationship Between Study Steps and Solver Configurations
	Parametric Sweep	Choose Parametric Sweep or Optimization, for example.	Study Extension Steps and Advanced Study Extension Steps
f(x)	Function Sweep	Run a function sweep, switching between different user-defined functions.	Function Sweep

TABLE 2-14: THE STUDY TOOLBAR

BUTTON OR MENU	NAME	OPTIONS DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
	Material Sweep	Run a material sweep, switching between different materials.	Material Sweep
	Combine Solutions	Combine solutions using concatenation or summation, or by removing solutions.	Combine Solutions
	Study Reference	Add a study step that refers to another study.	Study Reference
	Optimization	Perform an optimization study. This is available with the addition of the Optimization Module.	See the *Optimization Module User's Guide*.
	Parameter Estimation	Perform a parameter estimation study. This is available with the addition of the Optimization Module.	See the *Optimization Module User's Guide*.
	Sensitivity	Add a Sensitivity study step.	Sensitivity
	Model Reduction	Add a Model Reduction study step.	Model Reduction
Batch & Cluster			
	Batch	Add a Batch or Batch Sweep study node	Batch and Batch Sweep
	Cluster	Add a Cluster Computing or Cluster Sweep study node	Cluster Computing and Cluster Sweep
Operations			
	Create Solution Copy	Make a copy of a solution.	Solution Operation Nodes and Solvers
Evaluate			
	Statistics	Display statistics for the study, including the number of degrees of freedom (DOFs).	The Statistics Page
Clear			
	Clear Solutions	Clear the solutions in the current solver sequence.	Clearing Sequences and Deleting Sequences or Nodes
	Clear All Solutions	Clear all solutions.	

[1] For cross-platform users, this option is available from a different toolbar or menu. See Cross Platform (macOS and Linux) Toolbars and Menus.

Results Toolbar

The **Results** ribbon toolbar (Windows) and the **Results** contextual toolbar (macOS and Linux) contains many of the common features and actions required to work with studies and solvers.

For step-by-step instructions and general documentation descriptions, this is the **Results** toolbar.

TABLE 2-15: THE RESULTS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Plot Group			
	Plot	Plot the selected plot group.	Plot Groups and Plots. Also see Plot Group Contextual Toolbar.
—	Select Plot Group	Choose from a list of plots included in the model.	About Plot Groups. Also see Plot Group Contextual Toolbar.
	3D Plot Group	Create a new 3D Plot Group for 3D plots such as volume and slice plots.	
	2D Plot Group	Create a new 2D Plot Group for 2D plots such as surface and contour plots.	
	1D Plot Group	Create a new 1D Plot Group for 1D graph plots.	
	Polar Plot Group	Create a new Polar Plot Group for graph plots in a polar coordinate system.	
Definitions			
	Parameters	Add a Parameters node under Results for defining results parameters.	Parameters
Dataset			
	Cut Plane	Create a Cut Plane dataset for data on a plane in 3D.	Cut Plane
	Cut Line 3D	Create a 3D Cut Line dataset for data along a line in 3D.	Cut Line 2D and Cut Line 3D
	Cut Point 3D	Create a 3D Cut Point dataset for data at a point in 3D.	Cut Point 1D, Cut Point 2D, and Cut Point 3D
	Cut Line 2D	Create a 2D Cut Line dataset for data along a line in 2D.	Cut Line 2D and Cut Line 3D
	Cut Point 2D	Create a 2D Cut Point dataset for data at a point in 2D.	Cut Point 1D, Cut Point 2D, and Cut Point 3D
	More Datasets	See Table 21-7 for a list of all datasets.	Datasets
	Selection (under **Attributes**)	Add a selection to the current dataset.	Adding a Selection to a Dataset and Creating Named Selections
	Remesh Deformed Configuration (under **Attributes**)	Create a deformed geometry from a current dataset.	Deformed Configuration
Numerical			
	Evaluate All	Evaluate all derived values and evaluation groups.	Table Window toolbar and Menu Options
	Clear and Evaluate All	Clear all current table entries and then evaluate all derived values and evaluation groups.	Table Window toolbar and Menu Options

TABLE 2-15: THE RESULTS TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
	Evaluation Group	Add an Evaluation Group node to evaluate derived values and present them in an Evaluation Group table.	About Evaluation Groups
	Point Evaluation	Add a Point Evaluation node to evaluate some expressions or variables at points.	Point Evaluation
	Global Evaluation	Add a Global Evaluation node to evaluate some global expressions or variables.	Global Evaluation
	More Derived Values	See Table 21-9 for a list of all derived value types.	Derived Values, Evaluation Groups, and Tables
	Table	Add a table.	Derived Values, Evaluation Groups, and Tables
Export			
	Data[1]	Data, Plot, Mesh (Export), Table	Exporting Data and Images
	Image[1]	Image	
	Animation	Animation or Player (see Animation) Create a movie to animate a solution (as an animated GIF, Flash, AVI, or WebM movie file, an image sequence, or as a player directly in the COMSOL Desktop Graphics window).	
Report			
	Report[1]	Choose a Brief Report, Intermediate Report, Complete Report, Custom Report, Documentation, or create or choose a custom report template.	Creating, Exporting, and Using Custom Report Templates
Clear			
	Clear Plot Data[1]	Clear all plot data that is stored in the model.	Saving Plot Data in the Model

[1] For cross-platform users (macOS and Linux), the combination of buttons that display is dependent on the toolbar setting. For example, some buttons may not be visible if the Tools>Toolbar Display Mode is set to Normal and the Tools>Toolbar Button Label is set to Show Icon and Text. It also depends on whether you are using a Widescreen or Regular Screen Layout.

Plot Group Contextual Toolbar

The plot group contextual toolbar is available after clicking a specific plot group in the Model Builder. The available tools are based on the model and the type of plot.

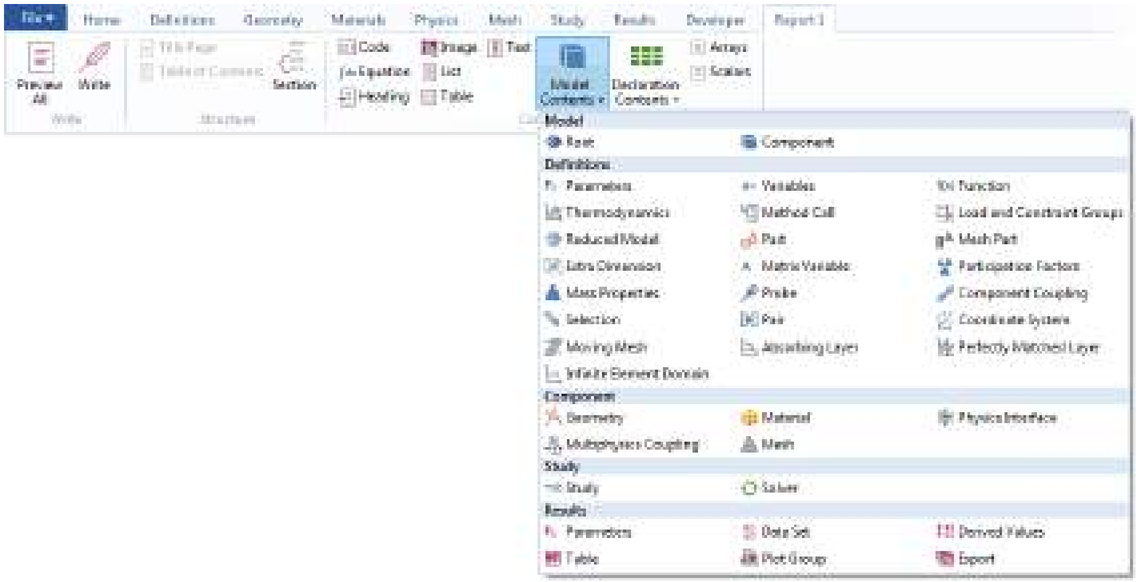
For step-by-step instructions and general documentation descriptions, this is generally referred to as the **Plot Group** toolbar, where the name of the toolbar changes based on the plot group (3D Plot Group, 2D Plot Group, 1D Plot Group, or Polar Plot Group). If the plot group is renamed, the toolbar name also changes to match the new name, as in the example above where the **3D Plot Group** toolbar was renamed **Stress (solid)**.

TABLE 2-16: THE PLOT GROUP CONTEXTUAL TOOLBAR

BUTTON OR MENU	NAME	LINK TO MORE INFORMATION
Plot		
	Plot	Plot Groups and Plots
	Plot In New Window	The Plot Windows
Add Plot		
Various	See Table 21-10 for the available plots by Plot Group (3D, 2D, 1D, or Polar Plot Groups).	About Plot Groups
Attributes		
	Add a Color Expression to the currently selected plot.	Color Expression
	Add a Deformation to the currently selected plot.	Deformation
	Add a Filter to the currently selected plot.	Filter
	Add a Height Expression to the currently selected plot. This is for 2D plots.	Height Expression
Select (available for creating cross-section plots)		
Various	See Table 21-11 for a list of the available buttons.	Creating Cross-Section Plots and Combining Plots and Plotting and Cross-Section Interactive Toolbar
Export		
Various	See Table 21-12 for a list of export options.	Exporting Data and Images

Report Group Contextual Toolbar

The report group contextual toolbar is available after clicking a **Report** node in the Model Builder. It contains tools for creating a model report.



For step-by-step instructions and general documentation descriptions, this is generally referred to as the **Report Group** toolbar, where the name of the toolbar name changes based on the report group. If the report group is renamed, the toolbar name also changes to match the new name.

TABLE 2-17: THE REPORT GROUP CONTEXTUAL TOOLBAR

BUTTON OR MENU	NAME	LINK TO MORE INFORMATION
Write		
	Generate	
	Preview All	The Report Node
	Write	
Structure		
	Title Page (available for custom reports)	The Title Page
	Table of Contents (available for custom reports)	The Table of Contents
	Section	Sections in the Report
Contents		
	Equation	Custom Report Components
	Heading	Custom Report Components
	Image	Custom Report Components

TABLE 2-17: THE REPORT GROUP CONTEXTUAL TOOLBAR

BUTTON OR MENU	NAME	LINK TO MORE INFORMATION
	List	Custom Report Components
	Table	Custom Report Components
	Text	Custom Report Components
	Model Contents	Model Contents — Report Components
	Declaration Contents	Declaration Contents
	Arrays	Arrays and Scalars
	Scalars	Arrays and Scalars

Template Group Contextual Toolbar

The template group contextual toolbar is available after clicking a **Template** node or its subnodes under **Reports** in the Model Builder. It includes tool for creating report templates.

TABLE 2-18: THE TEMPLATE GROUP CONTEXTUAL TOOLBAR

BUTTON OR MENU	NAME	LINK TO MORE INFORMATION
Export		
	Generate	
	Export	The Template Node
	Export All	
Structure		
	Title Page	The Title Page
	Table of Contents	The Table of Contents
	Section	Sections in the Report
Contents		
	Model Contents	Model Contents — Report Components

View Toolbar

Once physics interfaces are added to the model, the **View** ribbon toolbar (Windows) and the **View** contextual toolbar (macOS and Linux) contains many of the common features and actions required to work with the features found under the Materials node in the Model Builder.

	For step-by-step instructions and general documentation descriptions, this is the **View** toolbar.

TABLE 2-19: THE VIEW TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Reset View			
	Reset to Default	Restore the default settings for the View node.	User-Defined Views
Lights (3D only)			
	Directional Light	Add a Directional Light to the View.	About the 3D View Light Sources and Attributes
	Point Light	Add a Point Light to the View.	
	Spotlight	Add a Spotlight to the View.	
	Headlight	Add a Headlight to the View.	
Hide			
	Hide	Add a selection feature to hide geometry objects or entities.	Hide for Geometry, Hide for Physics, Hide for Mesh Import

Developer Toolbar

For creating and running methods, adding method calls and settings forms to the Model Builder, and testing applications, the **Developer** ribbon toolbar is available in the Windows version. Some of the functionality is connected to the Application Builder and the creation and testing of applications, but you can also create model methods and method calls and run them for custom extensions of a model created in the Model Builder and also add custom settings form for custom settings to control the workflow in the Model Builder.

TABLE 2-20: THE DEVELOPER TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Application A			
A	Application Builder (Ctrl+Shift+A)	Toggle between the Application Builder and Model Builder windows.	
	Data Access	Add data and properties that can be modified from a running application.	Data Access in the *Application Builder Reference Manual.*
	Test Application (Ctrl+F8)	Launch the application in a separate window.	Testing the Application in the *Application Builder Reference Manual.*
Create Methods			
	New Method	Create a new method with code to run in the model.	Creating and Running Methods in Models. Also see Creating Methods in the *Application Builder Reference Manual*.
	Record Method	Record changes to the embedded model to a new application method or model method.	Recording Code in the *Application Builder Reference Manual.*

TABLE 2-20: THE DEVELOPER TOOLBAR

BUTTON OR MENU	NAME	DESCRIPTION OR OPTIONS	LINK TO MORE INFORMATION
Method Calls			
	Method Call	Add any available model method as a method call in the Model Builder.	Method Calls
Run Methods			
	Run Method	Run one of the available methods.	Creating and Running Methods in Models
	Run Method Call	Run one of the added method calls.	
	Stop	Stop a running method.	
Forms			
	Settings Form	Add a form as a Settings window in the Model Builder.	Creating and Using Settings Forms and Dialogs
	Update Forms	Update forms in the Model Builder	
	Show Dialog	Show the form as a dialog box.	
Add-ins			
	Add-ins	Add an add-in as a Settings window in the Model Builder.	Creating Add-ins
	Add-in Libraries	Open the Add-In Libraries window.	
	Refresh Add-ins	Refresh the available add-ins in the Model Builder.	
	Clear Add-ins	Clear all imported add-ins in the Model Builder.	
Compare			
	Compare	Compare the current model and application with another MPH-file on the file system and then open the Comparison Result window.	Comparing Models and Applications

Keyboard Shortcuts

The following table summarizes the available keyboard shortcuts on Windows and Linux and on macOS (see also Keyboard Shortcuts for the Quick Access Toolbar and the Application Builder documentation for specific keyboard shortcuts):

SHORTCUT (WINDOWS, LINUX)	SHORTCUT (MACOS	ACTION
F1	F1	Display help for the selected node or window.
Ctrl+F1	Command+F1	Open the COMSOL Documentation front page in an external Help window.
F2	F2	Rename the selected node, file, or folder.
F3	F3	Disable selected nodes.
F4	F4	Enable selected nodes.

SHORTCUT (WINDOWS, LINUX)	SHORTCUT (MACOS	ACTION
F5	F5	Update solution with respect to new definitions without re-solving the model. Update reduced model data. Also, on Windows, to continue in the Method Editor's debugging tool in the Application Builder.
F6	F6	Build the preceding node in the Geometry branch or plot the previous plot for a time-dependent, eigenfrequency, or eigenvalue solution. Also, on Windows, to step in the Method Editor's debugging tool in the Application Builder,
F7	F7	Build the selected node in the geometry and mesh branches, compute the selected study step, compute to the selected node in the solver sequence, or plot the next plot for a time-dependent, eigenfrequency, or eigenvalue solution. Also, on Windows, to step into in the Method Editor's debugging tool in the Application Builder.
F8	F8	Build the geometry, build the mesh, compute entire solver sequence, update results data, update the plot, or run model method in a method call. Also, on Windows, to create an executable or an add-in in the Application Builder.
F9		On Windows, check syntax for a method in the Application Builder.
Del	Del	Delete selected nodes and selected geometry objects.
Left arrow (Windows); Shift + left arrow (Linux)	Left arrow	If you are at the top of a branch, pressing this collapses this branch in the Model Builder. Continue pressing the left arrow key to move upward in the tree to collapse all branches. If within a subbranch, pressing the left arrow key repositions you to the beginning of the branch.
Right arrow (Windows); Shift + right arrow (Linux)	Right arrow	Expand a branch in the Model Builder.
Up arrow	Up arrow	Move to the node above in the Model Builder. Highlight the next entity for a 3D geometry in the Graphics window when it has focus.
Down arrow	Down arrow	Move to the node below in the Model Builder. Highlight the previous entity for a 3D geometry in the Graphics window when it has focus.
Alt+left arrow	Ctrl+left arrow	Move to the previously selected node in the Model Builder.
Alt+right arrow	Ctrl+right arrow	Move to the next selected node in the Model Builder.
Ctrl+A	Command+A	Select all domains, boundaries, edges, or points; select all cells in a table or all contents of a table cell.
Ctrl+C	Command+C	Copy text in fields. Copy images in graphics and plot windows.
Ctrl+D	Command+D	Clear the selection of all domains, boundaries, edges, or points in the Model Builder. Clear all selections in form editor windows.
Ctrl+Shift+D	Command+Shift+D	Duplicate the selected node in the Model Builder.
Ctrl+F	Command+F	Find a search string in a model or application method.

SHORTCUT (WINDOWS, LINUX)	SHORTCUT (MACOS	ACTION
Ctrl+G	Command+G	Group nodes.
Ctrl+Shift+G	Command+Shift+G	Ungroup nodes.
Ctrl+K		Create, use, or rename a shortcut to an Model Builder node (for use in the Application Builder).
Ctrl+L	Command+L	Take a quick image snapshot.
Ctrl+N	Command+N	Create a new model.
Ctrl+O	Command+O	Open a model file.
Ctrl+P	Command+P	Print the contents of the plot window.
Ctrl+S	Command+S	Save a model file.
Ctrl+V	Command+V	Paste copied text.
Ctrl+Y	Ctrl+Shift+Z	Redo the last undone operation.
Ctrl+Z	Command+Z	Undo the last operation.
Ctrl+1		Create a local variable or correct the type of an existing variable in methods.
Ctrl+7		Toggle comments on and off in methods.
Ctrl+up arrow	Command+up arrow	Move a definitions node, geometry node, physics interface or feature node (except default nodes), material node, mesh node, study step node, or results node up one step.
Ctrl+down arrow	Command+down arrow	Move a definitions node, geometry node, physics interface or feature node (except default nodes), material node, mesh node, study step node, or results node down one step.
Ctrl+Tab	Ctrl+Tab	Switch focus to the next window on the desktop.
Ctrl+Shift+Tab	Ctrl+Shift+Tab	Switch focus to the previous window on the desktop.
Ctrl+Shift+A		Switch to the Application Builder window from the Model Builder.
Ctrl+Shift+M		Switch to the Model Builder window from the Application Builder.
Ctrl+F4	Command+W	Close the active window in COMSOL Desktop, if it is closable.
Ctrl+F8		Test an application.
Ctrl+Alt+left arrow	Command+Alt+left arrow	Switch focus to the Model Builder window.
Ctrl+Alt+right arrow	Command+Alt+right arrow	Switch focus to the Settings window.
Ctrl+Alt+up arrow	Command+Alt+up arrow	Switch focus to the previous section in the Settings window.
Ctrl+Alt+down arrow	Command+Alt+down arrow	Switch focus to the next section in the Settings window.
Shift+F10 or (Windows only) Menu key	Ctrl+F10	Open the context menu.
Ctrl+Pause	Command+. (Command + period)	Stop running a method when test running applications.
Ctrl+Space, Ctrl+/	Ctrl+Space, Ctrl+/	Open list of predefined quantities for insertion in text fields and table cells for expressions. Ctrl+/ is an alternative primarily intended for users of Asian Windows versions.

SHORTCUT (WINDOWS, LINUX)	SHORTCUT (MACOS	ACTION
+, -	+, -	Highlight the next or previous entity for a 3D geometry in the Graphics window. Expand or collapse a branch in the Model Builder.
R	R	Toggle between automatic and manual rotation center for mouse rotation in 3D.
X, Y, Z	X, Y, Z	Press to force rotation around the *x*, *y*, or *z* axis, respectively, when you move the mouse in 3D.

3

Building a COMSOL Multiphysics Model

This chapter explains a range of methods and topics used when building models in COMSOL Multiphysics®: From working with the Model Builder and fundamental concepts for building a model to the use of units. For examples of how to build a complete model and application step by step, see the application libraries for COMSOL Multiphysics and the add-on modules.

In this chapter:

- Building Models in the Model Builder
- Modeling Development Tools
- Working with Nodes in the Model Builder
- Modeling Guidelines
- Multiphysics Modeling Workflow
- Specifying Model Equation Settings
- Boundary Conditions
- Computing Accurate Fluxes
- Using Load Cases

- Numerical Stabilization
- Using Units

Building Models in the Model Builder

The power of COMSOL Multiphysics is the ease of working with all the features and functionality required to build a model in The Model Builder. The sections About the Sequence of Operations, The Component Node, Branches and Subbranches in the Tree Structure, Settings and Properties Windows for Feature Nodes, and Opening Context Menus and Adding Nodes further introduce you to key concepts about navigating in the Model Builder, the structure of the tree, and how to add features (nodes) as you build your model.

The physics feature nodes that are added to physics interfaces are flexible and several sections describe the ways to identify changes, status updates, and other ways to work with these nodes: The Physics Nodes, Physics Interface Default Nodes, Physics Feature Nodes by Space Dimension, Physics Interface Node Context Menu Layout, Physics Exclusive and Contributing Node Types, Physics Node Status, Dynamic Nodes in the Model Builder, and Errors and Warnings.

- The Root Settings and Properties Windows
- Creating a New Model
- The COMSOL Desktop

The Model Builder

The modeling procedure is controlled through the **Model Builder** window, which is essentially a *model tree* with all the functionality and operations for building and solving models and displaying the results. These are introduced to your modeling procedure by adding a *branch*, such as the Geometry branch. Branches can have further *nodes* (or *subbranches*) that relate to their parent node. It is all About the Sequence of Operations. See Figure 3-2 for an example.

A node has its own properties and *Settings* window that are characteristic to it. Branches and subbranches can also contain properties and settings. See Branches and Subbranches in the Tree Structure and Settings and Properties Windows for Feature Nodes for examples.

The Model Builder has many types of nodes to help you create models and visualize the model structure — for example, the Component node is categorized by space dimension, and nodes are dynamic, which helps you identify nodes that change status. See Component Nodes by Space Dimension, Physics Interface Default Nodes, and Dynamic Nodes in the Model Builder for more information.

Also learn about the context menu available when you right-click a node in the Model Builder (Opening Context Menus and Adding Nodes). In the next section (Working with Nodes in the Model Builder), there is also information about Going to the Source Node, Copying, Pasting, and Duplicating Nodes, Undoing and Redoing Operations, Clearing Sequences and Deleting Sequences or Nodes, and Disabling or Enabling Nodes.

- The Root Settings and Properties Windows
- Creating a New Model
- Basic Navigation
- The COMSOL Desktop

About the Sequence of Operations

COMSOL Multiphysics operates through *sequencing* and evaluates most of the branch nodes in the Model Builder from the top down as a *sequence of operations*. By adding nodes to a branch in the Model Builder in a certain order, you set up such sequences of operations, which makes it possible to, for example, parameterize a model and rerun the simulation. COMSOL Multiphysics then reevaluates each sequence, automatically updating the geometry, mesh, physics interfaces and features, and solution. A solver sequence, for example, could define your model with one solver and then, using the returned solution, solve it with an alternative solver.

For most sequences, you run the sequence by right-clicking the top node of the branch and selecting **Build All** (geometry) and **Build** (mesh), **Compute** (studies), or **Plot** (plot groups), or by pressing F8. These buttons are also on the **Settings** window and in the respective toolbars.

Some nodes under a physics interface branch can override other nodes higher up in the sequence. How the COMSOL Multiphysics software treats those nodes depends on whether they are contributing or exclusive nodes (see Physics Exclusive and Contributing Node Types).

The sequence of operations means that the order of the nodes in the tree is important. In the following branches of the model tree, the node order makes a difference, and you can move nodes up and down to change the sequence of operations for these nodes: Geometry, Material, physics interfaces and features, Mesh, and Solver.

Also, the order can have some importance in the plot groups in the Results branch and also for the Perfectly Matched Layer and Infinite Element Domain nodes in the Definitions branch (those nodes are available with some of the add-on modules).

- Physics Node Status
- Physics Exclusive and Contributing Node Types
- Creating a Geometry for Analysis and Working with Geometry Sequences
- Moving Nodes in the Model Builder

The Global Definitions Node

Under the **Global Definitions** node ( ) you can add functionality that is global and applies to the entire mode. Right-click the **Global Definitions** node to add global parameters, variables and variable utilities, user-defined functions, load and constraint groups, global materials, default model inputs, and more. See Global Definitions.

The Component Node

A model component is a fundamental part of the model and contains a geometry with its associated physics interface, mesh, and variables and other definitions that are local to that component. The **Component** node defines the namespace for each part of a model that is defined in a model component. A model can have several **Component** nodes. For example, if you are setting up a system model using both a 2D simplification — represented in one 2D **Component** branch — and a full 3D description in another **Component**, these can both be added to the Model Builder to represent different aspects or parts of the model. You can couple variables between different components in a model using coupling operators.

To **Add Physics** and **Add Mesh** to the Component, from the **Home** toolbar, or for any operating system, right-click the **Component** node. See The Add Physics Window, and Creating a Mesh for Analysis for more information.

The **Component** node icon also indicates the space dimension:.

TABLE 3-1: SPACE DIMENSION ICONS IN THE MODEL BUILDER

ICON	SPACE DIMENSION
	3D
	2D axisymmetric
	2D
	1D axisymmetric
	1D
	0D (space-independent models for chemical reactions and other ODEs and DAEs)

ADDING A COMPONENT TO A MODEL

You can create models with multiple geometries by adding one or more **Component** nodes to the Model Builder. Typically a component is added to the model in the Model Wizard when you select a space dimension.

To add a **Component** node or nodes:

- Right-click the **root** node (the topmost node) in the **Model Builder** and select **Add Component** (see The Root Settings and Properties Windows).
- In The Model Wizard on the **Select Space Dimension** page, select **3D**, **2D axisymmetric**, **2D**, **1D axisymmetric**, or **1D**. Continue defining the model as in Creating a New Model.

COPYING, PASTING, AND INSERTING COMPONENTS

You can copy and paste model components within a COMSOL Multiphysics session and also between COMSOL Multiphysics sessions, as long as the copied component information remains in the clipboard.

To copy a component with all its subnodes, right-click the **Component** node and choose **Copy** (or, for Windows users, click **Copy** on the Quick Access Toolbar). You can then paste it in the same or a new COMSOL Desktop session by right-clicking the root node and choosing **Paste Multiple Items** (or, for Windows users, click **Paste Multiple Items** on the Quick Access Toolbar).

It is also possible to insert components from another COMSOL Multiphysics model. To do so, right-click the top node (root node) and choose **Insert Components from Model**. An **Insert Components** dialog box appears where you can browse or type the path and name of the COMSOL Multiphysics model file from which you want to insert components in the **Model** field. Select one or more of the components in the model from the **Components** list and then click **OK** to insert them into the current model.

Some aspects when inserting or pasting a component into an existing model:

- Existing components in the open model may conflict with the inserted ones. In such cases, the inserted component will be renamed (for example, from `comp1` to `comp2`). Because inserting a component also inserts many other nodes (geometry, physics, materials, coordinate systems, and so on), these will also be renamed if there are existing ones in the open model.
- Most of the nodes under a component only requires that the other nodes under the same component is inserted with them. However, there are some situations when items outside the component must be included. Especially, the geometry sequence depends on such items. When inserting or pasting a component, the process automatically includes necessary global parameters, global functions, and geometry parts. It will not include references to items in other components.

- File references are inserted as is, and no attempt is made to check if the file exists. When inserting or pasting a component, the process does not include stored files but rely on the original file path to be valid.
- Global parameters, global variables, and global functions in variable expressions (either in other variables or in physics settings) will not be inserted or pasted automatically.
- Deformed geometries and their associated meshes are not included in the insertion or paste operation. These geometries and meshes are typically the result of computing a solution and contain binary data that is not included in the insertion either. They should appear again after re-solving, which you need to do after the insertion anyway.

Altogether, these aspects may cause an insert or paste operation to be incomplete to some degree. In some cases, the difference is reported in a message dialog box after the insertion process has finished. Click **Cancel** in this dialog box to revert the insertion process.

SWITCHING TO ANOTHER COMPONENT

In a model with more than one component, you can switch the focus to another component by selecting from the list of components in the **Home** toolbar's **Model** section. You can also switch to another component by clicking its **Component** node in the model tree (or any other node inside that component).

THE DEFAULT NODES

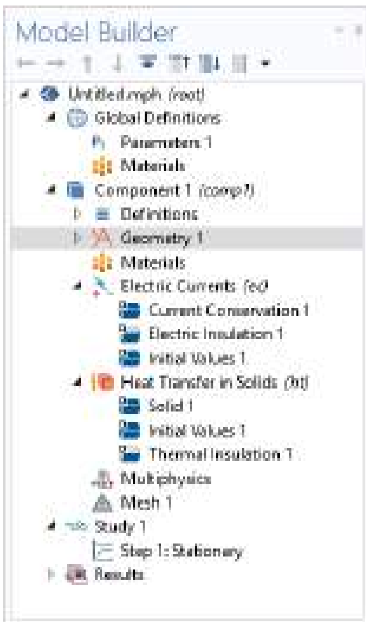


Figure 3-1: An example of the Model Builder default nodes for the Electric Currents and Heat Transfer in Solids interfaces.

These default nodes are normally added under a **Component** node:

- **Definitions**: Contains user-defined variables, selections, views, pairs, functions, probes, nonlocal couplings, and coordinate systems, which are defined locally for the model. See Global Definitions, Geometry, Mesh, and Materials for information about using these local **Definitions** ( ) and **Global Definitions** ( ). Use **Global Definitions** to define **Parameters**, **Variables**, **Functions**, and loads and constraint groups with a global scope — that is, groups that are not specific to one **Component** node.
- **Geometry** ( ): Contains the sequence of geometric objects and operations (or imported CAD data) that defines the model geometry.

- **Materials** ( ): Contains the materials and material properties used as sources for material data in the component. See Materials for detailed information.
- **Physics interfaces** ( ): Any added physics interface displays as a node under **Component** (**Solid Mechanics** in Figure 3-1 for example).
- **Multiphysics** ( ): When a multiphysics interface is added to the Model Builder, this node contains all the relevant multiphysics coupling features for that interface. See Multiphysics Modeling Workflow for more information.
- **Meshes** ( ): Contains the sequences of meshing operations that defines the computational meshes for the model. When there is only one mesh in the model, its **Mesh** node appears directly under the **Component** node.

Branches and Subbranches in the Tree Structure

The **Settings** window has the following sections (also see Figure 3-3):

The label appears on the node as the default node name. The default label is Component 1, but you can change it in the **Label** field.

The name is a string used to define a namespace for the model component and identify variables defined in that component. The default component name is `comp1`, `comp2`, and so on, but you can change it in the **Name** field. See Settings and Properties Windows for Feature Nodes and Displaying Node Names, Tags, and Types in the Model Builder for more information.

GENERAL

This section contains general settings that you normally do not need to change:

Unit System

The default setting in the **Unit system** list, **Same as global system**, is to use the global unit system, which you specify in the root node's **Settings** window. If you want to use another unit system in a model, select it from this list.

Underneath, you can define the coordinates for the frames in a model if you do not want to keep the default names. All frames are always defined. See About Frames for more information about frames.

Spatial frame coordinates

For **Spatial frame coordinates**, the default names are x, y, and z for 3D as well as planar 1D and 2D geometries. For axisymmetric geometries, the default names for the spatial frame coordinates are r, φ (**phi**), and z. If you use the geometry to represent something other than space, or if you for some other reason want to use other names for the spatial coordinates, you can change the names in the fields for the **First**, **Second**, and **Third** coordinate under **Spatial frame coordinates**.

Material frame coordinates

For **Material frame coordinates**, the default names are X, Y, and Z for 3D as well as planar 1D and 2D geometries. For axisymmetric geometries, the default names for the material frame coordinates are R, φ (**PHI**), and Z. You can change the names in the fields for the **First**, **Second**, and **Third** coordinate under **Material frame coordinates**.

Geometry frame coordinates

For **Geometry frame coordinates**, the default names are X_g, Y_g, and Z_g for 3D as well as planar 1D and 2D geometries. For axisymmetric geometries, the default names for the geometry frame coordinates are R_g, φ_g (**PHIg**), and Z_g. You can change the names in the fields for the **First**, **Second**, and **Third** coordinate under **Geometry frame coordinates**.

Mesh frame coordinates

For **Mesh frame coordinates**, the default names are X_m, Y_m, and Z_m for 3D as well as planar 1D and 2D geometries. For axisymmetric geometries, the default names for the mesh frame coordinates are R_m, φ_m (**PHIm**), and Z_m. You can change the names in the fields for the **First**, **Second**, and **Third** coordinate under **Mesh frame coordinates**.

You cannot use the variable for the time, t, as a frame coordinate name.

Geometry Shape Order

The setting in the **Geometry shape order** list determines the order of the curved mesh elements that determine the geometry shape. The default setting is **Automatic**, but it is also possible to select an order such as **Linear**, **Quadratic**, **Cubic**, **Quartic**, **Quintic**, **Sextic**, and **Septic**. The default setting allows for automatic reduction of the order in some cases.

By default, the software avoids inverted elements by an optimization of the element shapes. To deactivate that functionality, clear the **Avoid inverted elements by curving interior domain elements** check box. See Avoiding Inverted Mesh Elements for more information.

- Creating a New Model
- The Root Settings and Properties Windows
- Editing Node Properties, Names, and Labels
- About Frames
- Setting the Unit System for Models
- Using Extra Dimensions
- Curved Mesh Elements

Adding Extra Dimensions to a Model

To add an extra, abstract spatial dimension to a model, right-click the **Global Definitions** node ( ), and then from the **Extra Dimensions** context menu, choose **3D**, **2D Axisymmetric**, **2D**, **1D Axisymmetric**, or **1D** to add an extra dimension to the selected space dimension (requires that **Extra Dimensions** is selected in the **Show More Options** dialog box). An **Extra Dimension** node, in the chosen space dimension, is then added under the **Global Definitions** node in the Model Builder. You can add one or several **Extra Dimension** nodes. It is also possible to attach an extra dimension to several components. Extra dimensions can be useful, for example, to model transport and reactions in two different scales, where one scale is the homogenized scale of a set of larger pores between particles or larger cracks in rocks, and a second smaller scale is the one inside porous particles or in porous rock.

The added node then contains these default nodes: **Definitions**, **Geometry**, and **Mesh**. The settings for the **Extra Dimension** node are the same as for the **Component** node, except it has a unique **Name**.

The default nodes associated to the Extra Dimension are considered the extra dimension geometry and extra dimension mesh. The original geometry and mesh are called the base geometry and base mesh.

Before you can use the extra dimensions in physics interfaces, they must be attached on a selection in the base geometry.

The default node label in the **Label** field is **Extra Dimension 1** for the first **Extra Dimension** node. The component name is a string used to identify variables in the model. The default Extra Dimensions component name is `xdim1`, `xdim2`, and so on, but you can change it in the **Name** field.

Using Extra Dimensions

Branches and Subbranches in the Tree Structure

You can proceed through your modeling in the Model Builder by selecting the branches in the order suggested by the default positions, from the top down, or selecting and defining each branch as needed. One level below the main Component branch are subbranches as described in Table 3-2 and shown in Figure 3-2. The node appearance can also change depending on many factors. See Dynamic Nodes in the Model Builder for examples.

TABLE 3-2: THE MODEL BUILDER BRANCHES AND SUBBRANCHES

FIGURE REF.	ICON	NAME	DESCRIPTION AND LINK TO MORE INFORMATION
Main Branches			
1		Global Definitions	Define global parameters, and right-click to define global Variables, Functions, Load and Constraint Groups, a Materials branch, and optional Geometry Parts. Mesh Parts, and Extra Dimension branches, which are globally available in all model components. See The Global Definitions Node.
2	Various	Component	This branch includes the subbranches Definitions, Geometry, Materials, physics interfaces, and Mesh. You can also right-click the node to Add Physics and Add Mesh at this level. See The Component Node.
3		Study	This subbranch is where you set up study steps and solver configurations to solve a model using one or more study types for different analyses. See Studies and Solvers.
4		Results	The features contained in the subbranches for Datasets, Derived Values, Tables, Export, and Reports are used to present and analyze results. See Results Analysis and Plots.
Subbranches			
5		Definitions (Local)	This subbranch is used to create Variables, Functions, Selections, Coordinate Systems, Nonlocal Couplings, and Probes as well as other definitions that are local to a specific component in your model. See Global and Local Definitions.
6		Geometry (Local)	This branch contains the definition of the model's geometry, where you can import a geometry or build one yourself using the available tools. See Geometry Modeling and CAD Tools.
7		Materials (Global)	This branch makes it possible to add materials and a material switch for material sweeps at the global level. You can add materials in the same way as you do under a Component branch, but materials on the global level are available throughout the model and therefore have no Geometric Entity Selection section. See Global Materials.
7a		Materials (Local)	Collect all material properties organized in Material nodes with a defined geometric scope. Material properties required by any of the physics interfaces and features show up automatically in the defined material's Settings window. See Materials.
8	Various	Physics interfaces	Each physics interface forms its own branch based on the model definition requirements. See The Physics Interfaces and Creating a New Model to start.

TABLE 3-2: THE MODEL BUILDER BRANCHES AND SUBBRANCHES

FIGURE REF.	ICON	NAME	DESCRIPTION AND LINK TO MORE INFORMATION
8a		Multiphysics	This is a main branch but is associated directly with the physics interface branches above it. It contains multiphysics coupling nodes. See The Multiphysics Branch.
9		Mesh	This subbranch collects all meshes defined for a model. If there is only a single mesh in a model, its Mesh node appears directly under the corresponding Component node. See Meshing.
10		Datasets	Datasets refer to the source of data for creating Plots and Reports. It can be a Solution, a Mesh, or some transformation or cut plane applied to other datasets; that is, you can create new datasets from other datasets.
		Derived Values	Used to define evaluations of numerical results — globally, in a point, or integrated quantities. For 2D and 3D plots, you can also get numerical results directly in a table by clicking the plot. See About Derived Values.
		Tables	This subbranch displays the results from integral and variable evaluations defined in Derived Values nodes or by probes and stored in Table nodes. See The Table Window and Tables Node.
	Various	Plot Groups	After adding a 3D, 2D, or 1D Plot Group, plots are added and defined under this subbranch. See Plot Groups and Plots.
		Export	After a model is completed, you can add various components to this and then generate outputs (animations, data, images, or export), or export the information to your computer as image, movie, or data files for use in external documents or for other purposes. See Export Types.
		Reports	This subbranch opens the Report Generator, which is a tool for reporting and documenting models created in COMSOL. It creates a record of the entire model including all the settings made during the modeling process. The report is an overview of the model and includes model properties, geometry, physics interfaces and features, mesh, studies, and results and visualization. See Reports.

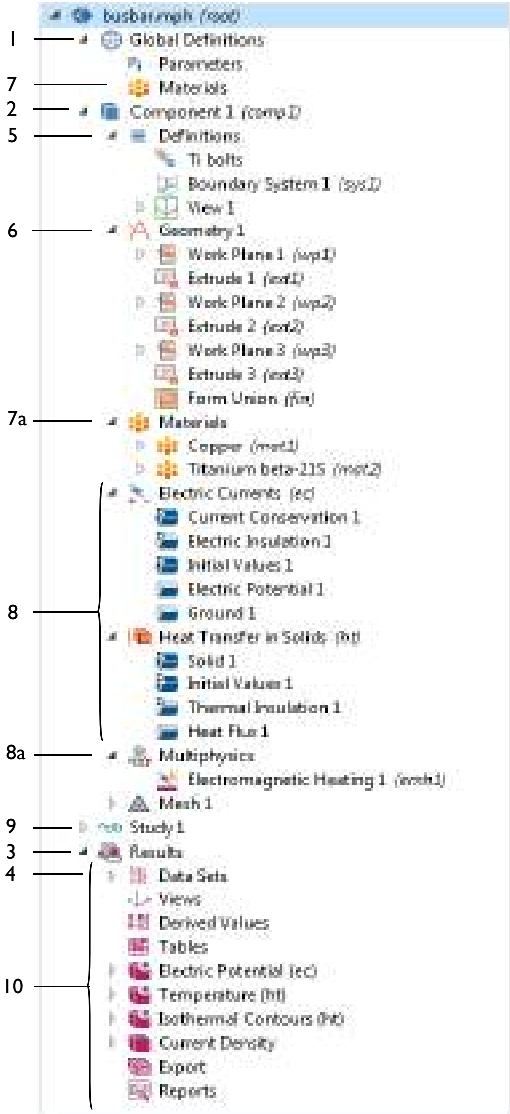


Figure 3-2: An example of the Model Builder tree structure showing the many different types of branches and subbranches available in a model. Refer to Table 3-2 *to learn more about a node. Use the numbers to locate the node in the table.*

SETTINGS WINDOW

For all operating systems, and when any node is clicked in the Model Builder (except a few *container nodes* such as Definitions and Datasets), a corresponding **Settings** window opens with the same name as the node. The **Settings** window contains settings for defining operations and properties specific to a node, as shown in Figure 3-3.

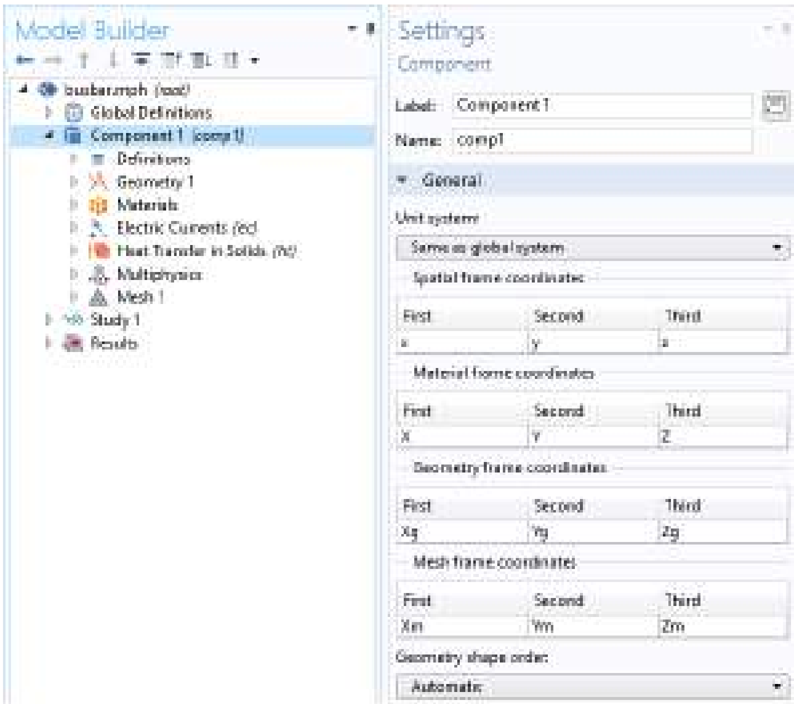


Figure 3-3: An example of a node Settings window. In this example, the Settings window for the Component node opens when the node of the same name is clicked. You can also toggle between the Settings and Properties window from the context menu.

SETTINGS WINDOW FUNCTIONALITY

When an operation or property is updated in the **Settings** window, its effect on the model is displayed in the Graphics window either instantaneously or by clicking the applicable button, which are available in some of the **Settings** window toolbars. If you update settings for the physics interfaces and features, you must recompute the solution to reflect the changes in the physics interface and features.

For most sequences, you can run the sequence by right-clicking the top node of the branch and selecting **Build All** (geometry) and **Build** (mesh), **Compute** (studies), or **Plot** (plot groups), or by pressing F8. These buttons are also on the **Settings** window and in the respective toolbars.

To select the parts of the model to define in a specific **Settings** window, select the relevant geometric entities directly in the displayed model in the Graphics window, from the Selection List window, or as, for example, **All domains** in the **Settings** window.

LABELS AND NAMES IN THE SETTINGS WINDOW

Every **Settings** window has the option to change the node **Label**. The **Label** is the default node description (it defaults to the node **Type** followed by an index suffix). For example, it might be the **Electric Currents** interface, or in Figure 3-3 it is **Component**. You can also right-click and choose **Rename** or press F2.

Some **Settings** windows have the option to change the **Name**. These include physics interfaces, components (as in Figure 3-3), multiphysics couplings, and some Definitions features. The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must begin with a lowercase or uppercase letter (a–z or A–Z). All other characters in the Name must be a lowercase or uppercase letter, a number between 0 and 9, or an underscore (_). See Variable Naming Convention and Namespace for more information.

You can choose to display any combination of the **Name**, **Tag**, and **Type** in the Model Builder. See Displaying Node Names, Tags, and Types in the Model Builder.

In most **Settings** windows, there is also a **Create Shortcut** button ( ) next to the **Label** field. Click it (or press Ctrl+K) to create a shortcut to that node for use in the Application Builder (see the Application Builder documentation for more information). If a shortcut already exists, you can click the **Rename Shortcut** button ( ) (or press Ctrl+K) to rename the shortcut.

PROPERTIES WINDOW

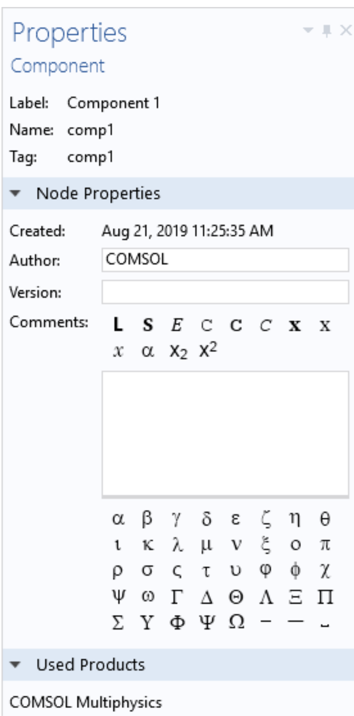


Figure 3-4: An example of a node Properties window. In this example, the Properties window for the Component node opens when you right-click the node and select Properties from the context menu. You can also toggle between the Settings and Properties window from this context menu.

The **Properties** window ( ) in Figure 3-4 is accessed by right-clicking the node and choosing **Properties** from the context menu. The information listed includes the **Label**, **Name**, **Tag**, and the **Node Properties**.

- The **Label** can be edited on the Settings window. The default or edited name is displayed here but cannot be changed in this window.

- The **Name** is available for Component, functions and other nodes under Definitions, Material, and physics interface and multiphysics coupling nodes. You can edit the name on the Settings window for Component nodes, the main physics interface nodes, and for some functions and other nodes under Definitions, where the names servers as an identifier in the namespace for variables or as the function name, for example.
- The **Tag** is unique for each node and is assigned automatically. Tags are primarily used when running COMSOL models in Java or MATLAB. To display the Tag in the Model Builder, click **Model Builder Node Text** on the toolbar and choose **Tag**. See Displaying Node Names, Tags, and Types in the Model Builder for more information.
- The **Node Properties** section includes the following information: **Created**, **Author**, **Version**, and **Comments** (the **root** node is a special case; see The Root Settings and Properties Windows). The **Created** field is automatically assigned by the software. You can edit the **Author**, **Version**, and **Comments** fields in this window. For the **Comments** field, you can add formatting and special characters that will appear in reports. This information can also be included when creating Reports.

- Toolbars and Keyboard Shortcuts
- About Selecting Geometric Entities
- The Graphics Window

Displaying Node Names, Tags, and Types in the Model Builder

SELECTING THE MODEL BUILDER NODE CONTENTS

The Model Builder always shows the label for the nodes. To add more information, on the **Model Builder** toolbar click **Model Builder Node Text** . Then select any combination of options from the list: **Name**, **Tag**, and **Type**. See Figure 3-5 for examples.

- A **Name** is only used in the Model Builder for short names (descriptions) of the nodes. The **Name** can only be changed for the top Component, physics interface nodes and multiphysics couplings, and for **Definitions** nodes. Some **Settings** windows have the option to change the **Name**. See Settings and Properties Windows for Feature Nodes for information about **Label** versus **Name**. The Name and Tag for top level features are often the same.

For **Definitions** features, the Name is displayed differently for Functions, Probes, Nonlocal Couplings, and Pairs. See Common Settings for the Definitions Nodes for more information.

- A **Tag** is unique for each node and is assigned automatically. Tags are primarily used when running COMSOL models in Java or MATLAB. Select **Tag** to display each node's feature name with the predefined tag in curly braces using an italic font. The Name and Tag can be the same.
- A **Type** is automatically assigned by the software and cannot be changed. Select **Type** to display each node's feature type (predefined name). This is the most useful if a node **Label** is renamed or if you use a local language other than English and want to see the predefined name; otherwise, the type and the label are the same (except that the label typically includes a number, such as *Boundary System 1*).

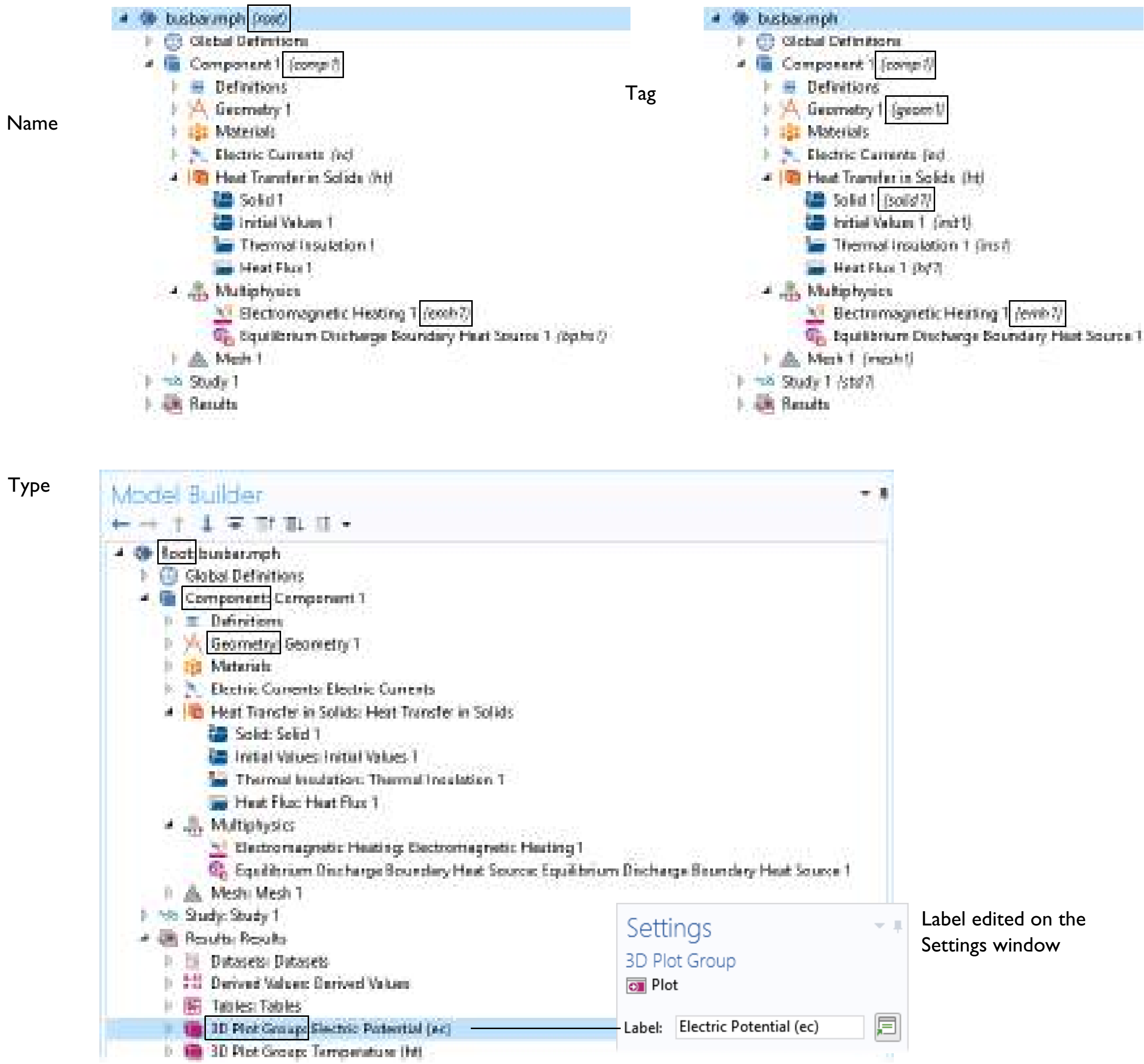


Figure 3-5: Examples of the available combinations on the Model Builder Node Label menu. The second example of a Type shows how this is useful when the Label is edited in the Settings window and you need to know the original type of node.

LABELS

Node **Settings** windows have a **Label** field where you can change the default node description for all levels (except the root node, which gets its name from the model filename). The label can also be changed by right-clicking and choosing **Rename** or by pressing F2.

The **Label** can also be displayed in the Model Builder where it is called a **Tag**. See Settings and Properties Windows for Feature Nodes.

- Editing Node Properties, Names, and Labels
- The Root Settings and Properties Windows

Opening Context Menus and Adding Nodes

In addition to using the toolbars and menus (see The COMSOL Desktop Menus and Toolbars), you can right-click a node to open a *context menu*. The context menu lists all the functionality available as properties and subnodes to a particular node in a branch of the tree. Figure 3-6 shows the context menu for some of the Geometry node options. From the menu you can add additional, and relevant, functionality, operations, or attributes to the sequence. Often there is a mixture of submenus, keyboard shortcuts, or specific features to choose from as in Figure 3-6 and Figure 3-7. There are also standard options such as **Rename**, **Properties**, and **Help**.

The context menu is also further divided and categorized for physics interfaces, as in the section Physics Interface Node Context Menu Layout and Figure 3-7.

To add physics feature nodes to physics interfaces, in general, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the galleries that open.

The layout of the context menu (especially for physics interfaces) depends on whether the nodes are grouped by space dimension. The default is ungrouped nodes. See Grouping Nodes by Space Dimension and Type for an example comparing the different context menus.

OPENING THE CONTEXT MENU

- Right-click any node in the **Model Builder** to open the context menu
- Once a node is highlighted, right-click anywhere in the **Model Builder** to open it.
- Use the shortcuts based on operating system:
  - Windows: Press Shift+F10.
  - macOS: Press Ctrl+F10.
  - Linux: Press Shift+F10.

After selecting an option from the list, an associated **Settings** window opens to the right (by default) of the **Model Builder** window. See Figure 3-3 for an example.

In the context menu, a plus sign next to any icon ( ) means a node of that type is added to the Model Builder.

Figure 3-6: A context menu opens when you right-click any node in the Model Builder. In this example, the options available for the Geometry node are shown.

- Settings and Properties Windows for Feature Nodes
- Grouping Nodes by Space Dimension and Type
- Clearing Sequences and Deleting Sequences or Nodes
- Disabling or Enabling Nodes
- About Geometric Entities

The Physics Nodes

An important part of building a model is where you add physics branches. For example, when Creating a New Model. This branch (see Figure 3-2 for an example) contains the nodes that define the material properties, equations, loads, initial values, boundary conditions, and other parts of the physics that the model describes. All **Settings** windows for the specification of the physics and equations accept parameters and variables as input data.

SPECIFYING PHYSICS INTERFACE SETTINGS

Each physics interface includes nodes for specifying all input data for a specific physics in a model:

- Material properties and material models
- Boundary and physics interface conditions
- Equations (for equation-based modeling)
- Initial conditions

In addition, you can specify weak form contributions and element types for additional flexibility.

Specifically, the settings are available on the following parts of the geometry:

- Domains
- Boundaries
- Edges
- Points
- Additional properties that are independent of the geometry

Not all of these options are available for all geometry types and physics interfaces.

PHYSICS FEATURE NODES BY SPACE DIMENSION

The physics feature nodes indicate the geometric entity level (domains, boundaries, edges, points, or pairs) based on the space dimension of the Component (see Table 3-3). The nodes also correspond to The Graphics Window Toolbar Buttons and Navigation, some of which are also based on space dimension.

See Physics Exclusive and Contributing Node Types and Physics Node Status for examples of other differences to how the nodes display in the Model Builder.

TABLE 3-3: PHYSICS FEATURE NODES BY SPACE DIMENSION

NAME	3D	2D AND 2D AXISYMMETRIC	1D AND 1D AXISYMMETRIC
Domain level			
Domain level, default node			
Boundary level			
Boundary level, default node			
Boundary level, Pairs			
Point level			—
Edge level		—	—

- The Physics Interfaces
- Physics Interface Default Nodes
- Physics Interface Node Context Menu Layout
- Physics Node Status

Physics Interface Default Nodes

When you add a physics interface, the software automatically adds a corresponding physics interface branch in the tree, which typically includes a number of default nodes, including but not limited to:

- A model equation or material model node, typically on the domain level. This node defines the domain equations (except optional sources, loads, reactions, and similar contributing domain quantities) and the related material properties or coefficients.
- A boundary condition node. For multiphysics interfaces there is one boundary condition for each participating physics.
- For axisymmetric models, the symmetry axis has an **Axial Symmetry** boundary condition (see Physics Interface Axial Symmetry Node).
- An **Initial Values** node for specifying initial values for a time-dependent simulation or an initial guess for the solution to a nonlinear model (see Specifying Initial Values).

In most cases, the default nodes' initial selections include all domains or all boundaries (or all instances of another geometric entity level). Their selection is always every instance that is not overridden by another node on the same geometric entity level. It is not possible to delete such default nodes, but you can copy and duplicate all default nodes. Some multiphysics interfaces also add default nodes with no initial selection, which are possible to delete from the model. Default nodes include a *D* (for "default") in the upper-left corner ( ) to indicate their special status. The copy or duplicate of a default node is a node of the same type but behaves as a normal node with an initially empty selection.

For example, for a geometry with four boundaries, the default boundary condition's initial selection includes all four boundaries. If another exclusive boundary condition for Boundary 3 is added, that boundary becomes overridden (inactive) in the default boundary condition's selection. If you disable or remove that boundary condition, the default boundary condition becomes active for Boundary 3 again. You cannot change a default node's selection.

Some physics interfaces also add standard nodes directly when you add them to a model. They represent functionality that is likely to be useful but that you might want to make only active on a part of the geometry or delete. Such nodes do not include a *D* in the upper-left corner.

- Physics Feature Nodes by Space Dimension
- Physics Node Status
- Physics Exclusive and Contributing Node Types
- Dynamic Nodes in the Model Builder

Physics Interface Node Context Menu Layout

The context menu opens when you right-click a physics interface node, or any node in the Model Builder (see Opening Context Menus and Adding Nodes). Depending on the space dimension, this menu is divided into these sections for most physics interfaces: the first section contains domain settings, the second boundary settings, the third edge settings, and the fourth has point settings.

!	There can be menu items with the same name but applied at different geometric entity levels.

!	To add physics feature nodes to physics interfaces, in general, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the gallery that opens.

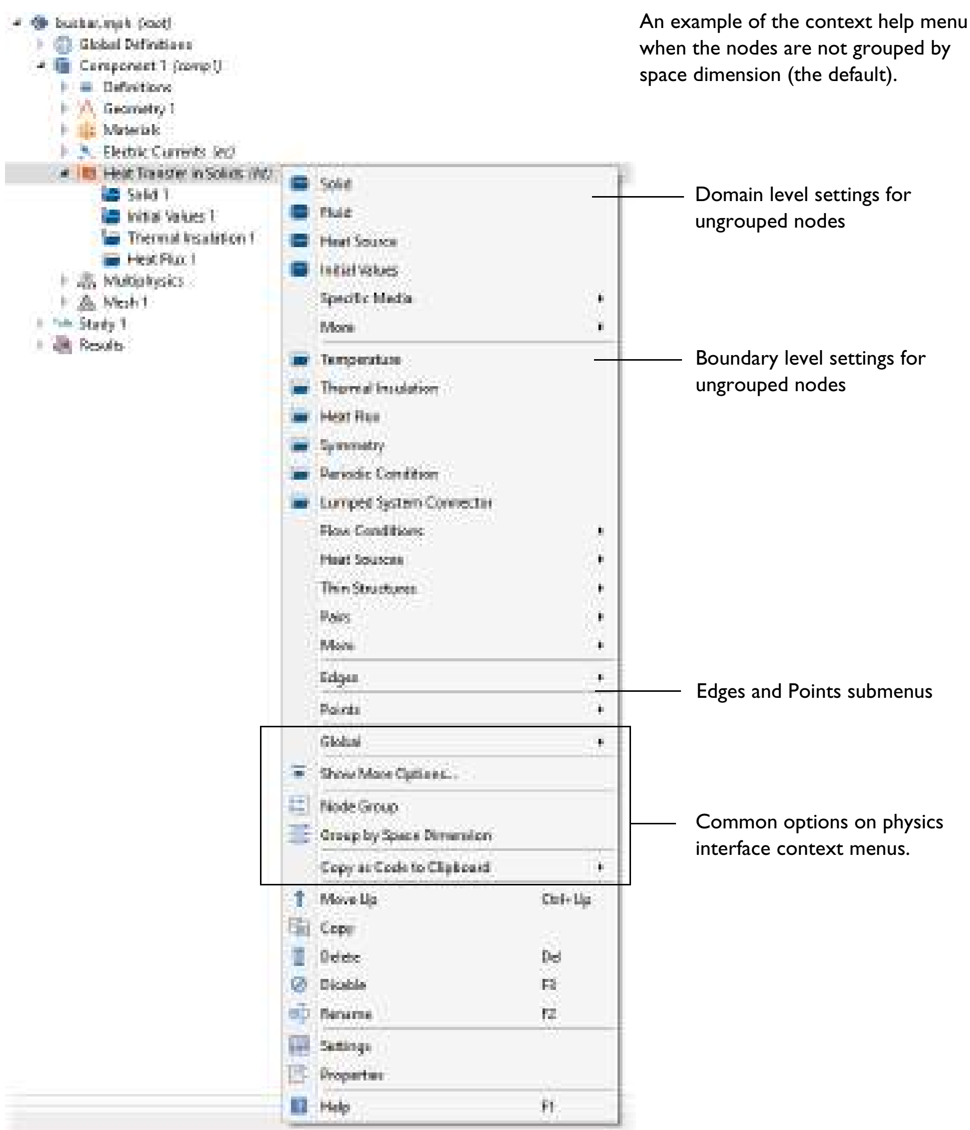


Figure 3-7: An example of a Heat Transfer in Solids interface context menu. The choices are based on the Component dimension (in this example it is 3D) as well as the physics interface. The menu is further divided by geometric entity level (domains, boundaries, edges, and points).

As shown in Figure 3-7, the context menu layout is also based on whether the nodes are not grouped (the default) or if **Group by Space Dimension** is selected.

Physics Exclusive and Contributing Node Types

The nodes for the physics interfaces and features are in a sequence, which acts like a macro that the software runs in a top-down order. Depending on the selection for each node, a node can totally or partially override, or shadow, a node earlier in the sequence. How the software treats these nodes depends on the relationship. There are two different types of nodes: *exclusive* and *contributing* (see Figure 3-8)..

The exclusive and contributing nodes maintain the described behavior only in relation to similar types of nodes within the same physics interface (for example, you can have a temperature constraint and a pressure constraint for the same boundary in the same model component).

What the node looks like in the Model Builder is based on the space dimension. See Physics Feature Nodes by Space Dimension.

EXCLUSIVE NODES

The use of an *exclusive node* means that only one can be active for a given selection. That is, if you add another exclusive node (for example, an identical node) with the same selection, the first exclusive node is overridden and thus has no effect.

Typical exclusive nodes include model equations, initial values, and boundary conditions that are constraints, such as prescribed values for displacements, temperatures, pressures, and so on, or other Dirichlet-type conditions, including special variants of these such as ground conditions in electromagnetics and fixed constraints in structural mechanics. Also some boundary conditions that are not constraints but have a definitive meaning are exclusive nodes — for example, electric insulation, thermal insulation, and no-flow conditions. Depending on the selections for each node, an exclusive node can override another node partially. Nodes are exclusive only within their specific physics interface. When a node is selected in the Model Builder tree, nodes that are overridden by the selected node have a red arrow in the lower-left corner of the icon ( ), and nodes that override the selected node display a red arrow in the upper-left corner of the icon ( ).

CONTRIBUTING NODES

A *contributing node* means you can have more than one of these nodes with the same selection and that the software adds these together when evaluating the model. Typical contributing nodes are loads, fluxes, and source terms, where you can have more than one of each type that is active on the same domain or boundary, for example. The total effect is then a sum of each contributing node. When a node is selected in the Model Builder tree, the tree shows other nodes, which the current node contributes with, indicated using a yellow dot to the left of the icon (for example, in this boundary level icon). See also Figure 3-8 for an example.

ORDER OF EXCLUSIVE AND CONTRIBUTING NODES

An exclusive node typically override all other nodes that share some common geometric entity and that appear above it in the list of nodes under a physics interface. Conversely, a contributing node can contribute with an exclusive node that appears above it. For example, in a heat transfer interface, a **Temperature** exclusive node overrides a **Heat Flux** contributing node defined above it (when defined on some common boundaries). If you switch the order of those nodes, so that the **Heat Flux** node appears below the **Temperature** node, it then contributes with the **Temperature** node. Because the **Temperature** node imposes a constraint on the temperature, the computed solutions are identical when evaluating the temperature. However, the contributing heat flux changes the reaction force of the temperature condition, which you can verify by integrating the reaction force on a boundary using the `reacf` operator, for example.

LISTING OVERRIDES AND CONTRIBUTIONS

If your preferences include showing the **Override and Contribution** section in the **Settings** windows for physics nodes, you can find the following information about how exclusive and contributing nodes interact in the model. Click the **Show More Options** button ( ) and select the **Override and Contribution** in the **Show More Options** dialog box to display the information as in Figure 3-8 and described below.

- The **Overridden by** list contains the names of the nodes that the selected node is overridden by. The selected node is then overridden by these nodes at least partially, and the **Selection** list contains **(overridden)** for the geometric entities (boundaries, for example) where it is overridden. The nodes that the selected node is overridden by are indicated using a red arrow in the lower-left corner of the icon such as in this boundary level icon .
- The **Overrides** list contains the names of the nodes that the selected node overrides (where the current node is active). The nodes that the selected node overrides are indicated using a red arrow in the upper-left corner of the icon such as in this boundary level icon .
- The **Contributes with** list contains the names of the nodes that the selected node contributes with for at least some shared selection. The nodes that the selected node contributes with are indicated using a yellow dot to the left of the icon such as in this boundary level icon .

If a physics node is disabled locally in a study step using the **Physics and Variables Selection** section in the study step's **Settings** window, the indications of overrides and contributions in the Model Builder are unchanged (but disabled physics nodes get an asterisk to indicate that their state has been changed in at least one study step). However, the local variables and physics tree in the study step's **Settings** window displays the overrides and contributions taking the disabled nodes into account.

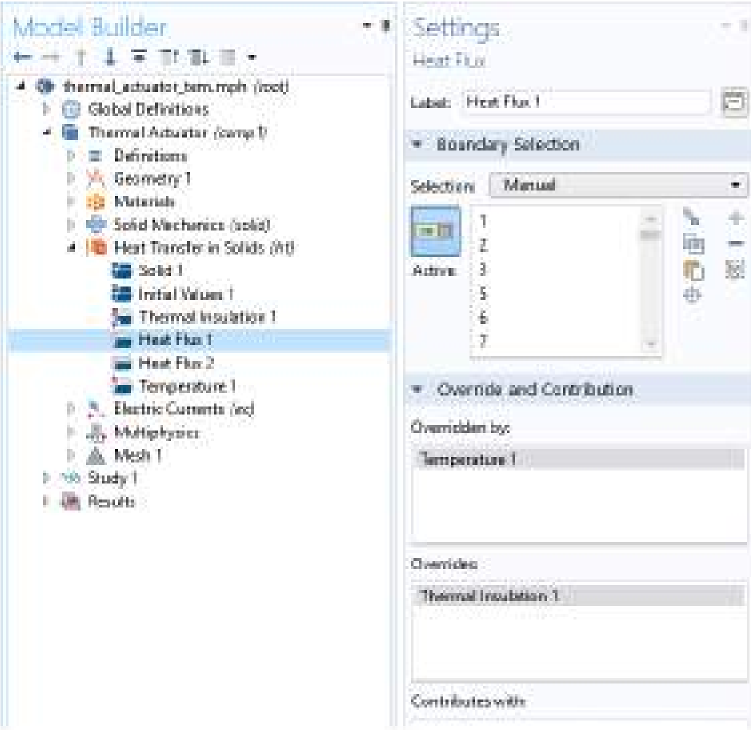


Figure 3-8: The Override and Contribution section lists other physics nodes that the selected node is overridden by, overrides, or contributes with.

- Physics Node Status
- Physics and Variables Selection
- Physics Interface Default Nodes

Physics Node Status

The status of a physics node depends on if it is a default node, the selection that it applies to, and other nodes in the same branch that can override nodes earlier in the sequence. You can change the order of nodes (except the default nodes) by moving them up or down.

OVERRIDDEN SELECTIONS

A node can be partially or completely *overridden* by another node further down in the same branch of the model tree that is of a similar, exclusive type. For example, if you specify a temperature boundary condition on boundary 1 and boundary 3, and then add another temperature boundary condition for boundary 3, the first temperature boundary condition is overridden on boundary 3. In the **Settings** window for the **Temperature** nodes that define the temperature boundary condition, the **Selection** list then shows **3 (overridden)** to indicate that the temperature boundary condition defined on this selection is overridden for boundary 3 but is still active on boundary 1. Deleting or disabling the other temperature boundary condition on boundary 3 reactivates the original temperature boundary condition, and then shows **3** (without the **(overridden)** indication).

SELECTIONS THAT ARE NOT APPLICABLE

For selections that are not applicable for a node (such as interior boundaries for an boundary condition that is only applicable for exterior boundaries), the **Selection** list then shows **(not applicable)** next to entries that are, in this case, interior boundaries.

EMPTY SELECTIONS

For physics nodes that you add to a model (that is, not for default nodes), a warning appears on the physics node when the node has no selection. An added source or boundary condition, for example, has no effect on the model, so leaving an added physics node with an empty selection might be an unintended state, leading to unexpected simulation results.

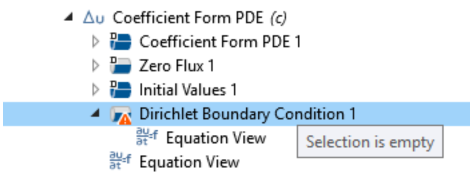


Figure 3-9: The warning and tooltip for an added physics node with an empty selection.

The warning for an empty selection does not appear in the following cases:

- For completely overridden selections.
- When no applicable entities are available.
- When selections are set to all entities, but the geometry has no entities on the given entity level (for example, no geometry or domain selection on a surface geometry).

ENABLING AND DISABLING NODES

By enabling or disabling physics nodes, you can activate and deactivate (shadow) other physics interface nodes that appear higher up in the physics interface branches.

- Physics Interface Default Nodes
- Physics Exclusive and Contributing Node Types
- Physics Feature Nodes by Space Dimension
- Clearing Sequences and Deleting Sequences or Nodes
- Disabling or Enabling Nodes

Dynamic Nodes in the Model Builder

The Model Builder is a dynamic environment. As your model is built and analyzed, there are numerous ways to quickly identify nodes that change status during the process. Table 3-4 lists generic examples and links to the dynamic visual aids that are used to help you.

- Branches and Subbranches in the Tree Structure
- The Component Node

TABLE 3-4: DYNAMIC NODES — VISUAL AIDS TO IDENTIFICATION

ICON	TYPE	NODE EXAMPLE AND LINK TO MORE INFORMATION (WHERE APPLICABLE)
	Error	For example, on a Material node . See Errors Relating to the Material Nodes.
	Error node	Errors and Warnings
	Current node, not built (yellow frame)	For example, on a Geometry node (). This node is also displaying the asterisk indicating the node is being Edited. The asterisk also appears on plot nodes when the plot has not been updated to reflect changes in the data or settings (for example, after re-solving), See The Current Node in Geometry Sequences.
	Current node (green frame)	A current node is used for Geometry and Meshing nodes and indicates that the feature or sequence of steps has been built. It is a green line on the left and upper edges of the node. For example, on a Geometry node , after building. Also see The Current Node in Geometry Sequences.
	Enabled sequence	During solution processing, the particular sequence that is enabled and runs when selecting Compute has a green border around its icon (). See Computing a Solution.
	Harmonic Perturbation	For example, on a boundary level node for the Electric Currents interface, Electric Ground node . See Harmonic Perturbation, Prestressed Analysis, and Small-Signal Analysis.
	Warning	For example, on a Mesh node .
	Editing, or in process of editing, a node	For example, on a Mesh node . This node is also displaying the asterisk indicating the node is being Edited. Also indicates physics interface nodes that have been disabled in a Study Step. See Editing and Building Geometry Nodes for Geometry nodes for example.
	Pairs	For example, on a 3D Boundary Level node . See Identity and Contact Pairs.
	Pairs — Fallback Features	For example, on a 3D Boundary Level pair node . See Identity and Contact Pairs.
	Contributing node	For example, on a 3D boundary level node . See Physics Exclusive and Contributing Node Types and Physics Node Status.
	Default node	For example, on a 2D boundary level node . See Physics Interface Default Nodes.
	Override	For example, on a 3D boundary level node . See Physics Exclusive and Contributing Node Types.
	Overridden	For example, on a 3D boundary level node . See Physics Exclusive and Contributing Node Types
STUDY STEPS ANALYSIS		
	Solve For	For example, a Laminar Flow interface where the green dot in the lower-right corner indicates that the study solves for the degrees of freedom in this physics interface. See Physics and Variables Selection.
	Disable in Solvers	For example, a Laminar Flow interface is enabled (not dimmed), shows that the study step provides degrees of freedom (the yellow dot in the lower-right corner), and has a change of state indicated by the asterisk. The yellow dot means that the study step provides degrees of freedom but does not solve for the physics interface. See Physics and Variables Selection.
	Change of State (editing)	An asterisk appears in the upper-right corner of nodes for which you change their state in the study step's selection tree compared to their state in the main model tree in the Model Builder. For example, for the Joule Heating interface .

TABLE 3-4: DYNAMIC NODES — VISUAL AIDS TO IDENTIFICATION

ICON	TYPE	NODE EXAMPLE AND LINK TO MORE INFORMATION (WHERE APPLICABLE)
	Disabled in Model (provides no degrees of freedom) and shows a change of state	In this example, a Transport in Diluted Species interface is disabled (unavailable), provides no degrees of freedom (red dot in the lower-right corner), and has a change of state indicated by the asterisk. See Physics and Variables Selection.
LOAD AND CONSTRAINT GROUPS		
	Load Group	This is an example of a Boundary Load node with a load group. This is for a 2D model at the boundary level. See Load Group and Using Load Cases.
	Constraint Group	This is an example of a Fixed Constraint node with a constraint group. This is for a 2D model at the boundary level. See Constraint Group and Using Load Cases.

Physics Symbols

There are physics symbols available with structural mechanics and some other physics features to help you to graphically indicate boundary conditions, loads, and other physics features:

- Physics Symbols for Structural Mechanics and Other Physics
- About Coordinate Systems and Physics Symbols
- Displaying Physics Symbols in the Graphics Window — An Example

- Solid Mechanics and Using Load Cases
- The Graphics Window

PHYSICS SYMBOLS FOR STRUCTURAL MECHANICS AND OTHER PHYSICS

To display the physics symbols listed in Table 3-5, select the **Enable physics symbols** check box under **Physics Symbols** in the main physics interface node's **Settings** window. This check box is not selected by default.

Once you have turned on the physics symbols for a certain physics interface, you can fine-tune the display. Every feature that has associated physics symbols now has a **Show physics symbols** check box, by which you can control the display of the symbols for that specific feature.

In the **Physics Symbols** section in the settings for the physics interface, you can click the **Select All** button, which displays all symbols in that physics interface by selecting all **Show physics symbols** check boxes in the **Settings** windows for the physics features that include symbols. Similarly, the **Clear All** button clears all **Show physics symbols** check boxes in the individual physics features.

The following symbols are available with the applicable structural mechanics feature nodes and with some other physics interfaces (this table is a partial list of available symbols).

TABLE 3-5: PHYSICS SYMBOLS

SYMBOL	SYMBOL NAME	DISPLAYED BY NODE	NOTES
	Added Mass[1]	Added Mass	
	Antisymmetry[1]	Antisymmetry	
	Body Load[1]	Body Load	
	3D Coordinate System		Green indicates the Y direction, blue indicates the Z direction, and red indicates the X direction.

TABLE 3-5: PHYSICS SYMBOLS

SYMBOL	SYMBOL NAME	DISPLAYED BY NODE	NOTES
	2D Coordinate System		Green indicates the Y direction and red indicates the X direction.
	Distributed Force	Boundary Load Face Load Edge Load	Can be displayed together with the Distributed Moment symbol, depending on the values given in the node.
	Damping[1]	Spring Foundation	Can be displayed together with the Spring symbol, depending on the values given in the node.
	Distributed Moment[1]	Boundary Load Face Load Edge Load	Can be displayed together with the Distributed Force symbol, depending on the values given in the node.
	Fixed Constraint	Fixed Constraint	
	No Rotation[1]	No Rotation	
	Pinned[1]	Pinned	
	Point Force	Point Load	Can be displayed together with the Point Moment symbol, depending on the values given in the node.
	Point Mass[1]	Point Mass	
	Point Moment[1]	Point Load	Can be displayed together with the Point Force symbol, depending on the values given in the node.
	Prescribed Displacement	Prescribed Displacement	
	Prescribed Velocity[1]	Prescribed Velocity	
	Prescribed Acceleration[1]	Prescribed Acceleration	
	Rigid Connector[1]	Rigid Connector	A line is drawn to each connected boundary,
	Roller	Roller	
	Spring[1]	Spring Foundation Thin Elastic Layer	Can be displayed together with the Damping symbol, depending on the values given in the node.
	Symmetry	Symmetry	
	Thin-Film Damping[2]	Thin-Film Damping	

[1] Requires the Structural Mechanics Module
[2] Requires the MEMS Module

ABOUT COORDINATE SYSTEMS AND PHYSICS SYMBOLS

Physics symbols connected to a node for which input can be given in different coordinate systems are shown together with a coordinate system symbol. This symbol is either a triad or a single arrow. The triad is shown if data are to be entered using vector components, as for a force. The single arrow is displayed when a scalar value, having an implied direction, is given. An example of the latter case is a pressure.

In both cases, the coordinate directions describe the direction in which a positive value acts. The coordinate direction symbols do not change with the values actually entered for the data.

Physics symbols are in most cases displayed even if no data values have been entered in the node.

In some cases, a single feature can display more than one symbol. An example is the Point Load node in the Beam interface, which can display either the Point Force symbol (→), the Point Moment symbol (↠), or both, depending on the data entered. In those cases, no symbol is shown until nonzero data is entered.

For cases when physics symbol display is dependent on values given in the node, it can be necessary to move to another node before the display is updated on the screen.

DISPLAYING PHYSICS SYMBOLS IN THE GRAPHICS WINDOW — AN EXAMPLE

1 Add a physics interface, for example, **Solid Mechanics**, from the **Structural Mechanics** branch.
2 In the Solid Mechanics node's Settings window, under **Physics Symbols**, select the **Enable physics symbols** check box and then click the **Select All** button.
3 Add any of the feature nodes listed in Table 3-5 to the physics interface. Availability is based on license and physics interface.

The physics symbols also display for any multiphysics interface that includes Structural Mechanics feature nodes or other physics feature nodes with symbols.

4 When adding the boundary, edge, or point (a geometric entity) to the **Selection** list in the feature **Settings** window, the symbol displays in the **Graphics** window. See Figure 3-10 and Figure 3-11.

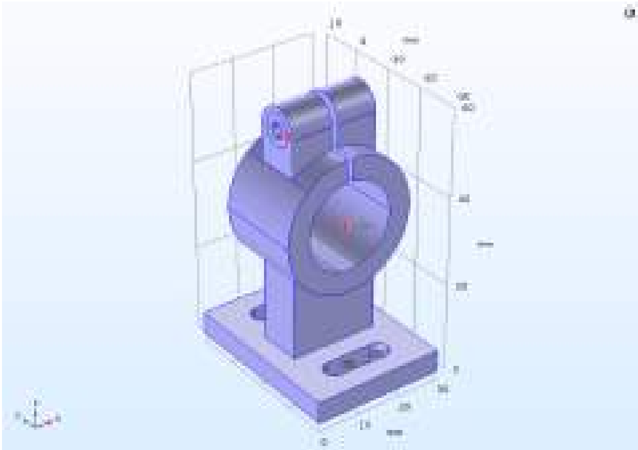


Figure 3-10: Example of Boundary Load physics symbols as displayed in the COMSOL Multiphysics model "Deformation of a Feeder Clamp".

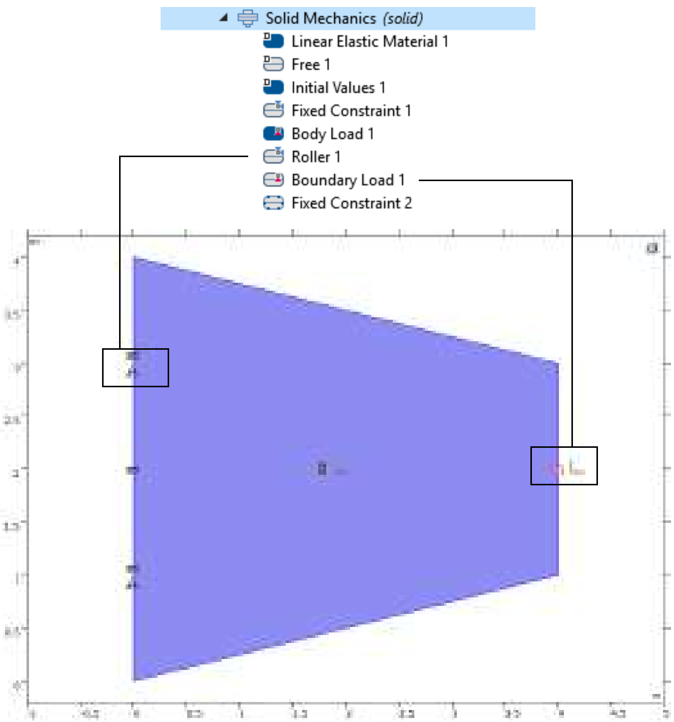


Figure 3-11: Example of Roller and Boundary Load physics symbols as displayed in the COMSOL Multiphysics model "Tapered Cantilever".

Errors and Warnings

COMSOL Multiphysics reports problems of two types: errors and warnings.

ERRORS

Errors prevent the program from completing a task. For errors, a **COMSOL Error** window appears with a brief error description and, in some cases, an **Open Log File** button for additional information. Under the node where the error occurred there is, in most cases, also an **Error** subnode ( ) that contains an error message that generally provides additional information. Also, for many error types, the icon for the node where the error occurred appears with a red cross in the lower-right corner. For some errors there is also a link to more diagnostic information on the COMSOL website.

LICENSE ERRORS

It is possible to open and postprocess models that include functionality that you have blocked or that your license does not include. Nodes with functionality that requires a license for a product that is blocked or not available get a **License Error** subnode ( ), where you find information about the missing but required product license. Unless you disable or remove such nodes, it is not possible to re-solve such models.

Some specialized plot types require a license for an add-on product and are then also unavailable if you postprocess models that include such plots and your license does not include the required product.

It is not possible to open models that require a license for the Material Library, ECAD Import Module, CAD Import Module, LiveLink™ *for* MATLAB®, or any of the CAD LiveLink™ products if your license does not include this required product.

WARNINGS

Warnings are problems that do not prevent the completion of a task but that might affect the accuracy or other aspects of the model. Warnings typically appear in the **Log** window ( ). The warning message also appears as a **Warning** subnode ( ) under the node from which the warning was sent.

INDICATION OF UNEXPECTED, UNKNOWN, OR INCONSISTENT UNITS

The unit display appears orange for the properties in the settings for the physics interface, physics features, and materials that have invalid or inconsistent units or a different unit than expected.

Inconsistent Units

An inconsistent unit can occur by summing terms with units that represent different physical quantities, such as `273[K]+3[ft]`. A tooltip displays a message at the corresponding field.

Unexpected Unit of Input

In the case of a valid but unexpected unit, this message contains the deduced and expected units in the current unit system.

Unknown Unit

This message appears when a unit bracket contains invalid units.

Syntax Errors

A unit display that appears red contains a syntax error, which can be due to, for example, missing or misplaced parentheses.

Evaluating Unexpected or Inconsistent Units

If an unexpected or inconsistent unit appears in a text field for a physical property, the COMSOL Multiphysics software ignores the unit and uses the numerical value, including an SI prefix if present, as the input to the model. For example, in a text field for density using SI units, the software interprets `2930[K]` as 2930 kg/m^3 and `2930[mK]` as 2.930 kg/m^3.

ERRORS AND WARNINGS IN A GEOMETRY SEQUENCE

If an error occurs when you build a node, the build stops. The node with the problem then gets an **Error** subnode ( ) that contains the error message. Also, the node's icon displays with a red cross in the lower-left corner.

After a successful build of a node, a warning message can sometimes display as a **Warning** subnode ( ). If a warning message exists, the node's icon displays with an orange triangle in the lower-right corner.

ERRORS AND WARNINGS IN MESHING SEQUENCES

If a problem occurs when you build a node, the build continues if it is possible; otherwise, the build stops. Continuing means that geometric entities where the operation failed are skipped and the problems are reported as **Error** subnodes ( ) under the operation node. The build process continues with remaining nodes in the meshing sequence.

When the building of the meshing sequence is completed, the error window appears to show the first error reported. If there are several errors, you have to inspect the sequence for nodes with a Warning status and corresponding **Error** nodes to find all errors. If a node has a Warning status, the node's icon includes an orange triangle in the lower-right corner (see Dynamic Nodes in the Model Builder).

In some cases, you get a **Warning** node ( ) even though meshing completed successfully. This happens, for example, when geometric entities are much smaller that the desired mesh element size, and you should interpreted the warning as a hint that the geometry needs to be simplified to avoid an unnecessarily fine mesh.

The **Error** and **Warning** nodes and their subnodes often contain selections that highlight where the problem is located in the geometry.

If meshing cannot continue, all building stops and the node gets an Error status, which the program indicates by adding a red cross in the lower-right corner of the node's icon. You find information about the error in an **Error** subnode ( ) of the node where the error occurred. If the node is part of a sequence build, the build stops and the preceding node becomes the current node.

ERRORS AND WARNINGS IN SOLVER SEQUENCES

Issues encountered when running a solver or generating a mesh are treated in two different ways depending on if it is possible to avoid the problem and continue the operation or if the operation must be stopped. In the first case, a **Warnings** node ( ) appears under the node in the model tree that caused the problem. In the second case, an **Error** node ( ) appears under the node in the model tree that caused the error.

A **Warnings** node ( ) can also appear under a **Compile Equations** node if some input to the solvers uses inconsistent units, for example.

WARNINGS DURING POSTPROCESSING

For things like empty plots, **Warnings** nodes ( ) can appear in the **Results** branch during postprocessing. However, you do not get any warnings in these cases:

- Plotting with the **Dataset** list set to **None** or a dataset that cannot be evaluated (for example, because the model does not contain any solution).
- Plotting without having set any expressions.

If you still have problems, contact technical support from the Support Center page at www.comsol.com/support.

- Using Units
- Unit Systems
- Dynamic Nodes in the Model Builder

Modeling Development Tools

Overview

In addition to the available functionality in COMSOL Multiphysics and the add-on products included in your license, you can add custom functionality by defining methods and settings forms and by combining them into add-ins for general use in any model. There are also built-in tools for comparing the contents of two models. All this functionality is available from the **Developer** toolbar (see Developer Toolbar). There is also an Add-in Libraries window for adding existing add-ins to a model. See the following sections for more information:

- Creating and Running Methods in Models
- Method Calls
- Creating and Using Settings Forms and Dialogs
- Creating Add-ins
- The Add-in Libraries Window
- Comparing Models and Applications

Creating and Running Methods in Models

You can use the method editor capabilities in the Application Builder to create methods that you can run to automate or extend operations in the **Model Builder** tree such as creating a geometry or running some special solver sequence. To add a method, go to the **Developer** ribbon and click **New Method** ( ) in the **Create Methods** section. In the **New Method** window, specify a method name in the **Name** field and click **OK**. The Application Builder then opens, and the new model method appears under **Methods** ( ) in the **Application Builder** tree. You can then record or write code for the method in the model editor window. See the Application Builder documentation for more information about methods. By default, the **Show in Model Builder** check box is selected in the settings for the **Method** node. All methods with that check box selected appear under **Run Method** in the **Model Builder**.

To run a method in the Model Builder, click **Run Method** ( ) in the **Run Methods** section and choose the method to run. The Model Builder tree is updated according to the changes that the method includes. Click **Stop** ( ) to stop a running method to debug it, for example. You can open the **Debug Log** window from the **Windows** menu to view debug information. You can also add breakpoints for debugging purposes. You can also click **Run Method Call** ( ) to run a method call in a **Method Call** node (see Method Calls below). Method calls support input arguments for the model methods, so that you, for example, can use two instances of the same model method with different input values. Model methods with inputs are not available for **Run Method**. For methods without inputs, running a method directly or through a method call is equivalent.

	Make sure that the model tree is in a state that is compatible with the method that you run. Otherwise, the method code may not work or may produce unexpected results. Also, there is no undo operation after running a model method.

	See the *Introduction to Application Builder* for an example of a model method and the *Application Builder Reference Manual* for information about the **Model Method** node and about creating and debugging methods in general.

Method Calls

You can add any available model method as a **Method Call** node ( ), which appears under **Global Definitions**. If the **Group by Type** option is enabled, the **Method Call** nodes are grouped under the **Method Calls** node ( ). In the **Settings** window, the model method that this **Method Call** node calls is listed as, for example, **Model method: modelmethod1**.

At the top of the **Settings** window, click the **Run** button ( ) or press F8 to run the model method for this method call. Click the **Stop** button ( ) to stop a running model method (this button is unavailable if a model method is not running).

Click the **Edit Method** button ( ) to open the model method in a method editor window in the Application Builder, where you can make changes and additions to the model method.

You can also right-click the **Method Call** node and choose **Run** (or press F8) or **Edit Method** .

The **Method Call** node's **Settings** window includes the following section:

INPUTS

In this section, any inputs that you have defined for the model method appear, using the input's description as the label and a text field for entering a value for the input (the text field contains a default value if it has been defined for that input). If the input is a Boolean, it appears as a check box instead. If the input is a 2D array, Array 1D Boolean, or Array 1D String, it is shown as a table, where you can add values and add rows and columns as applicable. Contrary to running model methods directly, method calls can use model methods with inputs defined in the **Settings** window for model methods in the Application Builder. That way you can, for example, use multiple instances of a model method with different input values.

Creating and Using Settings Forms and Dialogs

Forms that you create in the Application Builder can also be used as settings forms in the **Model Builder** tree that can provide customs settings connected to methods for performing some special task in a model. Such forms become available in the **Model Builder** if you select the **Show in Model Builder** check box. You can add them as nodes in the **Forms** section of the **Developer** toolbar by choosing them from the **Settings Form** list ( ). That form is then added as a **Settings Form** node under **Global Definitions** (or **Global Definitions>Settings Forms**, if **Group by Type** is active).

You can also show such forms as modal dialog boxes by choosing them from the **Show Dialog** list (in the **Forms** section of the **Developer** toolbar.

Forms that include Graphics objects cannot be used as settings form or dialog boxes. Instead you can plot in the main **Graphics** window or in other plots windows using the built-in `selectNode` method or the `window` property for a plot group.

To make use of the latest definition of a form in the Application Builder when it is shown in the Model Builder, click the **Update Forms** button ( ) in the **Forms** section of the **Developer** toolbar, which updates all settings forms' features. You can also click the **Update** button ( ) in the in the **Settings** window for a **Settings Form** node to update that particular settings form.

Click the **Edit Form** button ( ) the in the in the **Settings** window for a **Settings Form** node to move to its form editor window in the Application Builder.

See the Application Builder documentation for more information about creating forms and methods.

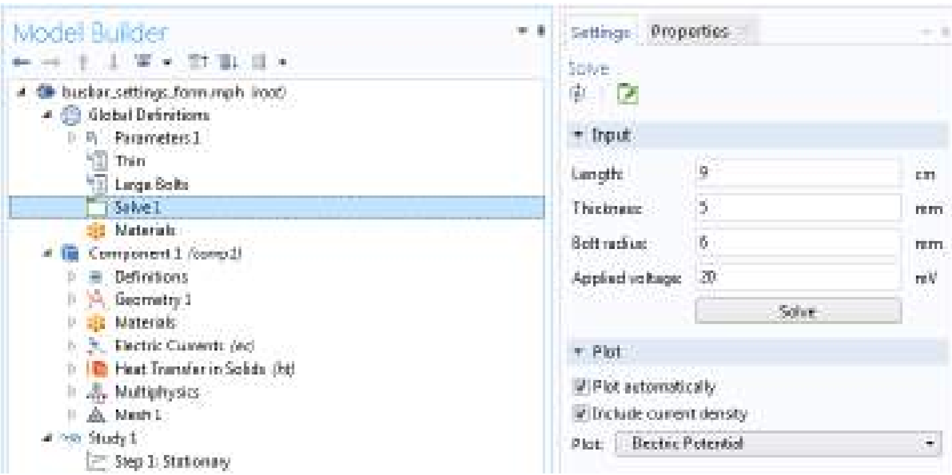


Figure 3-12: An added settings form under Global Definitions provides custom settings.

Creating Add-ins

Add-ins, or add-in programs, are extensions to the COMSOL Multiphysics software and make it possible to share methods and settings forms between several models. An add-in consists of a regular COMSOL Multiphysics MPH-file in which you have added forms and methods using the Application Builder to provide a settings window with some functionality that can simplify or extend the built-in functionality in the COMSOL Multiphysics software for some application that might be useful for a range of model files. You can collect add-ins into a user-defined add-in library (see The Add-in Libraries Window below). Opening the add-ins that are included in the COMSOL Multiphysics as MPH-files from the file system makes it possible to study the forms and methods in the Application Builder. See the Application Builder documentation for more information about how to create an add-in in the Application Builder and general information about forms and methods.

Using Add-ins

To use an add-in in the current model, choose it from the **Add-ins** list ( ) on the **Developer** ribbon toolbar. The add-in will then be added to the current model under **Global Definitions**. To add an add-in to the **Add-ins** list, select it in the **Add-in Libraries** window (see below). Once the add-in is available under **Global Definitions**, it works like any other node with a **Settings** window. Depending on its contents, it will affect the model by some functionality contained in the add-in.

The Add-in Libraries Window

The **Add-in Libraries** window contains available add-ins, which you can browse and add to the **Model Builder** by selecting the check box for the add-in and then click **Done**. This way, you enable the add-in so that it becomes available from the **Add-ins** menu. When you choose the add-in from that menu, it is imported and the selected form is added or a selected method is run. You can also make add-ins available by importing them without adding any form or running any methods. To do so, right-click an add-in and choose **Import Add-in**. Imported add-ins are also available from the **Add-ins** menu. If the add-in includes documentation, click the **Open PDF Document** button underneath the add-in description, or right-click the add-in and choose **Open PDF Document**.

Click the **Refresh** button ( ) to update the list of add-ins. You can also add separate user-defined add-in libraries (see below).

ADD-IN LIBRARY PREFERENCES

The following settings can be modified using the buttons at the bottom of the **Add-in Libraries** tree on the **Add-in Libraries** page in The Preferences Dialog Box and — if the **Allow managing libraries in the Add-in Libraries window** check box on that page is selected (the default) — also in the **Add-in Libraries** window itself.

Add User Add-in Library

Click the **Add User Add-in Library** button ( ) to add customized folders. In the **Add User Add-in Library** dialog box, navigate to a location on your computer and select an existing directory or click **Make New Folder** to create a custom folder. Click **OK** to save the changes and exit, or **Cancel** to exit without saving. The user-defined add-in library appears alongside the built-in add-in libraries,

> It is not possible to add an add-in library identical to, containing, or being contained in, an already used add-in library.

Optionally, you can replace the standard folder icon ( ) with custom icons of your choice that reflect the content of your library folders. To use a custom icon for a folder, create a PNG-file with an image size of 16-by-16 pixels and save it in the folder under the name `folder.png`.

Set the COMSOL Add-in Libraries Root

Click the **Set COMSOL Add-in Libraries Root Directory** button ( ) to edit or set the root folder. This redirects the COMSOL software to a different folder where customized add-ins can be stored.

In the **Set COMSOL Add-in Libraries Root Directory** dialog box, navigate to the new root folder location or click **Make New Folder**. Click **OK** to save the changes and exit, or **Cancel** to exit without saving.

Remove Selected Library

This button is enabled after a user add-in library folder has been created. Select a user add-in library root folder in the **Add-in libraries** tree and then the click the **Remove Selected** ( ) button to remove the library from the tree.

Comparing Models and Applications

It can be useful to compare two versions of the same model or application to get an overview of the differences between the two versions. To do so, click the **Compare** button ( ) in the **Compare** section of the **Developer** toolbar. A **Select Application** window then opens, where you can select the Model MPH-file (the remote file) to which you want to compare the current model in the COMSOL Desktop (the local file). A comparison then starts, and the results, if there are any differences, appear as a tree in the **Comparison Result** window. The comparison tool compares all the settings in the entire model, including visible and invisible settings in the Model Builder and the Application Builder.

THE COMPARISON RESULT WINDOW

At the top of the **Comparison Result** window there is a toolbar with the following buttons:

- **New Comparison** ( ), to make a new comparison. A **New Comparison** dialog box opens, where the default is that the **Use open file** check box is selected. Clear it to specify another **Local file**. Specify a **Remote file** and then click **OK** to perform a new comparison.

- **Update** ( ), to update the comparison
- **Collapse All** ( ) and **Expand All** ( ), to collapse or expand all branches in the comparison tree
- **Load Comparison from File** ( ), to load the results from another comparison from an XML-file.
- **Save Comparison to File** ( ), to save the results from the current comparison to an XML-file.
- **Show Next Difference** ( ), to move to the next difference in the tree with nodes that differ under **Differences**.
- **Show Previous Difference** ( ), to move to the previous difference in the tree with nodes that differ under **Differences**.

Under **Comparing files**, you find the file paths to the local model and the remote model. The local model is typically the model open in the COMSOL Desktop, and it can have been modified after the last save. The left column is called **Local file (Open application)** when the local model is the opened model or application. When the local model is a new model that has not yet been saved, the left column is called **Open application**.

Under **Differences**, you can use the **Filter results** list to control filtering with the options **No filter**, **Exclude all matching labels** (the default), and **Include all matching labels**. The entries in the **Regular expression** field define what node labels to filter. You can make the filtering case sensitive by selecting the **Case Sensitive** button ( ) box next to the list. The filter matching is done with regular expressions, and any label that contains a matching text will be either included or excluded. Filtering can be disabled by choosing **No filter** from the **Filter results** list. Click the **Show Only Active** button ( ) next to the list to exclude inactive settings from the comparison. Such settings are not actively used in the current model state and do not typically appear in the COMSOL Desktop.

The tree of nodes with differences has a structure according the structure of the underlying model object, not the Model Builder or Application Builder trees. This means that the hierarchy can be somewhat different compared to the hierarchy seen in the **Model Builde**r and **Application Builder** windows. The comparison includes all nonbinary differences in the model object that can be saved to disk, with some exceptions. For nodes that correspond to a node in the Model Builder or Application Builder trees, double-click the node (or right-click and choose **Go to Source**) in the tree of nodes with differences to display the corresponding node in one of those trees in the local file. When applicable, you can also right-click a node and choose **Go to Remote Source** to display the corresponding node in one of those trees in the remote file.

Below is a list of consequences of this design:

- The tree can show differences not visible in the Model Builder because they come from inactive settings hidden by another setting.
- Some settings and nodes in the Model Builder are not saved in the model object and will therefore not show in a comparison (Mesh Statistics, for example).
- No binary data are compared, so, for example, no differences in the solution will trigger a difference in the comparison.

Selecting a node in the tree will update the table under **Comparing values** below the tree. This table shows the attributes of the selected node, both in the local model (**Local value** column) and the remote model (**Remote value** column). This table is particularly interesting for the nodes labeled **Attributes differ** because they show the values that differ. Parent nodes can also show the same differences but typically with a larger number of attributes that have the same local and remote values. For differences that contain long entries or large arrays, there is an option to show a more detailed difference in a separate window for a selected table. Either double-click the table row or click the **Detailed Comparison of Selected Attribute** button ( ) below the table to open the **Compare Attribute** window.

Working with Nodes in the Model Builder

This section describes how you can move, copy, paste, duplicate, disable and enable nodes in the model tree that describes the contents of a model in the **Model Builder** window. There is also information about undoing and redoing operations in the model tree.

Moving Nodes in the Model Builder

Many of the nodes under the branches and subbranches listed in Table 3-2 can be moved around in the model tree. To move nodes use one of these methods:

- Select one node at a time (or by Ctrl-clicking or Shift-clicking to select more than one node at a time), and use the mouse to drop them in another applicable position in the model tree. A horizontal line indicates where in the model tree the moved (or copied) nodes get inserted when releasing the mouse.
- Right-click the selected nodes and select **Move Up** ( ) or **Move Down** ( ). Nodes that are the first or last of its kind can only be moved down or up, respectively.
- Use the keyboard shortcuts Ctrl+up arrow or Ctrl+down arrow to move nodes up or down.
- You can also create custom grouping of nodes to organize the nodes in the model tree. See Custom Grouping of Nodes.

For physics interface nodes it is not possible to move the default nodes (for the default boundary condition, for example). It is possible to create a copy of a default node, which initially has no selection. To click-and-drag a default node creates a copy whether or not the Ctrl key is pressed.

The order of the nodes in some of the branches affects the evaluation of the sequence that they define. In the following branches and subbranches it is possible to move nodes up and down to control the evaluation of the sequence or the order in which they appear within the branch or subbranch (also see Table 3-2):

- Definitions: nodes can be moved relative to other nodes of the same type (functions, selections, and so on).
- Geometry: The Geometry Nodes.
- Materials: Material nodes.
- Mesh: Mesh nodes (see Meshing).
- Physics interfaces: except for the default nodes, the nodes for physics interfaces (such as material models, boundary conditions, domains, edges, points, and sources) can be moved within the physics interface branches (see The Physics Interfaces).
- Study: Study and study step nodes can be moved (see Study and Study Step Types).
- Results: the order of the nodes can be rearranged within each of the subbranches (Derived Values, Tables, Plot Groups, Export, and Reports). Exceptions under the Export node are the Plot, Mesh, and Table nodes (see Results Analysis and Plots).

Copying, Pasting, and Duplicating Nodes

It is possible to copy and paste many of the nodes in the Model Builder to create additional nodes with identical settings or to paste it into another model. That is, you can paste the node into a model in a new COMSOL Desktop session. Some nodes can also be duplicated underneath the original node. You can also move, copy, and duplicate nodes using "drag-and-drop" of nodes in the Model Builder.

Duplicate ( ) is a convenient way to copy and paste in one step. In other words, it combines the **Copy** and **Paste** functions. When nodes are duplicated, the COMSOL Multiphysics software adds identical nodes underneath the original nodes on the same branch. You can duplicate most but not all nodes.

Nodes that can be copied (and duplicated) include the following:

- Functions, which are possible to copy from one **Definitions** or **Global Definitions** branch to another. Also see Functions and Global Definitions, Geometry, Mesh, and Materials.
- Variable utilities. Also see Matrices and Matrix Operations.
- Moving Mesh and Deformed Geometry nodes. See also Deformed Geometry and Moving Mesh.
- Material nodes. Also see Materials.
- Nodes for lighting and hiding in the **View** branch. Also see User-Defined Views.
- Physics and multiphysics feature nodes, which can be copied within the same physics interface or to another identical physics interface. You can also copy an entire physics interface to another model, for example. Also see The Physics Interfaces.
- Geometry sequences, for which there are two ways to copy and paste geometry objects. Using the Transforms>Copy operation (that keeps the nodes linked to one another), or a standard copy and paste (see Copying and Pasting Geometry Objects). It is also possible to copy, paste, and duplicate nodes corresponding to operation features, such as the Union node.
- Study steps, which are possible to copy from one **Study** branch to another. Also see Studies and Solvers.
- Derived value nodes. Also see Derived Values, Evaluation Groups, and Tables.
- Export nodes. Also see Exporting Data and Images.
- Plot nodes, which are possible to copy from one plot group to another, and their attributes (subnodes). Also see Plot Groups and Plots.

The copied object must be pasted into a model component with the same space dimension. For example, a Sphere can only be pasted into a 3D model.

HOW TO COPY, PASTE, OR DUPLICATE NODES

- On the Quick Access Toolbar (Windows users) or from the main **Edit** menu (macOS and Linux users), click **Copy** ( ), **Paste** ( ), or **Duplicate** ( ).
- Right-click a node and select **Copy**, **Paste**, or **Duplicate**.
- To paste a node, and after selecting **Copy**, click the parent node and right-click to select **Paste Heat Flux** to paste a copied node (a Heat Flux node in this case) to the parent node's branch.

- Create a copy of a node by Ctrl-clicking it and dragging a copy to an applicable location. A small plus sign at the cursor indicates that you drag a copy of the selected node.
- Ctrl-click and drag a duplicate to an applicable location. A small plus sign at the cursor indicates that you drag duplicates of the selected nodes.

Some nodes, typically default nodes in some physics interfaces, are so-called singleton nodes; that is, there can only be one such node in one and the same physics interface, and it may in some cases be created automatically. Such nodes, such as the **Gravity** node in fluid-flow interfaces when the license includes the CFD Module or Heat Transfer Module, when copied, are not possible to paste into the model.

Undoing and Redoing Operations

Undo is not possible for nodes that are built directly, such as geometry objects, solutions, meshes, and plots.

It is possible to undo the last operation for operations like adding, disabling, moving, and deleting nodes in the **Model Builder** as well as changing values in the **Settings** window. You can undo or redo several successive operations.

To undo the last operation or redo an undone operation:

- On the Quick Access Toolbar (Windows users) or from the main **Edit** menu (macOS and Linux users), select or click **Undo** ( ) or **Redo** ( ).
- Press Ctrl+Z (undo) or Ctrl+Y (redo).

- Copying, Pasting, and Duplicating Nodes
- Clearing Sequences and Deleting Sequences or Nodes
- Disabling or Enabling Nodes

Going to the Source Node

In the **Settings** window for many nodes, other nodes can be referenced in the model tree such as a component, solution, study or study step, or dataset, which provide data to the node where they are referenced.

Nodes where you refer to other nodes include plot groups, datasets, and solvers; in such nodes' **Settings** windows, click the **Go to Source** button ( ) to move to the node that the selection in the list next to the button refers to.

- Settings and Properties Windows for Feature Nodes
- Studies and Solvers
- Results Analysis and Plots

Clearing Sequences and Deleting Sequences or Nodes

You can change the contents, and actions, of the sequences in the model tree by clearing a meshing sequence or solution under a solver configuration, or delete nodes in the Model Builder.

!	**Undo** is not possible for nodes that are built directly, such as geometry objects, meshes, solutions, and plots.

CLEAR OR DELETE A MESH

Use a **Clear** function to keep the nodes and be able to recreate the mesh by rebuilding the meshing sequence.

Under the **Component** node where you want to clear or delete the mesh:

- In the **Mesh** toolbar, click **Clear Mesh** ( ) or right-click the **Mesh** node and select **Clear Mesh** ( ).
- To delete a meshing sequence, in the **Mesh** toolbar click **Delete Sequence** ( ) or right-click the **Mesh** node and select **Delete Sequence** ( ).
- If you have a model geometry with several meshes, you can clear all meshes at the same time. From the **Mesh** toolbar, click **Clear All Meshes** ( ).

CLEAR OR DELETE A SOLUTION

Use a **Clear** function to keep the nodes and be able to recreate the solution by computing the solution again.

- To clear a set of solutions under a specific study, from the **Study** toolbar, click **Clear Solutions** ( ) or right-click the **Study** node and select **Clear Solutions** ( ).
- To delete all solver nodes, right-click the **Solver Configurations** node and select **Delete Configurations** ( ). You can also choose whether or not to remove the Results nodes (datasets and plots, for example) associated with the solver configuration.
- If you have a model geometry with several studies, you can clear all solutions in all studies at the same time. From the **Study** toolbar, click **Clear All Solutions** ( ).

DELETE NODES

- To delete selected nodes, right-click the nodes and select **Delete** ( ) or press Del (the Delete key). Confirm the deletion of nodes for it to take effect. Also see Clear or Delete a Mesh.
- To delete a geometry sequence, in the **Geometry** toolbar click **Delete Sequence** ( ) or right-click the **Geometry** node and select **Delete Sequence** ( ). You cannot use the **Undo** command.
- To delete geometry objects or entities, in the **Geometry** toolbar click **Delete** ( ) or right-click **Geometry** and select **Delete** ( ). Or select objects in the **Graphics** window, and click the **Delete** button ( ) in the **Graphics** window toolbar.

If you use the **Delete** button to delete objects, the software deletes the selected objects that correspond to primitive features by deleting their nodes from the geometry sequence. If you delete objects that do not correspond to primitive features or if you delete geometric entities, a **Delete Entities** node appears in the sequence.

Disabling or Enabling Nodes

A disabled node does not take part in the evaluation of a sequence; see Figure 3-6. Some nodes, such as container nodes and default nodes in the physics interfaces (see Physics Interface Default Nodes), cannot be disabled (or deleted). When this is the case, the context menu does not have these options available. You can use Shift-click and Ctrl-click to select multiple nodes that you want to delete, disable, or enable.

- To disable selected nodes, right-click and select **Disable** ( ) or press F3. The nodes are unavailable (dimmed) in the model tree to indicate that they are disabled. For a geometry or meshing sequence, disabled nodes do not affect the finalized geometry or mesh.
- To enable disabled nodes, right-click and select **Enable** ( ) or press F4.

Instead of disabling and enabling variables and physics nodes to simulate different analysis cases (using different boundary conditions or sources, for example), use the selection of variables and physics interfaces in the study steps' **Physics and Variables Selection** sections, or use *load cases* for solving cases with varying loads or constraints. See Physics and Variables Selection and Using Load Cases.

Modeling Guidelines

To model large-scale problems and for successful modeling in general, the COMSOL Multiphysics software makes it possible to tune solver settings and to use symmetries and other model simplifications to reach a solution or — failing that — interrupt the solution process to retrieve a partial solution. This section provides some tips and guidelines when modeling.

Selecting Physics Interfaces

When creating a model in COMSOL Multiphysics, you can select a single physics interface that describes one type of physics or select several physics interfaces for multiphysics modeling and coupled-field analyses.

MODELING USING A SINGLE PHYSICS INTERFACE

Most physics interfaces contain Stationary, Eigenvalue, and Time Dependent (dynamic) study types. As already mentioned, these physics interfaces provide features and windows where you can create models using material properties, boundary conditions, sources, initial conditions, and so on. Each physics interface comes with a template that automatically supplies the appropriate underlying equations.

If you cannot find a physics interface that matches a given problem, try one of the interfaces for PDEs, which makes it possible to define a custom model in general mathematical terms using equation-based modeling. Indeed, the COMSOL Multiphysics software can model virtually any scientific phenomena or engineering problems that originate from the laws of science.

MULTIPHYSICS MODELING USING MULTIPLE PHYSICS INTERFACES

When modeling real-world systems, you often need to include the interaction between different kinds of physics: *multiphysics*. For instance, an electric current produces heat, and the properties of an electronic component such as an inductor vary with temperature. To solve such a problem, combine two or several physics interfaces into a single model using the program's multiphysics capabilities. For the example just mentioned, you can use the predefined Joule Heating multiphysics coupling, which is a combination of the Electric Currents and Heat Transfer interfaces. This way you create a system of two PDEs with two dependent variables: V for the electric potential and T for the temperature. There are many other predefined multiphysics couplings that combine two or more coupled physics interfaces for common multiphysics applications. If you have added physics interfaces for which predefined multiphysics couplings exist, they are available in the **Add Multiphysics** window (see The Add Multiphysics Window).

You can also combine physics interfaces and equation-based modeling for maximum flexibility.

To summarize the proposed strategy for modeling processes that involve several types of physics: Look for physics interfaces suitable for the phenomena of interest. If you find them among the available physics interfaces, use them; if not, add one or more interfaces for equation-based modeling.

When coupling multiple physics interfaces in a multiphysics model (without using a predefined multiphysics interface), the couplings can occur in domains and on boundaries. The COMSOL Multiphysics software recognizes some common multiphysics couplings, which then appear under the **Multiphysics** node. The program also automatically identifies potential *model inputs* for quickly forming couplings between physics interfaces. For example, a velocity field from fluid flow is a model input for convective heat transport in heat transfer. In that case, the model input automatically transfers the velocity field from the fluid to the heat transfer part.

Multiphysics Modeling Workflow

Using Symmetries

By using symmetries in a model you can reduce its size by half or more, making this an efficient tool for solving large problems. This applies to the cases where the geometries and modeling assumptions include symmetries.

The most important types of symmetries are axial symmetry, symmetry planes or lines, and antisymmetry planes or lines:

- *Axial symmetry* is common for cylindrical and similar 3D geometries. If the geometry is axisymmetric, there are variations in the radial (r) and vertical (z) direction only and not in the angular (θ) direction. You can then solve a 2D problem in the *rz*-plane instead of the full 3D model, which can save considerable memory and computation time. Many physics interfaces are available in axisymmetric versions and take the axial symmetry into account. During postprocessing, you can revolve the 2D axisymmetric solution to view the results in 3D.
- *Symmetry and antisymmetry planes or lines* are common in both 2D and 3D models. *Symmetry* means that a model is identical on either side of a dividing line or plane. For a scalar field, the normal flux is zero across the symmetry line. In structural mechanics, the symmetry conditions are different. *Antisymmetry* means that the loading of a model is oppositely balanced on either side of a dividing line or plane. For a scalar field, the dependent variable is 0 along the antisymmetry plane or line. Structural mechanics applications have other antisymmetry conditions. Many physics interfaces have symmetry conditions directly available as nodes that you can add to the model tree.

To take advantage of symmetry planes and symmetry lines, all of the geometry, material properties, and boundary conditions must be symmetric, and any loads or sources must be symmetric or antisymmetric. You can then build a model of the symmetric portion, which can be half, a quarter, or an eighth of the full geometry, and apply the appropriate symmetry (or antisymmetry) boundary conditions.

Effective Memory Management

Especially in 3D modeling, extensive memory usage requires some extra precautions. First, check that you have selected an iterative linear system solver. Normally you do not need to worry about which solver to use because the physics interface makes an appropriate default choice. In some situations, it might be necessary to make changes to the solver settings and the model. For details about solvers, see the Studies and Solvers chapter.

ESTIMATING THE MEMORY USE FOR A MODEL

Out-of-memory messages can occur when the COMSOL Multiphysics software tries to allocate an array that does not fit sequentially in memory. It is common that the amount of available memory seems large enough for an array, but there might not be a contiguous block of that size due to memory fragmentation.

In estimating how much memory it takes to solve a specific model, the following factors are the most important:

- The number of node points
- The number of dependent and independent variables
- The element order
- The sparsity pattern of the system matrices. The sparsity pattern, in turn, depends on the shape of the geometry and the mesh but also on the couplings between variables in a model. For example, an extended ellipsoid gives sparser matrices than a sphere.

The MUMPS and PARDISO out-of-core solvers can make use of available disk space to solve large models that do not fit in the available memory.

You can monitor the memory use in the lower-right corner of the COMSOL Desktop, where the program displays the amount of physical memory and total virtual memory used (see Information About Memory Use).

CREATING A MEMORY-EFFICIENT GEOMETRY

A first step when dealing with large models is to try to reduce the model geometry as much as possible. Often you can find symmetry planes and reduce the model to half, a quarter, or even an eighth of the original size. Memory usage does not scale linearly but rather polynomially (Cn^k, $k > 1$), which means that the model needs less than half the memory if you find a symmetry plane and cut the geometry size by half. Other ways to create a more memory-efficient geometry include:

- Avoiding small geometry objects where not needed and using Bézier curves instead of polygon chains.
- Using linear elements if possible (this is the default setting in some physics interfaces). See Selecting an Element Type.
- Making sure that the mesh elements are of a high quality. Mesh quality is important for iterative linear system solvers. Convergence is faster and more robust if the element quality is high.
- Avoiding geometries with sharp, narrow corners. Mesh elements get thin when they approach sharp corners, leading to poor element quality in the adjacent regions. Sharp corners are also unphysical and can lead to very large (even infinite, in theory) stress concentrations.

INFORMATION ABOUT MEMORY USE

In the lower-right corner of the COMSOL Desktop is information about how much memory the COMSOL Multiphysics software is currently using. The two numbers in Figure 3-13 displayed as **1.24 GB | 1.26 GB** represent the physical memory and the virtual memory, respectively. If you position the cursor above these numbers, the tooltip includes the numbers with the type of memory explicitly stated:

- The **Physical memory** number is the subset of the virtual address space used by COMSOL Multiphysics that is physically resident; that is, it is the amount of physical memory (RAM) in "active" use.
- The **Virtual memory** number is the current size of the virtual address space that the COMSOL Multiphysics software uses.

Figure 3-13: An example of memory use displayed in the COMSOL Desktop.

Selecting an Element Type

As the default element type for most physics interfaces and features, the COMSOL Multiphysics software uses first-order or second-order Lagrange elements (shape functions). Second-order elements and other higher-order elements add additional degrees of freedom on midpoint and interior nodes in the mesh elements. These added degrees of freedom typically provide a more accurate solution but also require more memory due to the reduced sparsity of the discretized system and the added number of degrees of freedom (DOFs). For many application areas, such as stress analysis in solid mechanics, the increased accuracy of a second-order element is important because quantities such as stresses involve space derivatives and become constant within an element when using first-order elements.

COMSOL recommends that you use the default element types. For some applications, it might be possible to use a lower-order element than the default element type, but you must then use care to ensure that the important quantities are resolved.

For information about editing shape functions, see Equation View.

Analyzing Model Convergence and Accuracy

It is important that the numerical model accurately captures local variations in the solution such as stress concentrations. In some cases you can compare your results to values from handbooks, measurements, or other sources of data. Many Applications Libraries examples are *benchmark models* that include comparisons to established results or analytical solutions.

If a model has not been verified by other means, a *convergence test* is useful for determining if the mesh density is sufficient. Here you refine the mesh and run the study again, and then check if the solution is converging to a stable value as the mesh is refined. If the solution changes when you refine the mesh, the solution is mesh dependent, so the model requires a finer mesh. You can use adaptive mesh refinement, which adds mesh elements based on an error criterion, to resolve those areas where the error is large. See the “Stresses and Strains in a Wrench” model in the *Introduction to COMSOL Multiphysics* book for an example of a convergence test.

For convergence, it is important to avoid singularities in the geometry.

Avoiding Singularities and Degeneracies in the Geometry

Achieving Convergence When Solving Nonlinear Equations

Nonlinear problems are often difficult to solve. In many cases, no unique solution exists. The COMSOL Multiphysics software uses a Newton-type iterative method to solve nonlinear systems of PDEs. This solution method can be sensitive to the initial estimate of the solution. If the initial conditions are too far from the desired solution, convergence might be impossible, even though it might be simple from a different starting value.

You can do several things to improve the chances for finding the relevant solutions to difficult nonlinear problems:

- Provide the best possible initial values.
- Solve sequentially and iterate between single-physics equations; finish by solving the fully coupled multiphysics problem when you have obtained better starting guesses.
- Ensure that the boundary conditions are consistent with the initial solution and that neighboring boundaries have compatible conditions that do not create singularities.
- Refine the mesh in regions of steep gradients.
- For convection-type problems, introduce artificial diffusion to improve the numerical properties. Most physics interfaces for modeling of fluid flow and chemical species transport provide artificial diffusion as part of the default settings.
- Scaling can be an issue when one solution component is zero. In those cases, automatic scaling might not work.
- Turn a stationary nonlinear PDE into a time-dependent problem. Making the problem time-dependent generally results in smoother convergence. By making sure to solve the time-dependent problem for a time span long enough for the solution to reach a steady state, you solve the original stationary problem.

- Use the parametric solver and vary a material property or a PDE coefficient starting from a value that makes the equations less nonlinear to the value at which you want to compute the solution. This way you solve a series of increasingly difficult nonlinear problems. The solution of a slightly nonlinear problem that is easy to solve serves as the initial value for a more difficult nonlinear problem.
- The residual operator can provide insight into the location and development of the algebraic residual in models with convergence issues.

- Stabilization Techniques
- Convergence Plots
- Introduction to Solvers and Studies

Avoiding Strong Transients

If you start solving a time-dependent problem with initial conditions that are inconsistent, or if you use boundary conditions or sources that switch instantaneously at a certain time, you induce strong transient signals in a system. The time-stepping algorithm then takes very small steps to resolve the transient, and the solution time might be very long, or the solution process might even stop. Stationary problems can run into mesh-resolution issues such as overshooting and undershooting of the solution due to infinite flux problems.

Unless you want to know the details of the transients, start with initial conditions that lead to a consistent solution to a stationary problem. Only then turn on the boundary values, sources, or driving fluxes over a time interval that is realistic for your model.

In most cases, turn on your sources using a smoothed step over a finite time. What you might think of as a step function is, in real-life physics, often a little bit smoothed because of inertia. The step or switch does not happen instantaneously. Electrical switches take milliseconds, and solid-state switches take microseconds.

- Introduction to Solvers and Studies
- Stationary and Time Dependent

Physics-Related Checks and Guidelines

There are some important checks and guidelines that primarily apply to different areas of physics. Making these checks ensures that the model input is sufficient and increases the chances for successful modeling. See also the modeling sections of the documentation for the physics interfaces and the modules for more information related to modeling different physics.

FLUID FLOW AND TRANSPORT PHENOMENA

The following checks and guidelines primarily apply to fluid-flow modeling but also to modeling of other transport phenomena:

- If none of the boundary conditions include the pressure (most outlet conditions do, however), then you should specify the pressure at some point in the fluid domain. Without a specified pressure, the problem is underconstrained and it is difficult to get convergence.
- Make sure that the mesh if sufficiently fine, so that it contains at least 4–6 mesh elements across the thickness of a channel, for example.
- Make sure that the boundary conditions and the initial conditions match for time-dependent problems. For example, instead of starting with a full velocity on the wall, compared to a zero initial velocity field in the fluid,

ramp up the velocity with a smoothed step function or a ramp function that takes the inlet velocity from zero, which matches the initial value for the velocity field, to the full velocity. See Avoiding Strong Transients.

- For fluid-flow models it is important to estimate the flow regime (laminar or turbulent) using the Reynolds number, for example. If the flow is in the turbulent regime, a turbulence model is typically required.

ACOUSTIC, STRUCTURAL, AND ELECTROMAGNETIC WAVE PROPAGATION

For models that describe wave propagation, it is important to fully resolve the wave in both time and space. In practice that means using a maximum mesh element size that provides about 10 linear or five second-order elements per wavelength and also, for transient simulations, a fixed time step that is small enough.

STRUCTURAL MECHANICS

The following checks and guidelines primarily apply to modeling of structural mechanics:

- Make sure that the model is fully constrained. At a minimum, you typically need to constrain the model to avoid all rigid-body movement, which for a 3D solid mechanics model means 6 constraints for three translations and three rotations. Otherwise, the solution is not well defined and does not converge.

  The structural mechanics interfaces include a Rigid Motion Suppression feature, which eliminates all rigid-body movement. If you do not use that feature, it is not possible to add all 6 constraints in a single point, where you can constrain at most three translational degrees of freedom. For a 3D solid model you can use a "3–2–1 approach" to constrain 3 degrees of freedom at one point (a fixed constraint), 2 at another point, and 1 at a third point. To do so, select three convenient points (vertices) that are well separated. Then fix the first point in all three directions. Constrain the second point in the two directions orthogonal (normal) to the vector from point one to point two making sure that there is no restriction to deformation along the line from point one to point two. Finally, constrain the third point in a direction normal to the plane formed by the three points. To test this approach, the model should expand or contract under temperature loading and have small stresses throughout with no stress concentrations. The corresponding minimum constraints for a 2D model are a fixed constraint at one point for the 2 translational degrees of freedom and an additional constraint in one direction at another point to constrain the single rotational degree of freedom.
- Consider if you can assume that the material is linear elastic and that the deformations are small. If not, consider using a nonlinear material model.
- Avoid sharp corners in the geometry, which are unphysical and lead to unbounded stress concentrations.

Results With Unphysical Values

WHERE AND WHY DO UNPHYSICAL VALUES APPEAR?

In some models small unphysical values can occur due to numerical artifacts or other model-related reasons. Examples include:

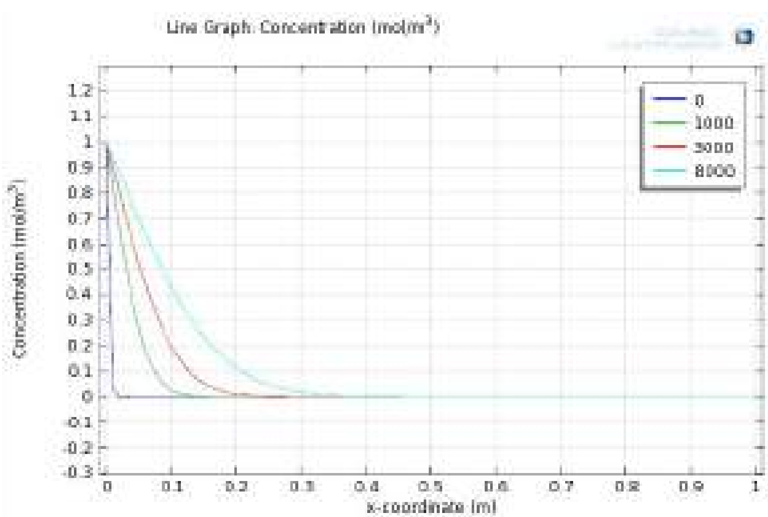
- Negative concentrations in mass transfer.
- A temperature that is slightly higher than the initial condition in time-dependent heat transfer studies.
- Small reaction forces that appear in unloaded directions in structural mechanics models.
- Small negative gaps in a contact analysis.
- Small negative effective plastic strain values.
- Stresses above the yield limit for an ideally plastic material in solid mechanics.

Some reasons for why these unphysical values occur:

- Numerical noise is a common cause. When the values of the dependent variables approach zero, the numerical noise can become relatively significant and cause some of the results to be slightly negative even if that is not physically possible.
- Interpolation and extrapolation of values can cause some values to become unphysical. Take care when using interpolated data or a piecewise polynomial function to define a temperature-dependent material property, for example. If you allow extrapolation outside of the defined range of input values, the material property values may not be valid. Also, results for an elastoplastic material are correct (within some tolerance) at the integration points (Gauss points) inside the finite elements, but values might become unphysical when extrapolating the data to the element boundaries.

	The Plasticity feature is available as a subnode to a Linear Elastic Material with the Nonlinear Structural Materials Module.

- Discontinuities in the model is another source of, for example, small negative concentrations due to a discontinuous initial value. With an initial value that is zero along a boundary for convective transport models, for example, the physical interpretation is an initially sharp, gradually diffusing front moving away from the boundary. However, for the default shape function (second-order Lagrange elements), only continuous functions are admissible as solutions. COMSOL then modifies the discontinuous initial value before the time stepping can begin. This often results in a small dip in the solution at the start time. In the example model that the following figure shows, the concentration is locally slightly negative at $t = 0$:



- Lack of mesh resolution is another cause of unphysical values such as negative concentrations. The resulting convergence problems are often the underlying issue when negative concentrations are observed in high convection regimes (high Peclet number) and in those with large reaction terms or fast kinetics (high Damkohler number).
- Incorrect physics in the model can also cause these types of problems. For mass transfer, for example, the use of a constant sink in a reaction term is an approximation that only works for large concentrations. When the concentration reaches zero, the reaction term continues to consume the species, finally resulting in a negative concentration.

AVOIDING UNPHYSICAL VALUES

This section contains some ways to avoid computing or displaying unphysical values:

- In some cases it is possible to add a baseline to the dependent variable so that the numerical noise does not affect the solution in the same way as when the values of the dependent variable approach zero. This scaling is not possible with, for example, a reaction term that depends on the concentration because then the scale and origin do matter.
- Avoid discontinuities in the model using, for example, smoothed step functions.
- Formulate logarithmic variables as a way of eliminating mesh resolution problems and negative dips using the logarithm of the original dependent variable (the concentration, for example) as the dependent variable. The reason for this is that a linearly varying mesh sometimes does not capture the exponential behavior of the changes in the dependent variable. Modeling the logarithm of the dependent variable also ensures that the real concentration, for example, cannot become negative during the solution process.
- Avoid displaying small unphysical values due to numerical noise by clipping the values for the plot. You can do this by plotting, for example, `c*(c>0)` instead of `c`, which evaluates to 0 everywhere where c is smaller than 0. You can also adjust the range of the plot data and colors to only show nonnegative values. Parts of the plots where values are outside the range then become empty.
- It can also be useful to check how the mesh affects the solution by refining the mesh and checking if the problem with unphysical values gets better or worse. If it gets better, then continue to refine the mesh. If it gets worse, you probably need to check the physics of the model.

Multiphysics Modeling Workflow

The ability to create multiphysics models — those with more than one type of physics or equation such as coupled-field problems — is one of the most powerful capabilities of COMSOL Multiphysics. In such models, the software can solve all the equations, taken from various areas of physics, as one fully coupled system.

Within the COMSOL software you can choose from several ways to work with multiphysics modeling and coupled-field analysis, including predefined multiphysics interfaces, predefined multiphysics couplings, and setting up user-defined multiphysics couplings using model inputs or expressions that include dependent variables or other expressions from another physics interface.

In this section:

- Creating a Multiphysics Model
- Advantages of Using the Predefined Multiphysics Interfaces
- The Add Multiphysics Window
- The Multiphysics Branch
- Uncoupling a Multiphysics Coupling
- Model Inputs and Multiphysics Couplings

Creating a Multiphysics Model

There are two ways to create and use the available predefined multiphysics couplings: using Predefined Multiphysics Interfaces or Adding Predefined Multiphysics Couplings to Physics Interfaces using The Add Multiphysics Window or in The Multiphysics Branch. You can also create multiphysics couplings, in the physics interface settings, using a model input or by directly typing an expression using a dependent variable from another physics interface, for example (see Specifying Equation Coefficients and Material Properties for information about what you can include in such expressions).

PREDEFINED MULTIPHYSICS INTERFACES

The Joule Heating Interface is an example of a predefined multiphysics interface. Many other multiphysics interfaces are available depending on the products included in your COMSOL license. After **Joule Heating** is selected from The Model Wizard, the **Heat Transfer in Solids** interface, the **Electric Currents** interface, and a **Multiphysics** node, including the default feature applicable to the multiphysics coupling (**Electromagnetic Heating**), are displayed under the **Added physics interfaces** list as in Figure 3-14. Figure 3-15 shows you what is included in the Model Builder when a predefined multiphysics interface is added. Compare to Figure 3-16 where individual physics interfaces are added, and these features are initially accessible only from the context menu. There can also be moving mesh nodes added as part of a multiphysics interface; they then appear under **Definitions** in the **Added physics interfaces** list.

You can add physics interfaces when you start creating the model with The Model Wizard or at any time with The Add Physics Window.

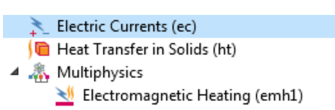


Figure 3-14: When Joule Heating is selected in the Model Wizard, the default physics interfaces and coupling feature are displayed under Added physics interfaces.

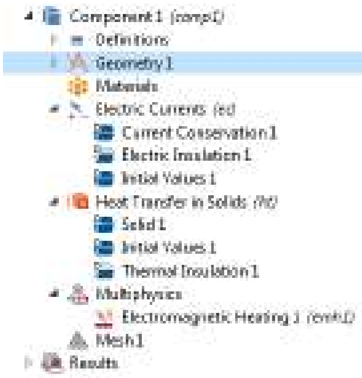


Figure 3-15: An example of what is added to the Model Builder when Joule Heating is selected in the Model Wizard. The Electromagnetic Heating feature is automatically included under the Multiphysics node.

ADDING PREDEFINED MULTIPHYSICS COUPLINGS TO PHYSICS INTERFACES

An empty **Multiphysics** node is added automatically when two (or more) physics interfaces are set up in a model and when there is the possibility to couple the physics interfaces. In other words, if you add physics interfaces one at a time, and the software identifies these physics interfaces as being of the multiphysics category, the **Multiphysics** node is automatically added to the Model Builder. The relevant predefined multiphysics coupling features are then available from the context menu (right-click the **Multiphysics** node) as well as from the **Physics** toolbar, in the **Multiphysics** menu. See Figure 3-16. You can also add predefined multiphysics couplings from the **Add Multiphysics** window (see The Add Multiphysics Window), which then adds all necessary multiphysics coupling nodes under the **Multiphysics** node. Using a workflow where you add physics interfaces and multiphysics couplings manually makes it possible to analyze and validate one physics at the time before solving the full multiphysics model.

Figure 3-16: An example of when a Multiphysics node is automatically added to the model. The specific multiphysics features are made available from the context menu (left) or (partially shown here) Physics toolbar, Multiphysics menu (right) based on the physics interfaces in the model. The difference when the predefined Joule Heating interface is added is that these features are included under Multiphysics and there are some modified settings automatically applied. In either case, the available features depend on the COMSOL products that the license includes.

Advantages of Using the Predefined Multiphysics Interfaces

One advantage of using the predefined multiphysics interfaces is that specific or modified settings are included with the physics interfaces and the multiphysics coupling features. If physics interfaces are added one at a time, followed by the predefined multiphysics coupling features, these modified settings are not automatically included.

For example, if you add single Electric Currents and Heat Transfer in Solids interfaces to the Model Builder, the COMSOL Multiphysics software adds an empty **Multiphysics** node. The applicable multiphysics couplings are then available as subnodes that you can add. If you instead choose the predefined Joule Heating multiphysics coupling, for example, which then adds the **Electromagnetic Heating** node under the **Multiphysics** node, so that you do not need to remember which multiphysics coupling nodes to add for a specific type of multiphysics. See The Add Multiphysics Window.

In general, it is useful to use any type of multiphysics coupling because you can turn multiphysics on and off (that is, enable and disable features), giving you more flexibility to test and observe multiphysics effects.

Even if you do not start with a predefined coupling, another benefit of this approach is that you are no longer constrained by the use of specific physics interfaces, nor do these have to be added in any specific order. The order in which physics interfaces are added does not matter for the end result.

An example of this is if you start modeling by adding a Heat Transfer in Solids interface. As you continue to build the model, you add an Electric Currents interface. At this stage of the process, you may have defined several boundary conditions, chosen materials, or experimented with other settings. You may have also solved the model successfully at this point and now you want to continue building on this design. The COMSOL Multiphysics software recognizes this and adds a **Multiphysics** node, which you can right-click to access and add any of the available predefined multiphysics couplings.

For multiphysics interfaces that consist of participating physics interfaces, the default solver settings use a segregated solver approach with one segregated step for each physics interface and each of these steps calling an iterative solver. These solver settings are suitable for large models, but if possible, a fully-coupled solver approach using direct solvers can be more robust. You can switch to such solver settings by right-clicking the **Study** node and choosing **Show Default Solver**. Then the solver nodes that the predefined multiphysics interface specifies appear under the **Solver Configuration** node, and you can right-click the solver node to add a **Fully Coupled** solver node to replace the **Segregated** node, for example.

For some multiphysics interfaces, a side effect of adding physics interfaces one at a time is that two study types — Frequency-Stationary and Frequency-Transient — are not available for selection until *after* at least one coupling feature is added. In this case, it is better to first add an **Empty Study**, then add the coupling features to the **Multiphysics** node, and lastly, right-click the **Study** node to add the study steps as needed.

The Add Multiphysics Window

The **Model Wizard** and the **Add Physics** window contain predefined multiphysics interfaces, which typically add two or more physics interfaces and some predefined multiphysics coupling features that define the multiphysics couplings between those physics interfaces. When building a model, it can sometimes be useful to start with a single physics before adding other physics and the multiphysics couplings that connect them. To add any applicable predefined multiphysics coupling in a model, open the **Add Multiphysics** window ( ) by right-clicking a **Component** node or from the **Physics** ribbon toolbar. The predefined couped multiphysics couplings that the selected physics interfaces support then appear in the tree. Choose the wanted multiphysics couplings and add them to the component under **Multiphysics** by pressing Enter, clicking the **Add to Component** button ( ), or right-clicking a multiphysics coupling and choosing **Add to Component**. The required multiphysics coupling nodes are then added to the model, and the participating physics interfaces are modified by setting the correct physics property values and adding any needed features for the selected multiphysics couplings.

You control which multiphysics couplings that appear using the settings under **Select the physics interfaces you want to couple**. You can clear and select all physics interfaces in the current component. By default, all physics interfaces are selected and appear with a check mark ( ) in the **Couple** column. The available multiphysics couplings depend on which COMSOL Multiphysics products your license includes. If no multiphysics coupling is available, **No Coupling Features Available for the Selected Physics Interfaces** appears. You must select at least two physics interfaces for any multiphysics couplings to appear. With more than two physics interfaces in the component, any combination of two or sometimes more physics interfaces typically results in a different set of available multiphysics couplings, whereas most combinations of three or more physics interfaces result in no available multiphysics couplings.

The existing **Studies** are listed under **Multiphysics couplings in study**. By default, the studies appear with a check mark ( ) in the **Solve** column, which indicates that the study solves for the equations that the multiphysics couplings add. Click in the column to clear the check mark and exclude the equations in the multiphysics coupling from that study. Some multiphysics couplings do not add any extra equations and are then not affected by this setting.

The Multiphysics Branch

The **Multiphysics** branch ( ) contains, or has available, any predefined multiphysics coupling features that are likely to be used as multiphysics couplings for a particular set of physics interfaces added to the Model Builder. See Figure 3-16. There are no settings required for the node itself.

Predefined multiphysics interfaces provide you with a quick entry point for common multiphysics applications. You can create the same multiphysics couplings using any of the other methods for multiphysics modeling, and you can continue to add, modify, disable, and remove physics features or interfaces in a model when you start using one of the predefined multiphysics interfaces. If you instead decide to add physics interfaces one by one, you can verify that each type of physics or equation gives the expected results before adding more complexity to the model by adding another physics interface, physics feature, or multiphysics coupling.

For links to more information about the add-on modules and the multiphysics interfaces available go to www.comsol.com/comsol-multiphysics.

Uncoupling a Multiphysics Coupling

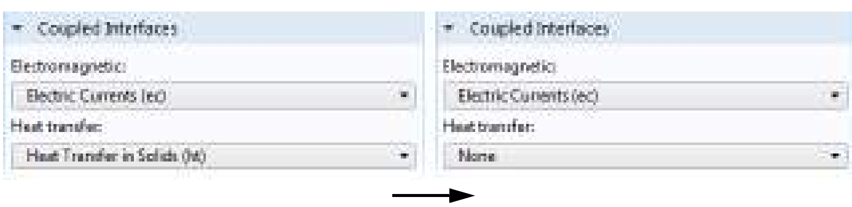


Figure 3-17: Uncoupling a predefined multiphysics coupling feature.

For each multiphysics coupling feature (for example, Electromagnetic Heating), there is a section that defines the physics interfaces involved in the multiphysics coupling. By default, the applicable physics interfaces are selected in the lists to establish the coupling.

You can also select **None** from the lists to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** (for example, if a **Heat Transfer in Solids** interface is deleted), then the **Heat transfer** list for the **Electromagnetic Heating** reverts to **None** (Figure 3-17) as there is no heat transfer interface to couple to. To avoid that the multiphysics coupling is turned off unintentionally, a warning **A coupled physics is set to 'None'** tooltip appears when you change a list under **Coupled Interfaces** to **None**.

Figure 3-18: A warning tooltip appears when a multiphysics coupling is turned off.

If a physics interface is deleted and then added to the model again, and in order to re-establish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present physics interface.

Model Inputs and Multiphysics Couplings

Model inputs can appear in an equation or material model node's **Model Inputs** section. Model inputs are typically fields such as temperature and velocities that act as inputs for material models and model equations, but they can be any available physical property. They appear in the **Model Inputs** section if a material is defined so that a material property becomes a function of the temperature, for example. The COMSOL Multiphysics software connects the model input to an existing field (dependent variable) within the physics interface (but not to available fields in other physics interfaces). Default model inputs are always available. You can define scalar values for default model inputs under **Global Definitions** for use throughout the model (see Default Model Inputs).

For frequently used multiphysics couplings, predefined multiphysics coupling nodes are available under the **Multiphysics** node (see The Multiphysics Branch). The following part of this section is mostly useful for cases when such predefined couplings are not available.

With more than one physics interface in the model, coupling of the fields is easy: all applicable fields that can serve as inputs in another physics interface automatically appear in the other physics interface's **Settings** window's **Model Inputs** section. For example, with a Heat Transfer in Fluids (ht) interface and a Laminar Flow (spf) interface, you can select **Velocity field (spf)**, which the **Fluid Properties 1 {fp1}** node in the **Laminar Flow** branch defines, from the **Velocity field** list in the **Model Inputs** section of the **Settings** window for the **Fluid** node under **Heat Transfer in Fluids**. The velocity field from the Laminar Flow interface then becomes the velocity field for the convective heat transfer. You can also choose **Common model input** to use its value to define the velocity field, or choose **User defined** to enter a user-defined velocity field.

When you have selected a model input from one of the lists, click the **Go to Source** button ( ) next to the list to move directly to the node in the other physics interface that provides the model input. If more than one node contribute to the model input, choose which one to go to in the **Contributing Entities** dialog box. Then click **OK** to move to the selected node.

You can also, if you have selected **Common model input**, click the **Create Model Input** button ( ) to create a local **Model Input** node in the current component (see Model Input) for defining a model input for some or all of the geometry in the current component, overriding the default model input.

By default, the **Model Input** section in the **Fluid Properties** node under the **Laminar Flow** node is empty. But if you, for example, add a temperature-dependent material property such as the dynamic viscosity, a **Temperature** list appears in the **Model Input** section where you can explicitly choose the **Temperature (ht)** field or use a user-defined temperature or a **Common model input** like any other model input.

A list in the **Model Inputs** section becomes unavailable if the physics itself defines the field because it is then automatically connected to that field. For example, with a Heat Transfer in Fluids (ht) interface the **Temperature** list is unavailable in the **Fluid** node under **Heat Transfer in Fluids**. This automatic connection selects the

Temperature (ht) field. As long as the list is unavailable, you cannot change it. If you want to use another temperature field or an expression, you first make the list editable by clicking the **Make All Model Inputs Editable** button ( ). Using this option can be useful in order to, perhaps temporarily, break a multiphysics couping and use a user-defined value instead to, for example, investigate a simulation that does not converge.

For this type of fluid-thermal coupling, the **Multiphysics** branch provides a predefined **Nonisothermal Flow** node, which provides an easy way to set up this coupling without having to explicitly specify the model input.

See *Joule Heating of a Microactuator* for an example of combining the Electric Currents and Heat Transfer in Solids interfaces through a Joule Heating multiphysics interface (Application Library path **COMSOL_Multiphysics/Multiphysics/thermal_actuator_jh**).

Specifying Model Equation Settings

The fundamental mathematical model, representing the physics in a physics interface, is contained in physics nodes with selection on the same space dimension as the physics itself. The first node under a physics branch is of this type and sets up default equations where the physics interface is active. These equations are controlled by specifying:

- Material properties, which COMSOL Multiphysics uses as coefficients in the equations
- A coordinate system, which makes it possible to specify anisotropic material properties and vectors in a more convenient coordinate system than the global Cartesian coordinate system.
- A material model (a mathematical model for a constitutive relation, for example), which selects an equation suitable for a given type of material

Not all physics features allow anisotropic materials or more than one material model. Therefore, these settings cannot be present.

The default node uses the same material model, and thus the same equations, everywhere. Material properties can vary between different parts of the feature's selection, if the property is specified as taken **From material**. Add additional nodes to use different material models for different parts of the geometry, or to use different **User defined** material property values.

In equation-based modeling, provided by the Mathematics branch interfaces, the form of the equation is fixed for each particular node type. Each given equation form contains a number of free PDE coefficients, which you can be specify in the settings to define the specific equation that you want to solve.

Equation-Based Modeling

Specifying Equation Coefficients and Material Properties

To specify an equation coefficient or a material property, enter a value or an expression directly in the corresponding field. The expressions in those fields are interpreted, providing the possibility to enter expressions that include variables and coordinates in addition to constants and numerical values. Such expressions can contain:

- Numerical values.
- Units (see Using Units).
- Built-in Mathematical and Numerical Constants.
- Spatial coordinates, time, and the dependent variables in any physics feature in the model as well as the spatial derivatives and time derivatives. Using such variables makes it possible to create user-defined multiphysics couplings.
- Physical Constants — built-in universal physical constants.
- User-defined parameters, variables, coupling operators, and functions, including external functions and MATLAB® functions (requires the COMSOL LiveLink™ *for* MATLAB®). See Operators, Functions, and Constants.
- Built-in functions and operators such as `d` and `mean`.

You can use these types of variables, constants, functions, and operators in all settings for the physics interfaces; many types of variables are also available anywhere in the model.

In most cases where you can enter an expression, you can press Ctrl+Space to choose from a number of applicable variables, parameters, functions, operators, and constants that you can insert into the expression at the position of the cursor.

Modeling Anisotropic Materials

Anisotropic materials respond differently to an excitation depending on its direction. Because excitations are generally vectors and the corresponding response is a vector density, material properties are usually rank-2 tensor densities. For example, the following material properties are anisotropic tensor densities: diffusion coefficient, permittivity, thermal conductivity, and electrical conductivity.

These properties are, in principle, specified in matrix form and defined by their components in the coordinate system selected in the node settings. At most four components are used in 2D and at most nine components in 3D. When the material contains symmetries, you can specify only a few coefficients, which are expanded to a matrix using the following patterns:

- **Isotropic** (the default) — enter only one value c.

$$C = \begin{bmatrix} c & 0 & 0 \\ 0 & c & 0 \\ 0 & 0 & c \end{bmatrix}$$

- **Diagonal** — enter the diagonal components for an anisotropic material with the main axes aligned with the model's coordinate system.

$$C = \begin{bmatrix} c_{11} & 0 & 0 \\ 0 & c_{22} & 0 \\ 0 & 0 & c_{33} \end{bmatrix}$$

- **Symmetric** — enter a symmetric matrix using the diagonal components and the upper off-diagonal components.

$$C = \begin{bmatrix} c_{11} & c_{12} & c_{13} \\ c_{12} & c_{22} & c_{23} \\ c_{13} & c_{23} & c_{33} \end{bmatrix}$$

- **Full** — enter the full 2-by-2 (2D) or 3-by-3 (3D) matrix for an anisotropic material:

$$C = \begin{bmatrix} c_{11} & c_{12} & c_{13} \\ c_{21} & c_{22} & c_{23} \\ c_{31} & c_{32} & c_{33} \end{bmatrix}$$

Specifying Initial Values

An **Initial Values** node is added by default to each physics interface.

In some types of analyses initial values must be provided:

- As the initial condition for a time-dependent analysis.
- As an initial guess for the nonlinear stationary solver.
- As a linearization (equilibrium) point when solving a linearized stationary model or when performing an eigenvalue study.

To enter initial values, in the **Model Builder**, click the **Initial Values** node under a physics interface node. In the **Settings** window, enter the **Initial Values** for all dependent variables (fields) in the physics interface. The default initial values are usually zero.

For some physics interfaces you can also enter initial values for the first time derivative of the dependent variables. These are used when solving time-dependent problems containing second time derivatives (wave-type applications). Like other default settings, these initial values apply to all domains where no other values are specified.

To use different initial values in different domains, add another **Initial Values** node from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (macOS or Linux users), or right-click to access the context menu (all users).

See Dependent Variables for more information about handling and plotting initial values.

Equation View

Equation View ( ) is a subnode available for all physics feature nodes. To display these subnodes, click the **Show More Options** button ( ) and select **Equation View** from the **Show More Options** dialog box.

The **Settings** window for **Equation View** contains detailed information about the implementation of each physics feature: variables, shape functions, weak-form equation expressions, and constraints.

To update the values in the **Settings** window for **Equation View** to reflect the latest changes in a physics feature, click the **Refresh Equations** button ( ) in the **Settings** window's toolbar.

Editing the predefined expressions for variables, equations, and constraints means that the equations are altered and that COMSOL Multiphysics solves the model using the new expressions.

You can edit the expressions or values of variables, weak-form expressions, and constraints in the corresponding tables under **Variables**, **Weak Expressions**, and **Constraints**, respectively. This makes it possible to introduce custom changes to the equations and variable definitions. If the expression that defines a variable, for example, does not fit inside of the text field, a tooltip displays the entire expression. Press Ctrl+Space or use the **Insert Expression** button ( ) below the tables to choose from a number of applicable variables, parameters, functions, operators, and constants that you can insert into the expression at the position of the cursor. In the table of variables under **Variables**, you can click any of the column headers to sort the table contents alphabetically based on the contents of that column (in ascending order; click again for descending order; click yet again to restore the original order).

For a changed definition of a variable or a change to a weak-form expression or constraint, a warning icon ( ) appears in the leftmost column, and a small padlock is added to the lower-right corner of the icon for the physics node where you have made modifications in its equation view. To restore only the change in the selected variable, weak-form expression, or constraint, click the **Reset Selected** button ( ) under the table in the **Variables**, **Weak Expression**, or **Constraints** section. To reset all changes in the equation view, click the **Reset All** button ( ) in the **Settings** window's toolbar. If no changes remain, the padlock disappears from the corresponding physics node. An orange color for the expression that defines the variable is a warning that the unit of the expression does not match the expected unit for the variable that it defines. To store all information in the tables under **Variables**, **Shape Functions**, **Weak Expressions**, or **Constraints** to a text file, click the corresponding **Save to File** button ( ).

For information about the Equation displays available, see Physics Nodes — Equation Section.

STUDY

From the **Show equation view assuming** list, choose **No study** or any of the available studies. The equation view of the parent feature is then recomputed with the assumption that the selected study step was solved. This operation also updates all children to the parent feature, so the lists in their equation views are also updated. When solving a study (or study step), the list also changes to represent the last computed study step. The default is **No study** and represents a default behavior, which computes the equations without any study type information. The equation form used is then undefined and depends on the physics that the parent feature belongs to. The equation view reverts to **No study** if you change some setting in the parent feature to indicate that the equation view no longer represent a specific study step.

VARIABLES

This section has a table with the variables that the physics node defines. The table includes these columns:

- **Name**: the name of the variable.
- **Expression**: the expression, using COMSOL syntax, that defines the variable.
- **Unit**: the unit for the variable (in the active unit system). If the unit of the expression does not match the unit of the variable, the expression is displayed in orange.
- **Description**: a description of the variable.
- **Selection**: the geometric entities (domains, boundaries, edges, or points) where the variable is defined (**Domain 1**, for example).
- **Details**: this column contains some details about the variable's behavior. See About the Details Column below.

If you click a single variable, its selection, as indicated in the **Selection** column, appears in the **Graphics** window.

SHAPE FUNCTIONS

This section has a table with the dependent variables that the physics node defines and their shape functions. This is primarily applicable to equation model nodes; for most physics nodes such as boundary conditions, the table is empty. The table has these columns:

- **Name**: the name of the variable.
- **Shape function**: the type of shape function (element) for the variable (for example, **Lagrange** for Lagrange elements, which are the most common elements).

Selecting an Element Type

- **Unit**: the unit for the variable (in the active unit system).
- **Description**: a description of the variable.
- **Shape frame**: the frame type (typically either a spatial or a material frame) for the shape function.
- **Selection**: the geometric entities (domains, boundaries, edges, or points) where the shape function is defined (**Domain 1**, for example).
- **Details**: This column contains some details about the shape function's behavior. See About the Details Column below.

WEAK EXPRESSIONS

This section has a table with the weak-formulation equation contributions that the physics node generates. The table consist of the following columns:

- **Weak expression**, the equation expressed in a weak formulation. It is possible to modify these expressions, but you then override the equation as specified by the physics interface, and a warning appears in the leftmost column of the table.
- **Integration order**, the order for the integration of the weak expression (see integration order in the *Glossary)*. Polynomials of at most the given integration order are integrated without systematic errors. For smooth expressions, a sufficient integration order is typically twice the order of the shape function. For example, the default integration order for second-order Lagrange elements is 4. You can modify the integration order if desired. A warning then appears in the leftmost column of the table to indicate that the predefined order has been changed.

See the following blog post for information about modifying the integration order: https://www.comsol.com/blogs/introduction-to-numerical-integration-and-gauss-points/

- **Frame**, the frame type (typically either a spatial or a material frame) used when integrating the expression.
- **Selection**: the geometric entities (domains, boundaries, edges, or points) where the weak expression is defined (**Domain 1**, for example).

Each equation contribution appears on its own row under **Weak expression**, but the order is not significant.

The PDE interfaces and the ODEs and DAEs interfaces do not display any weak expressions. They are either implemented using strong formulations, directly display the weak formulation, or define equations discretized in the time domain only.

CONSTRAINTS

This section has a table with the constraints that the physics node generates. This is typically the case for boundary conditions of constraint types, such as prescribed displacements, temperature, or velocities. Many other physics nodes do not generate any constraints, and the table is then empty. The table consists of the following columns:

- **Constraint**: the expression for the constraint.
- **Constraint force**: the expression that defines the associated constraint force, which is typically the test function of the constraint.
- **Shape function**: the type of shape function (element) for the constraint (for example, **Lagrange** for Lagrange elements).
- **Selection**: the geometric entities (domains, boundaries, edges, or points) where the constraint is defined (**Boundaries 1–5**, for example).

ABOUT THE DETAILS COLUMN

The **Details** column shows some information about the behavior of variables and shape functions. For variables:

- An empty cell indicates that overlapping contributions are overridden.
- **+ operation** indicates that overlapping contributions are added.
- For some variables, **Meta** indicates that the variable definitions are not fully updated until you solve the model. It is therefore not possible to edit the expressions for such variables.
- In rare cases, other operations (* **operation**, for example) can occur.

For shape functions:

- **Slit** means that the shape function creates a slit for the degree of freedom.

Physics Nodes — Equation Section

For each physics node there is an **Equation** section that is available by default on the **Settings** window. This section has options to display mathematical equations applicable to the node.

> If you do not want to display the **Equation** section, click the **Show More Options** button ( ) and clear the **Equation Sections** check box in the **Show More Options** dialog box.

> Equation View

The display options available from the lists depend on the study types and other physics-specific factors. See Figure 3-20 for an example comparing the equations that display for a **Stationary** or **Time Dependent** study for a **Heat Transfer in Solids** interface. Some **Settings** windows do not have any options and only display the relevant equation and other windows have additional sections that become available for the **Equation** display based on the study type selected.

> Study and Study Step Types

Node Contributions Display a Dotted Line Under Part of the Equation

For all physics nodes (excluding the main physics interface node level), the equation that displays includes a dotted line underneath where the node's contribution is made to the equation. See Figure 3-19 for an example where a section of the heat transfer equation is underlined, indicating where the **Solid** node contributes to it.

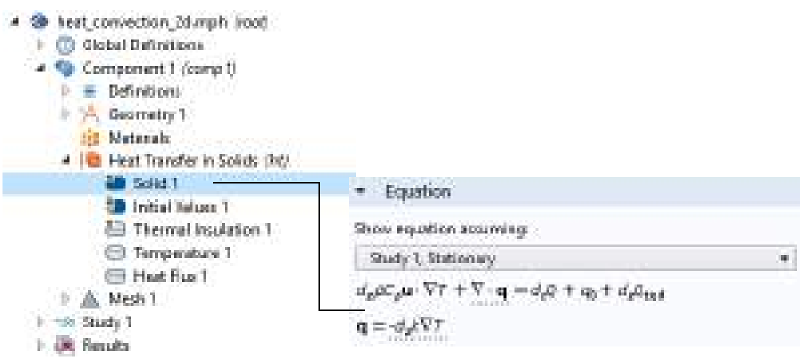


Figure 3-19: The Solid contribution to the equation for a 2D model.

Equation Form

When you add physics interfaces to a Component, the supported **Study types** are listed in the **Equation form** list. **Study controlled** is the default; select another option as needed. When the setting is **Study controlled**, the study controls the equation form — stationary or time dependent, for example — for the physics interface. In some cases, that equation form might not be compatible with the physics covered by the physics interface; the physics interface then uses its default equation form (typically, a stationary equation form). You can then instead choose one of the supported study types.

Show Equation Assuming

The **Show equation assuming** option is available by default when **Study controlled** is selected (or left as the default) as the **Equation form**. Options availability is based on the studies added and defined for the model.

For the following options — frequency and mode analysis frequency — you also have the option to use another frequency than the one used by the solver. This can be necessary if you need two different frequencies for two physics interfaces.

Frequency

This option is available if **Frequency domain** is selected as the **Equation form**. The default uses the frequency **From solver**. If **User defined** is selected, enter another value or expression (SI unit: Hz).

Mode Analysis Frequency

This option is available if **Mode Analysis** or **Boundary Mode Analysis** is selected as the **Equation Form**. Enter a value or expression in the field (SI unit: Hz). Specify a frequency (it is not present as a solver variable).

Port Name

This option is available with the RF Module **Electromagnetic Waves** interface and if **Boundary Mode Analysis** is selected as the **Equation Form**. Enter a value in the field (unitless).

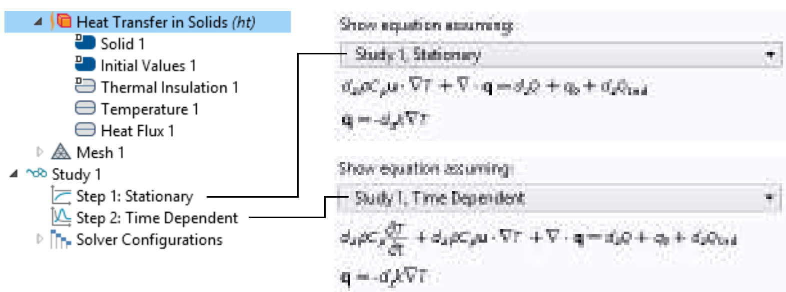


Figure 3-20: An example of the Equation section on a Heat Transfer interface. Selecting the study type updates the equation accordingly.