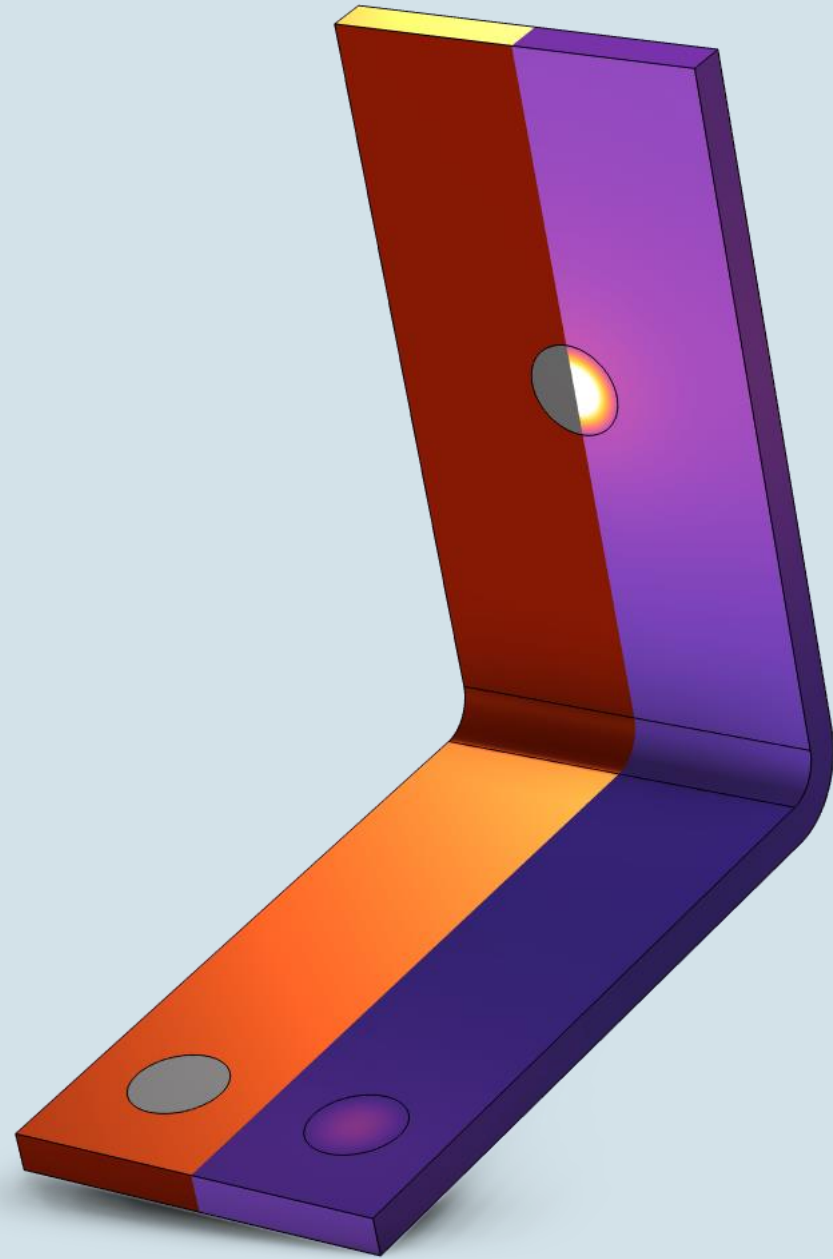


WORKSHOP - COMSOL Multiphysics

Tomáš Vrbata
HUMUSOFT
vrbata@humusoft.com

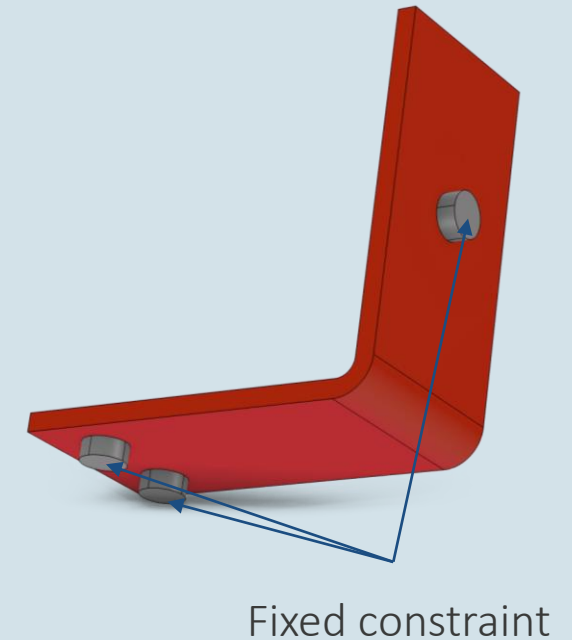
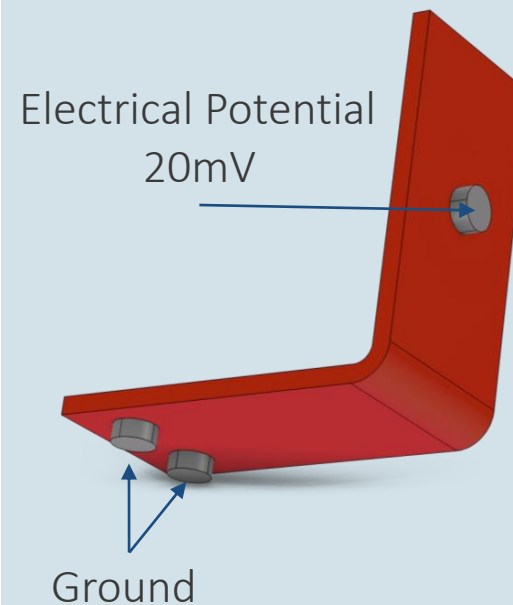
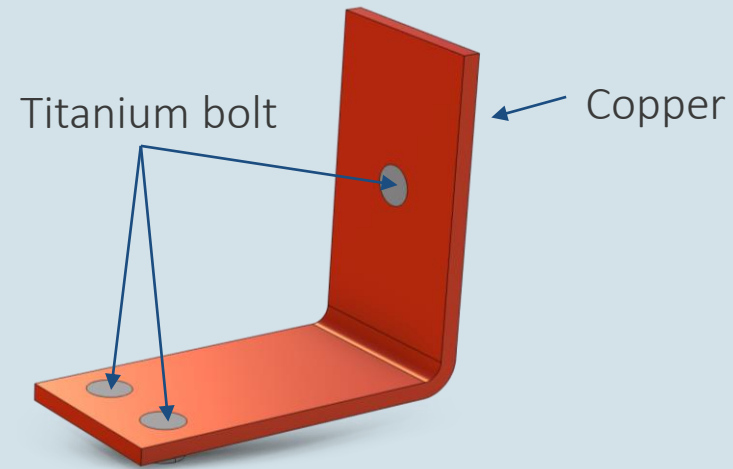
Simulation of a Busbar in COMSOL Multiphysics

- Multiphysics simulation
- Electrical heating in a busbar
 - Thermal stresses
- Surrogate model (Digital twin)

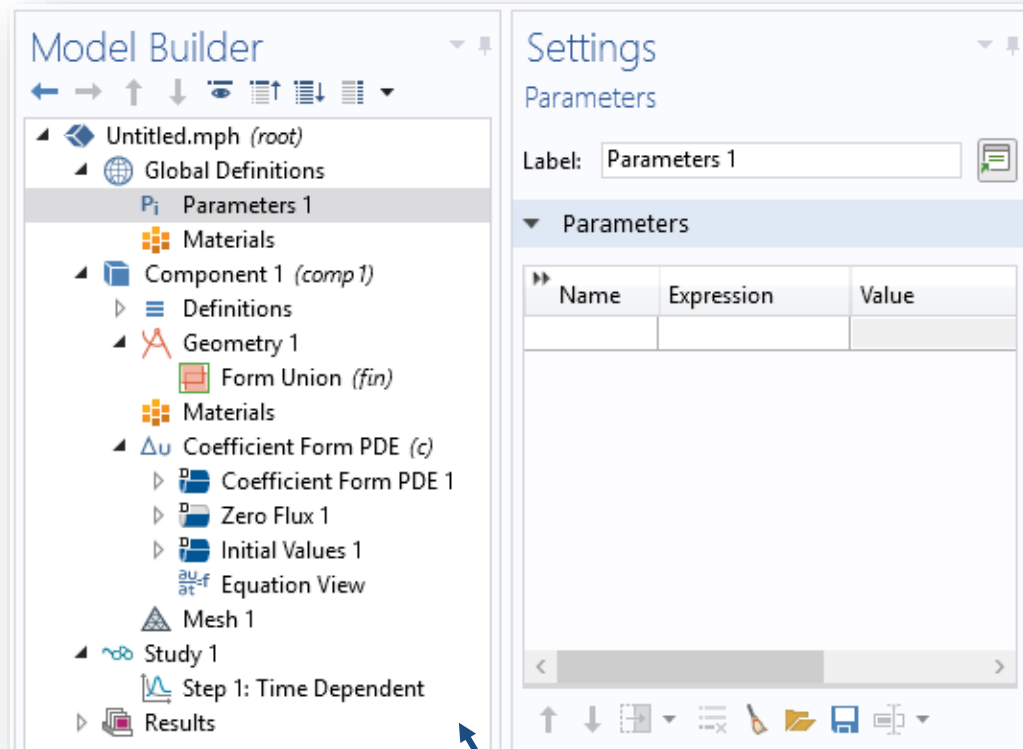


Simulation setup

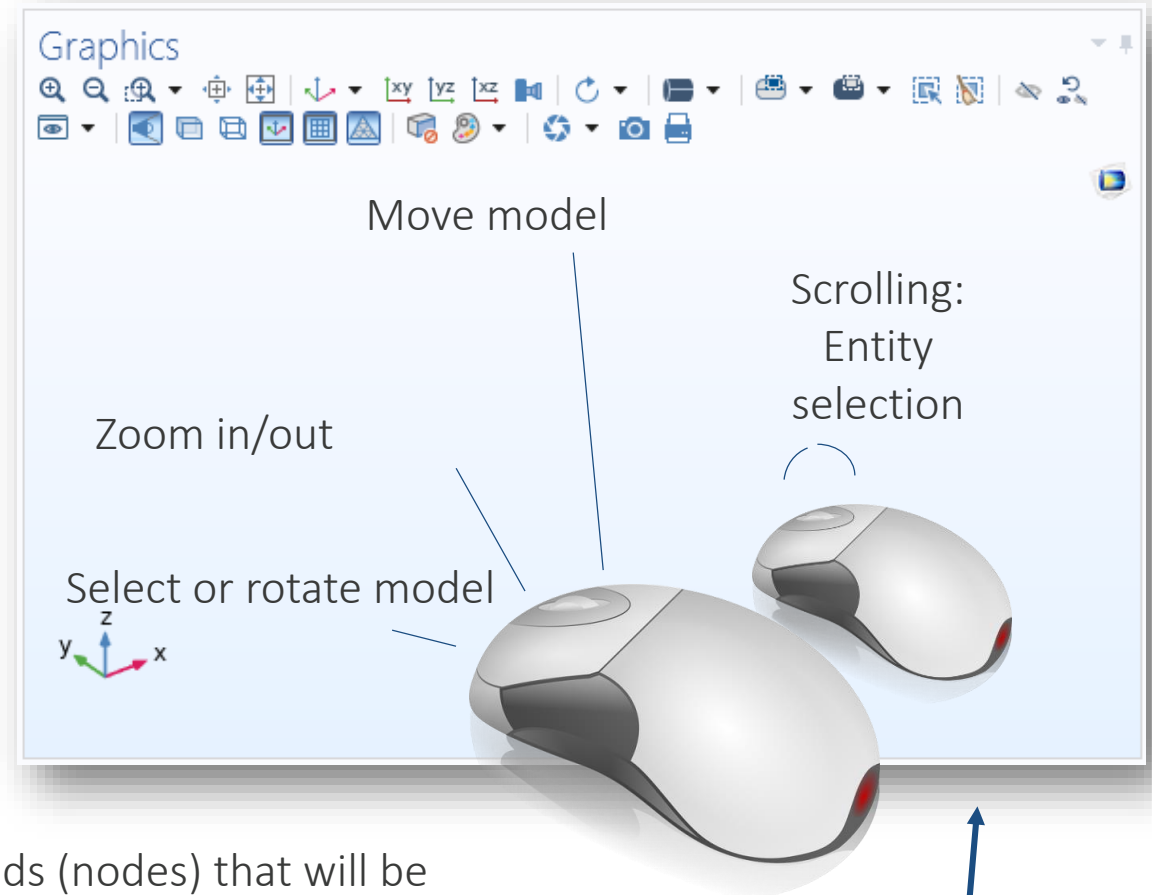
- 1) Parameter upload and geometry creation
- 2) Assignment of materials
- 3) Setup physics
 - Electrical potential + Ground
 - Heat Flux
 - Fixed constraint
- 4) Multiphysics node
- 5) Mesh creation – mesh dependency of results
- 6) Postprocessing



Fundamentals of Controlling COMSOL Multiphysics



The "*Model Builder*" displays a tree structure of commands (nodes) that will be run top-down to create a simulation. By right-clicking on the nearest parent command, a menu of other commands in that branch can be displayed. Next to "*Model Builder*" is the "*Settings*" window, which sets the selected command.



Window with graphical representation of the model.