COMSOL Multyphysics: Overview of software package and capabilities

Lecture 5

Special Topics:
Device Modeling

Outline

- Basic concepts and modeling paradigm
- Overview of capabilities
- Steps in setting-up a model
- Hands on: assembling and running sample models
- Next time: Brief literature survey of problems/results

COMSOL Multiphysics Introduction

- Partial differential equation solver package with front-end developed for visual input and output
- Most used through different "modules" with predefined "physics", greatly simplifying modeling of device geometry, governing equations, boundary conditions, etc.
  - Also takes input in terms of user-defined equations
- Available across platforms (Windows, MAC, Linux); since version 5 can create modeling apps

COMSOL: Physics-based modeling (main program)

- Electrostatics, and electric currents
- Heat transfer in solids and fluids
- Joule heating
- Laminar flow
- Pressure acoustics
- Solid mechanics
- Transport of diluted species
- Additional physics interfaces through *modules*

COMSOL Multiphysics: modules

COMSOL Multiphysics: modules

COMSOL Multiphysics: available modules

Comsol Classroom License Table

Product	Number of licenses
Comsol Multiphysics	30
AC/DC Module	30
CFD Module	30
Chemical Reaction Engineering Module	30
ECAD Import Module	30
Heat Transfer Module	30
LiveLink for Excel	30
LiveLink for MATLAB	30
MEMS Module	30
Microfluidics Module	30
Nonlinear Structural Materials Module	30
Particle Tracing Module	30
RF Module	30
Structural Mechanics Module	30

Modules available in P&A computer room for the duration of course

COMSOL modeling flow

- Select the appropriate model attributes (Wizard)
  - Dimension 3D vs.2D vs. 1D, etc.
  - Choose physics elements
- Draw or import the model geometry, add materials
- Set up the subdomain equations (satisfied internally within geometry) and boundary conditions
- Mesh geometry
- Solve the model
- Apply postprocessing, plot results

Creating a new model

- The two main components of the COMSOL Desktop environment are the Model Builder and Application Builder
- The Model Builder is the tool where you define the model and its components; accomplished by building a model tree
- The Application Builder allows you to create an application (with a specialized user interface) based on a model created with the Model Builder

Creating a new model

- When open COMSOL Multiphysics model or create a new model, you follow the Model Builder tree: here you add nodes, populate parameters, and control all program settings
- Start with Model Wizard: select model dimension, physics elements, and study type (stationary, time-dependent, etc.)
- Alternatively, start with Blank model and add components, physics, study type, etc. to the tree

Adding global parameters

- Global Definitions (top of the Model Builder tree)
- Specify parameters applicable to the whole model
  - Parameterizing geometric dimensions
  - Specifying mesh element sizes
  - Defining parametric sweeps
  - Specify value, units in [] – will be converted to the standard units (e.g., degF – degree on the Fahrenheit scale, standard unit of T is K, Kelvin)
- All expressions are evaluated before a simulation begins, therefore may not depend on the dependent variables for which your equations are solving (including time and coordinates)

Setting up geometry

- Select model dimension first
- Create geometry in COMSOL
  - Work Plane with 2D geometry modeling
- Or import geometry file
  - The DXF (2D), VRML (3D), and STL (3D, used for 3D printing) file types are available for import without any add-on products
  - Other CAD files are supported with add-on CAD Import Module and LiveLink products for CAD; expand import/processing functionality

Setting up geometry

Creating geometry in COMSOL from the Work Plane:

- Start with 2D geometry modeling, then use extrude, revolve, sweep to convert to 3D
- Primitive solid objects: block, cone, cylinder, sphere, etc.; parametric helix, curves, surfaces
- Interpolation curves; boolean operations union, intersection, difference, and partition
- Hybrid modeling with solids, surfaces, curves, and points

Adding materials

- Material can be added from Materials library (either global, or local)
  - Each material is specified in terms of relevant parameters, e.g., thermal expansion coefficient
  - Properties required by the physics but missing in the material are marked with a warning sign
  - Blank material with user-defined properties can be added
- Extended Material library can be added as a module (2,500 materials)

Applying physics to your model

- Connect geometry (domains and sub-domains) with governing equations
- If use different physics (e.g., electrical currents and heat transfer), add Multyphysics node, containing coupled physics features (e.g., electromagnetic heat source term)
- Specify boundary conditions (initial values) for each physics input; verify default assignments, override as needed

Meshing

- Free tetrahedral meshing
- Swept mesh with prism and hex elements
- Boundary layer meshing
- Free triangular meshing of 3D surfaces and 2D models
- Mapped and free quad meshing of 3D surfaces and 2D models
- Copy mesh operation; virtual geometry operations
- Mesh partitioning of domains, boundaries, and edges

Solving your model

- Choose study type: stationary, time-dependent, eigenvalue, etc.
- Corresponding solver will be utilized
- Specify solver configuration information: which physics interface and geometry to use, which variables to solve for, and which solvers to use for the type of study to perform
- May want to examine initial values
- Compute the model – switch to the Results node

Results

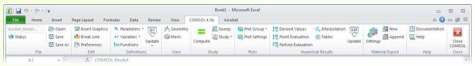
- Visualization: surface, isosurface, arrow, slice, streamline, and contour plots
- Export graphs as images and data (not straight-forward for 3D plots)
- Post-processing
  - Integration, average, max, and min of arbitrary quantities over volumes, surfaces, edges, and points
  - Custom mathematical expressions including field variables, their derivatives, spatial coordinates, time, and complex-valued quantities

Report generation

- To document your models, the COMSOL Report Generator provides a comprehensive report of the entire model, including graphics and parameters of the geometry, mesh, physics parameters, boundary conditions, and postprocessing quantities
- You can save the report as an HTML (with all links) or Word (with TOC) file for viewing and further editing

Connecting to other packages

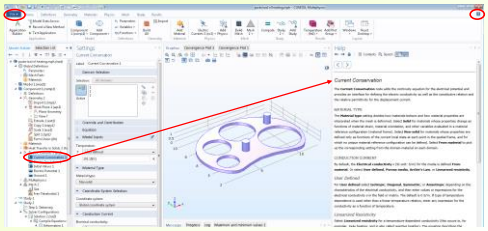
- COMSOL provides access to scripting environment through LiveLinks (up to version 3.4 there was an internal scripting language)
- In addition to CAD-type programs, LiveLink interfaces are available for Excel and Matlab
- Data processing (e.g., statistical analysis), access to modeling tools from Excel/Matlab interface



Connecting to other packages

- The native COMSOL model file format is *.MPH
  - Full MPH-files include all meshes and solutions (can be huge)
  - Compact MPH-files include all settings but has no built meshes and solution data to save space; can open these to study the settings, to mesh and re-solve
- COMSOL can save a model as *.java,*.vba or *.m file
  - Compile Java files and run as separate applications
  - Use with Excel's VBA (Visual Basic for Applications)
  - Edit with Matlab scripting environment

COMSOL Multiphysics

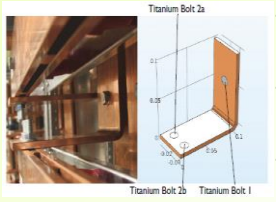


Use topic-based help:
press F1
click ?
access from File->Help

Some drawbacks of COMSOL

- The error messages are often obscure
- Model inconsistencies are difficult to identify (user-related mostly)
- Backward compatibility is problematic
- Inter-module+main program compatibility is problematic (do not add physics from modules if not necessary)

Model for hands-on: Electrical Heating in a Busbar



- Example 2 from Introduction to COMSOL Multiphysics, ver. 5.2
- The resistive (ohmic) heating Q_e due to the electric current
- Solve for electric potential and temperature

- Potential applied to Bolt 1; Bolts 2a,b are at ground
- Busbar is in air, cooled by convection

Model for hands-on: Electrical Heating in a Busbar

- Governing equations:

$$\nabla \cdot (k\nabla T) + Q_e - Q_0 = 0$$
$$Q_0 = h(T_{ext} - T)$$
$$Q_e = \mathbf{J} \cdot \mathbf{E}$$
$$\nabla \cdot \mathbf{J} = 0, \quad \mathbf{J} = \sigma\mathbf{E}, \quad \mathbf{E} = -\nabla V \Rightarrow \nabla^2 V = 0$$

Parameters:
Heat source Q_e due to the electric current; heat sink Q_0 due to convection; solid thermal conductivity k, temperature T, electric conductivity σ; **J** - current density (A/m^2), **E** - electric field strength (V/m)

Model for hands-on: Electrical Heating in a Busbar

- Solver does Laplace equation $\nabla^2 V = 0$ for electric potential with boundary conditions
- From V – finds E, from E finds J, from J·E – heat source Q_e
- Next, heat transfer equation is solved: Poisson equation for temperature with Q_e heat source and convection heat sink

Model for hands-on: Electrical Heating in a Busbar

- Introduction to COMSOL Multiphysics provides step-by-step guide
- Extension of the electrical heating problem:
  - Add structural mechanics: solve for Joule heating and thermal expansion
  - Add cooling by airflow: solve for fluid flow and Joule heating

Summary

- COMSOL Multiphysics is a versatile commercial PDE solver
- A number of sample models available for each module
- Well-developed run-time (through Excel, MatLab) and post-processing facilities
- Capabilities to develop apps with desired user interface for model distributions

References

- Introduction to COMSOL Multiphysics, www.comsol.com
- W. B. J. Zimmerman, Multiphysics modelling with finite element methods, World Scientific, 2008