Technische Universität München
Institut für Informatik
Dr. Tobias Neckel

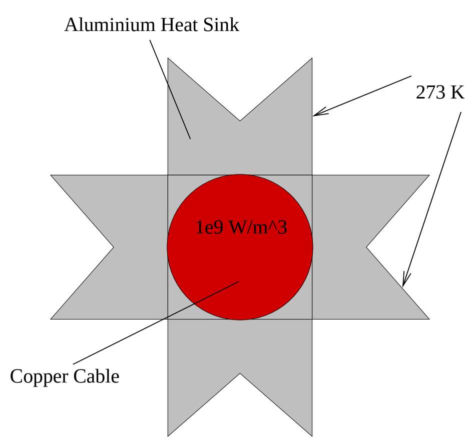
WS 2012/2013

Comsol Tutorial

Heat conduction in a simple heat sink (aluminium) cooling a high voltage copper cable.

In this tutorial, we will solve one physical problem using a numerical toolbox called `COMSOL Multiphysics (FEMLAB)`. We calculate the heat conduction in a simple heat sink (aluminium) cooling a high voltage copper cable. In part **a)** of the tutorial we construct the geometry of the problem to be solved. As alternative we can use the already constructed geometry by applying the steps from part **b)**.

1 Copper Cable in a Simple Heat Sink (2D)



a) Build the heat sink geometry.

1. Start `COMSOL`, go to the **New** page in the **Model Navigator**.
2. Select Space dimension: 2D.
3. Expand COMSOL Multiphysics, expand Heat Transfer, expand Conduction, select Steady-state analysis, and press OK.

4. Press **Draw Mode** icon if this is not pressed by default.
5. Hold SHIFT and press **Rectangle/Square** icon in the **Draw** tool bar (left). Draw a rectangle with the parameters Width = 0.09, Hight = 0.03, Position x = 0.0, and Position y= 0.03.
6. Hold SHIFT and press **Rectangle/Square** icon. Draw a rectangle with the parameters Width = 0.03, Hight = 0.09, Position x = 0.03, and Position y= 0.0.
7. Select all objects in the work space by pressing CTRL-A. Press **Zoom Extents** icon. Click on **"Create Composite Object"** icon and press the Union button in the Shortcut frame of the opening dialog box. Note that $R1 + R2$ appears in the Set formula field. Uncheck the Keep internal borders check box and press OK to create the composite solid object.
8. Copy the composite solid object by pressing the **Copy** icon in the icon bar. Press the **Paste** icon to open the **Paste** dialog box. Enter 0 in the field for both the X-displacement and the Y-displacement and press OK.
9. Press the **Rotate** icon and enter a Rotation of 45 degrees in the opening dialog box. Enter 0.045 in both the field for Center point x and Center point y and press OK.
10. Select all objects in the work space by pressing CTRL-A and press the **Intersection** icon.
11. Hold SHIFT and press **Ellipse/Circle (Centered)** icon. Enter 0.015 in the field for Radius and 0.045 for the x and y coordinate of the Center.

b) Importing existing heat sink geometry

1. Start COMSOL, go to the **Open** page in the **Model Navigator**.
2. Browse to the file with heat sink model and press OK .

c) Set boundary conditions and material parameters.

1. Open the **Physics** menu in the menu bar. Select **Boundary Settings** to specify the boundary conditions. Go to the **Boundaries** page in the left frame. Hold SHIFT and mark all outer boundaries with the mouse. Switch to the **Boundary Condition** page in the right frame and select *Temperature* in the Boundary condition selection box. Enter 273 in the edit field for T_0 and press OK.

2. Open the **Physics** menu in the menu bar. Select **Subdomain Settings** to specify the material parameters. Mark the round inner subdomain in the left frame. Switch to the **Physics** page in the right frame and load the Library material *Copper*. Enter $1e9$ for the Heat source Q. Mark the outer subdomain in the left frame. Switch to the **Physics** page in the right frame and load the Library material *Aluminium*. Enter 0 for the Heat source Q because their is no heat source in the cooling element. Press OK to close the dialog box.
3. **<u>Save</u>** now your "raw" `COMSOL` model. So that you can start from this state later on! Don't overwrite this file!

d) Solve the Problem and visualise the result.

1. Select all objects in the workspace. Open the **Mesh** menu in the menu bar. Select **Free Mesh Parameters** to open the corresponding dialog box. Select Normal in the Predefined mesh sizes selection box and press Remesh. Press OK to close the dialog box.
2. Press the **Solve** icon. After solving the problem you will get a surface plot of the distribution of temperature in the domain.
3. Open the **Postprocessing** menu in the menu bar. Select **Plot Parameters** to open the corresponding dialog box. Open the page **General**. Uncheck auto and set the value of the element refinement to 1. Open the page **Surface** and select *Flat* in the Coloring selection box and press Apply. Plot the *Temperature gradient* instead of the *Temperature*.
4. What is your observation? (regarding the quality of the solution, what does the plot show?)

5. Open the page **Streamline** in the **Plot Parameters** dialog box. Activate the Streamline plot by checking the box at the top of the page. Select *Heat flux* in the Predefined quantities selection box, select *Magnitude controlled* in the Streamline plot type selection box, use 30 start points and press Apply.
6. Export your plot to an eps-file! Open **file** in the menu bar. Select **Export** and **Image**. Use the Preview to see if the plot looks fine. Enlarge the plot, if its components look as they were pressed together. (If you have problems with exporting the image, just make a screen shoot.)
7. Open the **Mesh** menu in the menu bar. Select **Mesh Statistics** to get more information about the mesh. Write the global number of elements into the table below (in the first column).

8. Open the **Postprocessing** menu in the menu bar. Select **Boundary Integration** to open the corresponding dialog box. Select all outer boundaries! Select *Normal heat flux* in the Predefined quantities selection box and press Apply to integrate the heat flux over the outer boundary. Read the value from the logging screen and write it into the table below.
9. Refine the mesh and calculate a new solution (Button next to the "Mesh Initialization"). Integrate once more the flux over the outer boundary and write the result together with the number of elements into the table below. Round the values to the third significant digit. Refine the the mesh and calculate the flux over the outer boundary until the rounded third significant digit does not change anymore (or until the solving takes too much time or space, which will be the case). Write your results down in the table! Change the linear system solver to *Geometric multigrid* (**Solve →Solver Parameters**) if you run into memory problems. (the #elements you find in **Mesh - Mesh Statistics**)

# elements					
value of the integral					
# elements					
value of the integral					

e) Solve the Problem with hand-made mesh refinement.

1. Open your "raw" model! Open the **Mesh** menu in the menu bar. Select **Free Mesh Parameters** to open the corresponding dialog box. Open the page **Boundary**! Mark all outer edges! Open the sub-page **Distribution** and check Constraint edge element distribution. Set the number of edge elements to 12. Mark the Distribution check box. Set element ratio to 5 and select *Exponential* for the Distribution method. Press **Remesh**.
   Take care that the mesh is refined in the neighbourhood of the "inner corners". So, check the direction of the exponential distribution along the outer edges and reverse the direction of some of the single edges if necessary! (this means for each edge separatley (not grouped) if refinement take place at the wrong end then you have to reverse the direction for that particular edge.)
2. Save this `COMSOL` model to a new file!
3. Calculate a new solution. Integrate the flux over the outer boundary and write

the result together with the number of elements into the table below. Round the values to the third significant digit. Refine the the mesh and calculate the flux over the outer boundary until the rounded third significant digit does not change anymore (or until the solving takes too much time and space). Write your results down in the table! Change the linear system solver to *Geometric multigrid* (**Solve** →**Solver Parameters**) if you run into memory problems.

# elements					
value of the integral					
# elements					
value of the integral					

f) Solve the Problem with an adaptive mesh refinement.

1. Open your "raw" model! Press the Initialize mesh icon.
2. Open the **Solve** menu in the menu bar. Select **Solver Parameters** to open the corresponding dialog box. Activate Adaptive mesh refinement and press OK.
3. Calculate a new solution. Open the **Postprocessing** menu in the menu bar. Select **Plot Parameters** to open the corresponding dialog box. Open the page **General**. Uncheck auto and set the value of the element refinement to 1. Open the page **Surface** and select *Wireframe* in the Fill style selection box and press Apply. What is your observation? (regarding the refinement)
4. Integrate the flux over the outer boundary and write the result together with the number of elements into the table below.
5. Open the page **Adaptive** in the dialog box **Solver Parameters**. Set the value for Maximum number of refinements to 3. Initialise the mesh! Calculate a new solution. Integrate the flux over the outer boundary and write the result together with the number of elements into the table below. Round the values to the third significant digit. Increment the number of mesh refinement steps for the adaptive solver and calculate the flux over the outer boundary until the rounded third significant digit does not change anymore (or until the solving takes too much time and space). Initialise the mesh before each calculation! Write your results down in the table! Change the linear system

solver to *Geometric multigrid* (**Solve** →**Solver Parameters**) if you run into memory problems.

# elements					
value of the integral					
# elements					
value of the integral					

6. Compare your results with the ones from the parts c) and d)!
7. Save your `COMSOL` model.