

COMSOL tutorial 3: Joule heating, thermal expansion and fluid flow cooling

With the geometry created last week, simulate Joule heating by following the steps in “Example 2: The Busbar — A Multiphysics Model”, in the document:

IntroductionToCOMSOLMultiphysics.pdf

This pdf file should be in the following folder:

COMSOL (Installation directory)/Multiphysics/doc/pdf/COMSOL_Multiphysics/

Follow the tutorial, but skip the section “Mesh”. Test by yourself the output features. Skip the section “Building an Application”. When you have finished this first part, continue the tutorial (Advanced Topics, Adding Physics) to simulate thermal expansion caused by Joule heating, and cooling by air flow. If you are interested, you can continue with “Parametric Sweeps”.

