Technische Universität München
Institut für Informatik
Emily Mo-Hellenbrand
Kaveh Rahnema
Benjamin Uekermann

Lab Course
Scientific Computing

## Worksheet 5

distributed: 10.12.2014
due to: 12.01.2015, 3:00 pm (on the Moodle page)
personal presentation: 13.01.2015, see Moodle page for exact slots

In this worksheet, we will solve two physical problems using a numerical toolbox called `COMSOL Multiphysics (FEMLAB)`. In the first part, we calculate the heat conduction in a simple heat sink (aluminium) cooling a high voltage copper cable. In the second part, we simulate the fluid flow around a cylinder.

Save both scenarios in model files, answer all the questions, and fill the tables for the personal presentation.

1   Copper Cable in a Simple Heat Sink (2D)
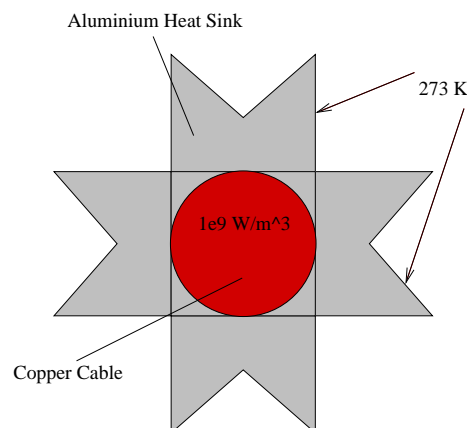
a) Build the heat sink geometry.

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 **Inter-section** 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) 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!

c) 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?)

short answer

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					

d) 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					

e) 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)

short answer

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.

f) Solve the Problem with an adaptive mesh refinement and "round" corners.

1. Open your "raw" model and switch to the **Draw** mode.

2. Press the **Fillet/Chamfer** icon in the draw tool bar. Mark all "inner corners". Select Fillet, set the radius to 0.0005 and press OK.

3. Initialise the mesh! Calculate a new solution, using an adaptive solver. 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. 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					

- Compare your results with the ones from the parts c), d) and e)!

- Save your COMSOL model.

2 Flow around a cylinder (2D)
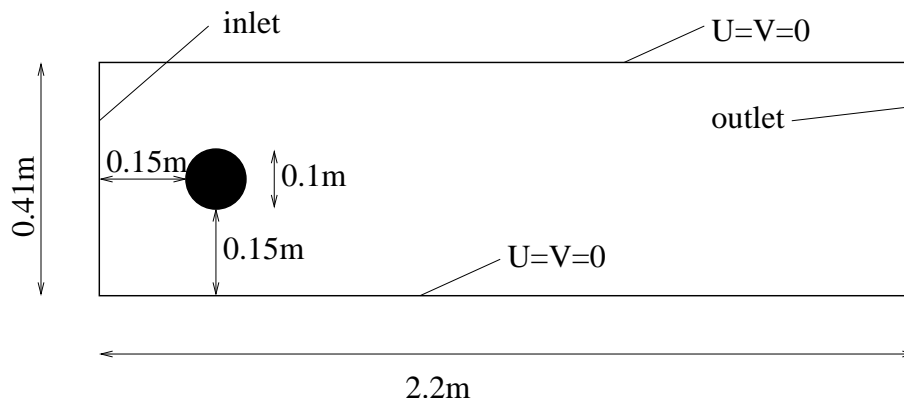
a) Setup and simulate a stationary flow scenario

1. Start COMSOL and initialise an Incompressible Navier-Stokes (steady-state now, later you will change it to transient) model in 2D (COMSOL Multiphysics → Fluid Dynamics) in the **Model Navigator**. (Take the difference area between the rectangle and the circle.)

2. Build a flow channel using the geometry information given in the figure above. Hint: Use the *Create Composite Object*

3. All external forces are neglected. The fluid is specified by its density $\rho = 1.0 kg/m^3$ (1kg pro m^3 is not a typose, it is a very light fluid) and its dynamic viscosity $\eta = 10^{-3} Pas$.

4. Define a new constant (**Options→Constants**) *Umax* and assign the value 0.3 to this constant.

5. Specify an *inflow* boundary condition at the short edge (of the flow channel) near to the cylinder. Use the expression $4 * Umax * y * (0.41 - y)/(0.41 * 0.41)$ to give the velocity in x-direction at the inflow boundary. Set the velocity in y-direction at the inflow boundary to zero. Specify an *outflow (neutral)* boundary condition at the opposite edge and *no-slip* boundary conditions at all other edges of the channel including the surface of the cylinder.

6. Choose a Stationary nonlinear solver with Adaptive mesh refinement. Use the GMRES solver with the Incomplete LU preconditioner to simulate this stationary flow scenario. Start with a *fine* mesh (Mesh→ Free mesh parameters→Predefined mesh sizes→Fine) and force the solver to perform three adaptive refinement steps.

7. **Hint**: Since we have non-linear equation (Navier-Stokes), choose one direct solver (e.g.: UMFPACK), which should work. Make sure that Comsol is not using any stabilization method for the Navier-Stokes (in **Physics - Subdomain - Stabilization** uncheck all stabilization options, for Comsol 3.4, deactivate all boxes in **Physics - Subdomain - artificial diffusion**) !

b) Visualise a stationary flow scenario

1. Plot the mesh in the neighbourhood of the cylinder and save the plot in a file.

2. Plot the *Velocity field* in a Surface plot, add a Streamline plot based on the *Velocity field* as well. Specify the start point coordinates of the streamlines by the x-coordinates $0.25, 0.25, 0.0, 0.0, 0.0, 0.0$ and the y-coordinates $0.19, 0.23, 0.19, 0.22, 0.15, 0.25$.

3. Determine the pressure up-stream and down-stream of the cylinder (**Postprocessing → Point Evaluation**). Hint: the pressure is accessible by the expression p.

pressure up − stream	*pressure down − stream*

4. Determine both components of the force acting on the surface of the cylinder by calculating the integral of the predefined quantity *Total force per area* over the boundary of the cylinder (**Postprocessing → Boundary Integration**).

force in x − direction	*force in y − direction*

5. Save your `COMSOL` model.

c) Simulate a time dependent flow scenario

1. Change the constant *Umax* to 3.0 in oder to specify a time dependent flow problem.

8

2. Change consequently the solver properties to Time dependent. Simulate 3.5 seconds with an output time step of 0.025 (Solver parameters → General → Time stepping → Time). Try to use a direct solver. (Make sure that you use a transient solver and analysis !)

3. Use the **adaptively refined mesh** from the stationary solution to solve the new scenario. (In case if you do not have that mesh, then just refine regulatly an inital mesh.)

4. Create an animation of the flow field of the time dependent solution (**Postprocessing → Plot Parameters → Animation**).

5. Create a new variable Fy (**Options → Integration Coupling Variables → Boundary Variables**) using the expression T_y_ns and all parts of boundary of the cylinder. Click on **Update Model** in the **Solve** menu in order to calculate the values for the new defined variable. (Try to keep the size of the movie file low. In case you have problems creating or submitting a movie, just make a screen shot at the last time step.)

6. Plot the variable Fy over the simulation time and estimate the frequency of the oscillating flow (**Postprocessing → Domain Plot Parameters → Point**).

$frequency$

7. Save your COMSOL model.

d) Simulate a time dependent flow scenario with additional heat transfer

1. Start your model from the previous part. Open the **Model Navigator** in the **Multiphysics** menu. Add the (transient) Convection and Conduction Heat Transfer mode in addition to the existing Navier-Stokes mode.

2. Take care that the Convection and Conduction mode is active in the **Multiphysics** menu and open the **Subdomain Settings**. Set the thermal properties of the fluid: The density $\rho = 1$, the conductivity $k = 40W/(mK)$, and the heat capacity $Cp = 800J/(kgK)$. Initialise the fluid with a temperature of $273K$.

3. Couple the velocity field in the heat equation with the velocity field in the Navier-Stokes (NS) equation. Hint: The expressions for the two components of the velocity in the NS mode are u and v. Enter them into the corresponding

fields in the dialog box. (In **Physics - Subdomain** input the u (velocity field).)

4. Open the **Boundary Settings** dialog box. Set the temperature at the inlet, the southern, and the northern boundary to $273K$. Set the temperature at the boundary of the cylinder to $323K$. Assume that the convective flux normal to the outlet boundary is zero.

5. Simulate 3.5 second with an output time step of 0.025. Start with the last solution of the previous part, by modifying the initial values of the flow domain, using the expressions for primary variables of the flow domain. Change the initial behaviour of the solver (**Solve** $\rightarrow$ **Solver Manager** $\rightarrow$ **Initial value** $\rightarrow$ Current solution).

6. Check the initialisation with **Solve** $\rightarrow$ **Get initial value**. Start the simulation, if everything works.

7. Plot the distribution of the temperature and save the plot to a file (at the final stage).

8. Create an animation of the temperature field. (Try to keep the size of the movie file low. In case you have problems creating or submitting a movie, just make a screen shot of the temperature at time 5s.)

9. Save your `COMSOL` model.