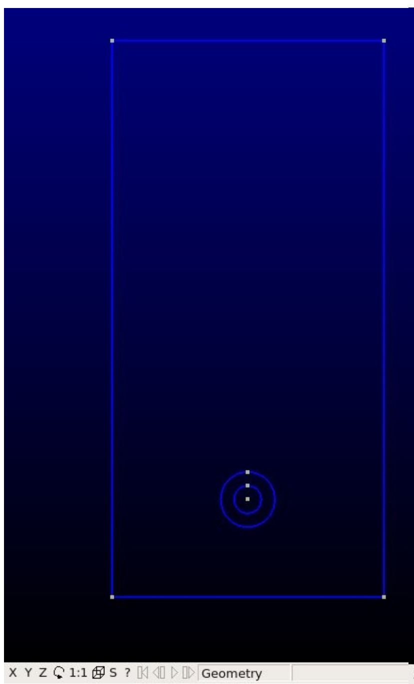
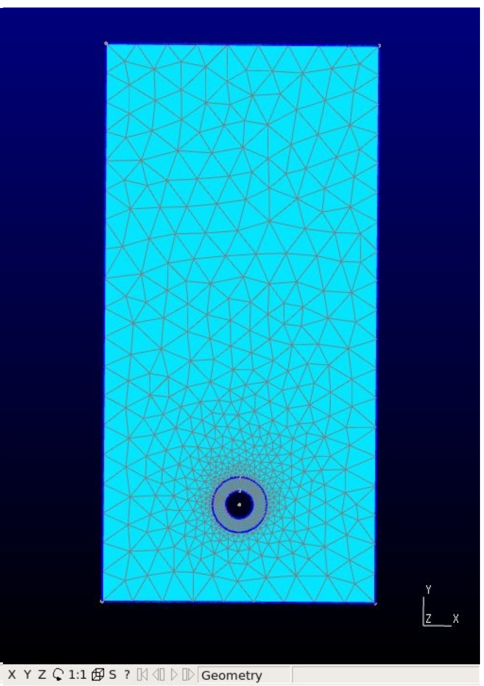
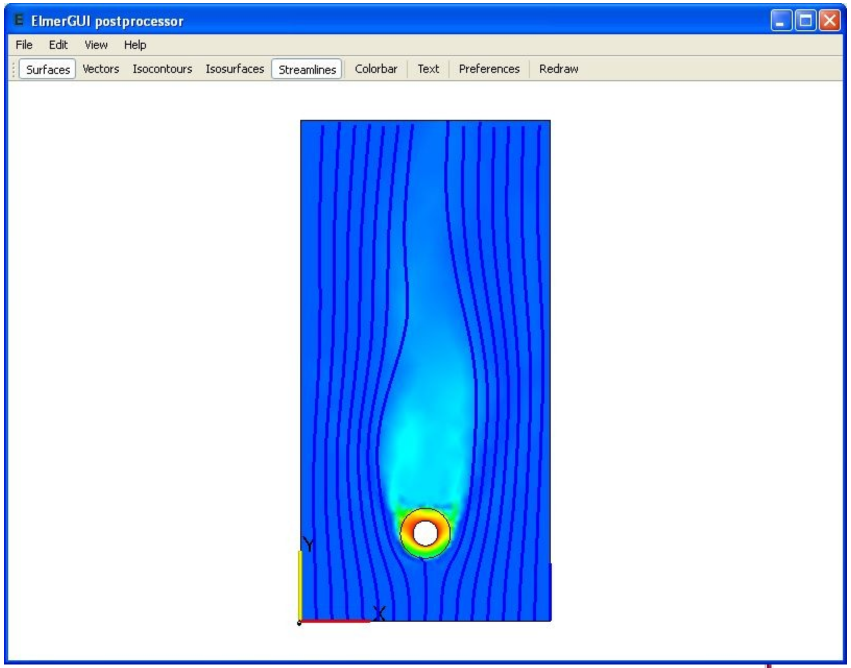
# Introductory Example

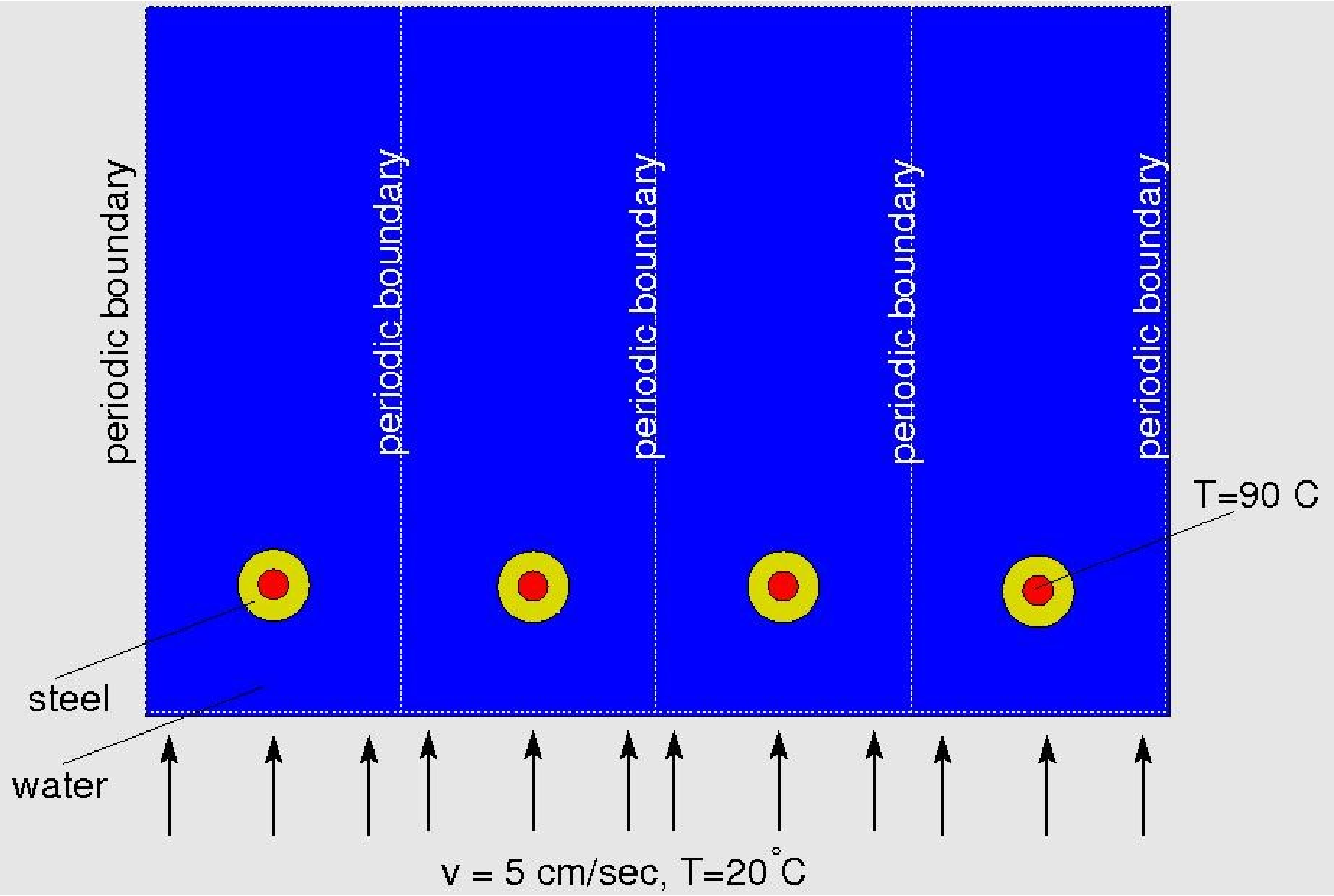
Example of coupled flow and heat transfer problem using ElmerGUI



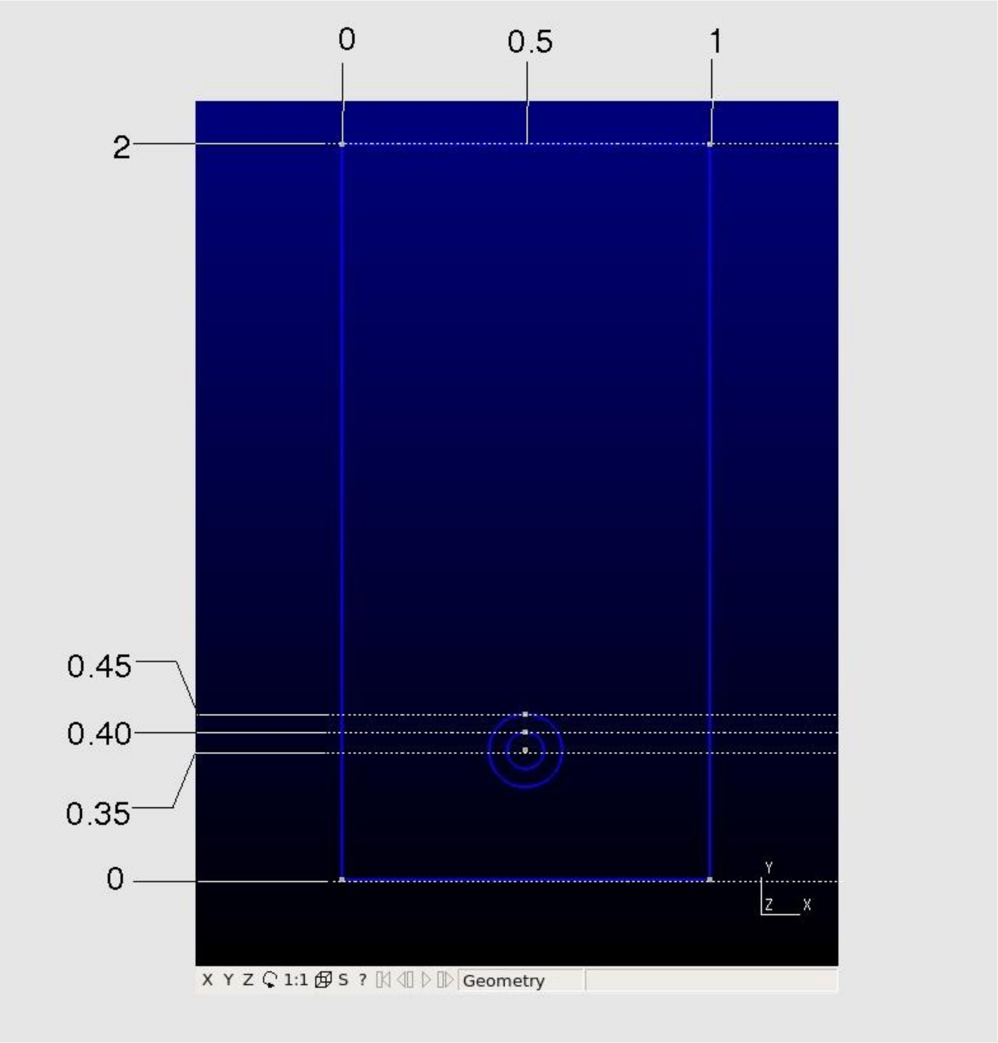
# Problem Outline




- Heat transfer through periodically arranged steel pipes into flowing water
- Utilizing periodicity
- Transient effects
- Different types of boundary conditions

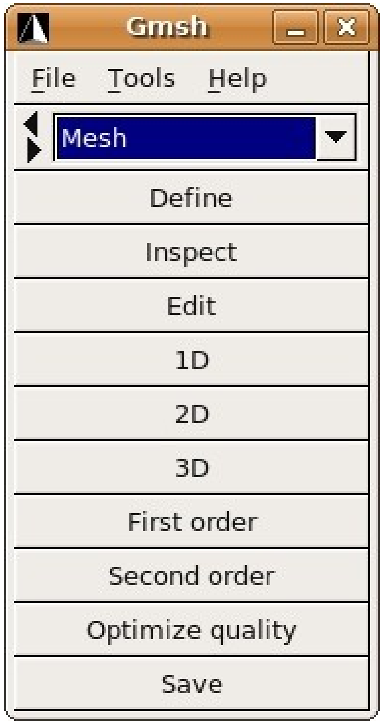
# Mesh

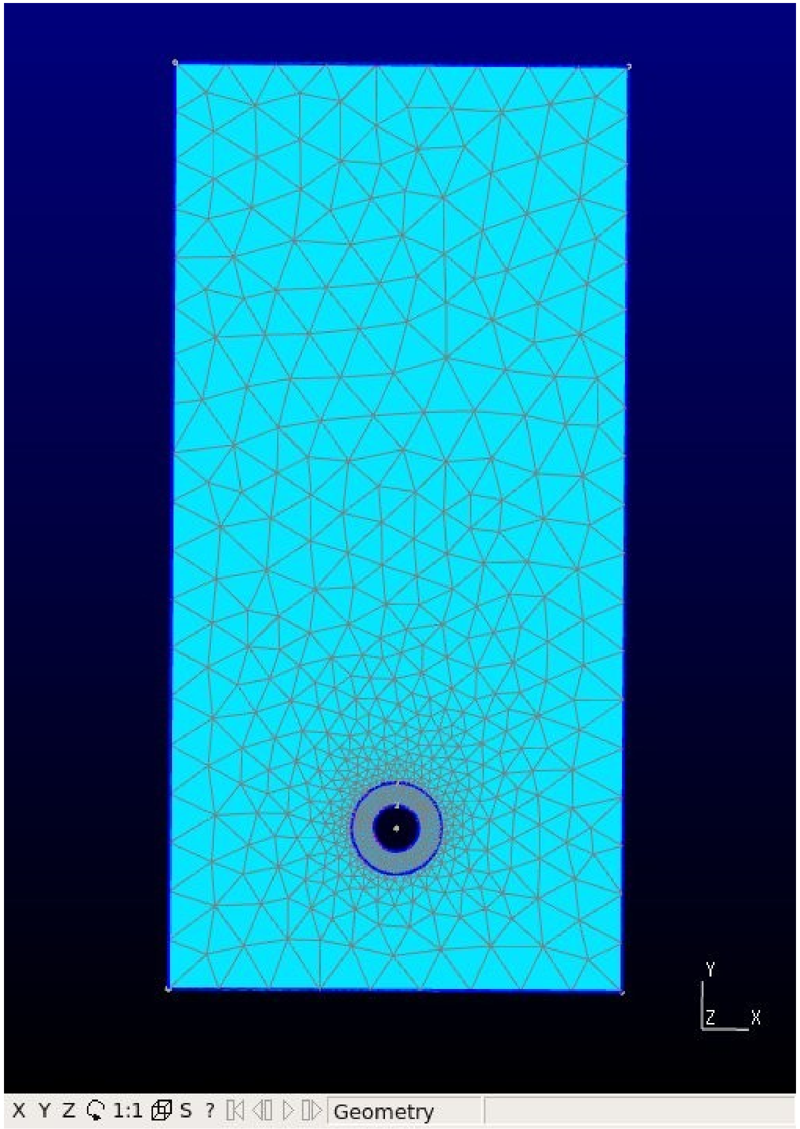
- Using GMSH
- Units in dm (0.1m)
- Bottom up strategy
- Create Points
- Construct lines from points
- Construct surfaces from lines
- Apply smaller (1 mm) mesh size around and in pipe
- Apply larger (1 cm) mesh size around
- If in doubt, just run the *.geo file



# Mesh

- After construction of geometry, choose **Mesh** in the main menu
- Then click on **2D**

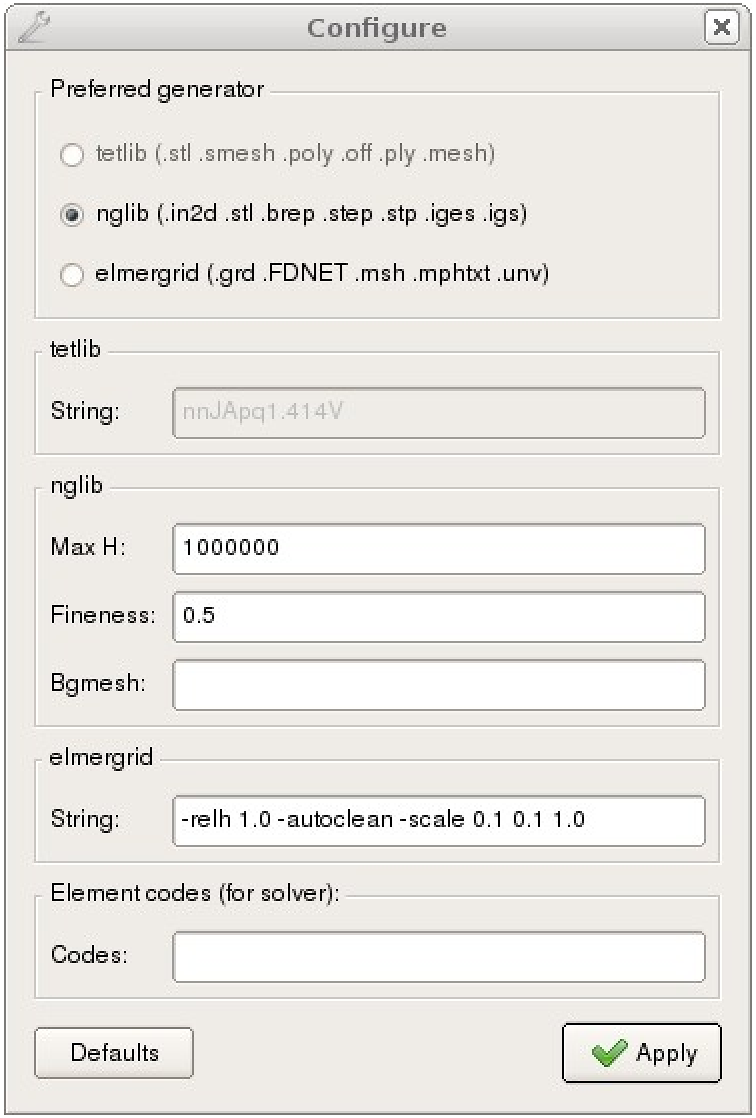



CSC

# ElmerGUI

- After launching ElmerGUI, open **Mesh -> Configure**
- Change the elmergrid-string to the value given in the r.h.s. Picture
- Explanation:
  - -autoclean clears up unused entities in GMSH output
  - -scale 0.1 0.1 1.0 brings us back from $10^{-1}$ m units into the SI world
- In **File -> Open** choose *periodiccyl.msh*



# Alternative mesh import

- Outside ElmerGUI
- In the comand line:

```
ElmerGrid 14 3 periodiccyl.msh –autoclean –scale 0.1 0.1 1.0
```

- This directly creates the mesh in ElmerSolver format:

```
mesh.{header,nodes,elements,boundary}
```

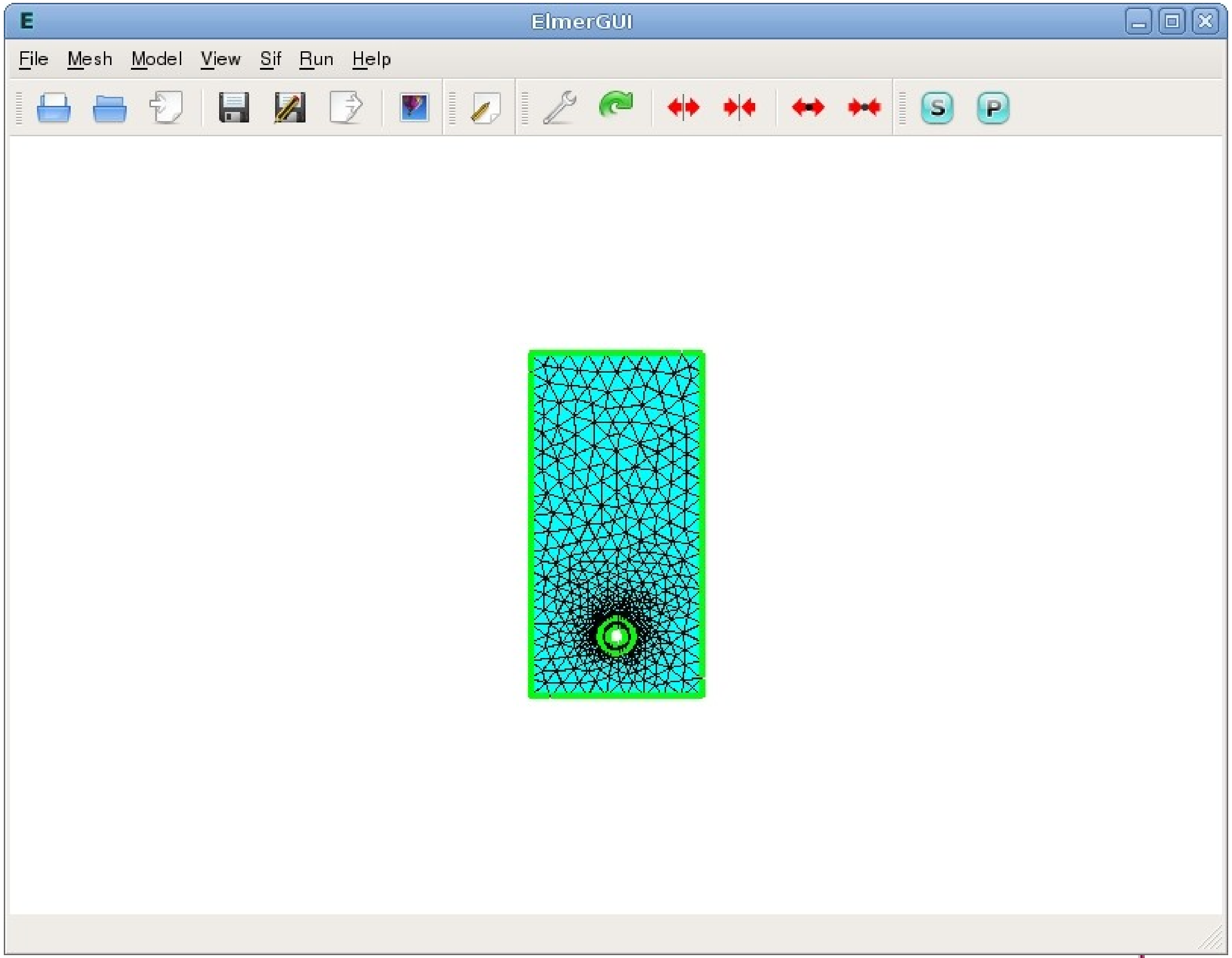
- Import into ElmerGUI: **File** -> **Load Mesh**
- Possibility to create an ElmerPost readable format (checking the boundaries):

```
ElmerGrid 14 2 periodiccyl.msh –autoclean –scale 0.1 0.1 1.0
```

- This creates the file `periodiccyl.ep` that can be opened within ElmerPost
- Existing meshes (ElmerSolver format) are imported via **File** -> **Load Mesh**

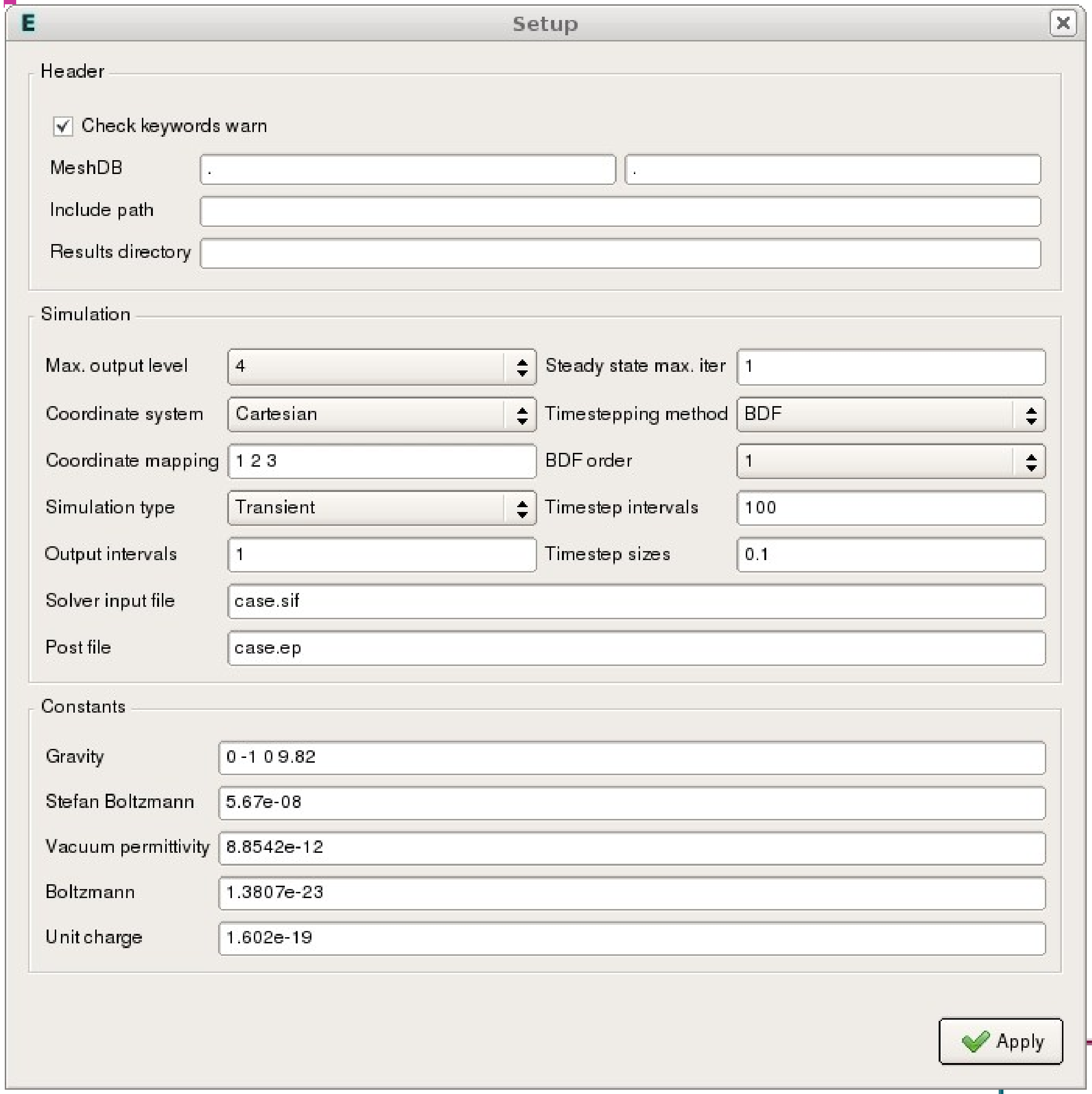
# ElmerGUI main window

- Menu bar
  - ”Chronological” order (from left to right)
- Tool bar
  - Quick access to most important menu options
- Display
  - Enables some interaction (selection of bodies/ boundaries)



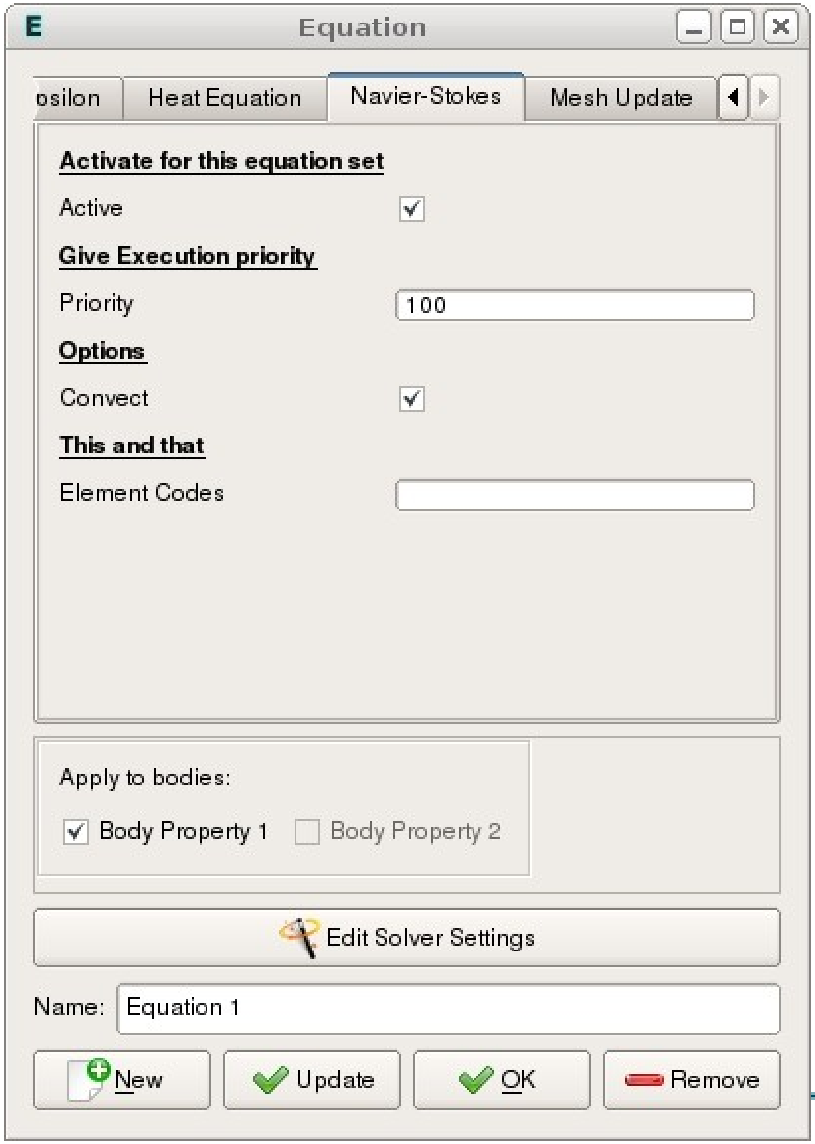
# ElmerGUI Setup

- In **Model** -> **Setup**
- Change **Simulation Type** to *Transient*
- Add *100* to **Timestep intervals**
- Choose **Timestep sizes** to *0.1*
- For the remaining options the defaults can be adopted
- Click **Apply**



# ElmerGUI Equation

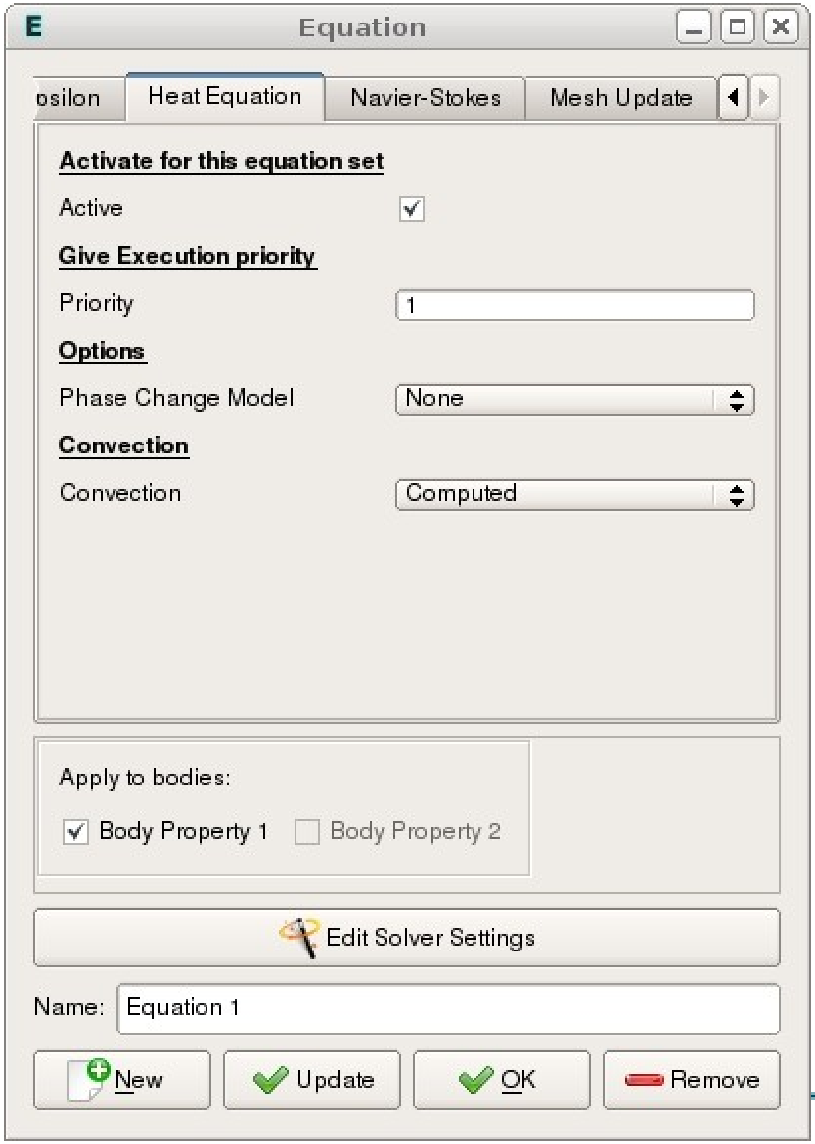
- In **Model -> Equation**
- Press **Add**
- Toggle **Body Property 1** (the part of the domain filled with fluid)
- Activate Tab **Navier-Stokes**
- Toggle **Active**
- Insert *100* in **Priority**
- Toggle **Convect**
- Then press **Edit Solver Settings:**
  - In this case we can go with the default options




CSC

# ElmerGUI Equation  ctd.

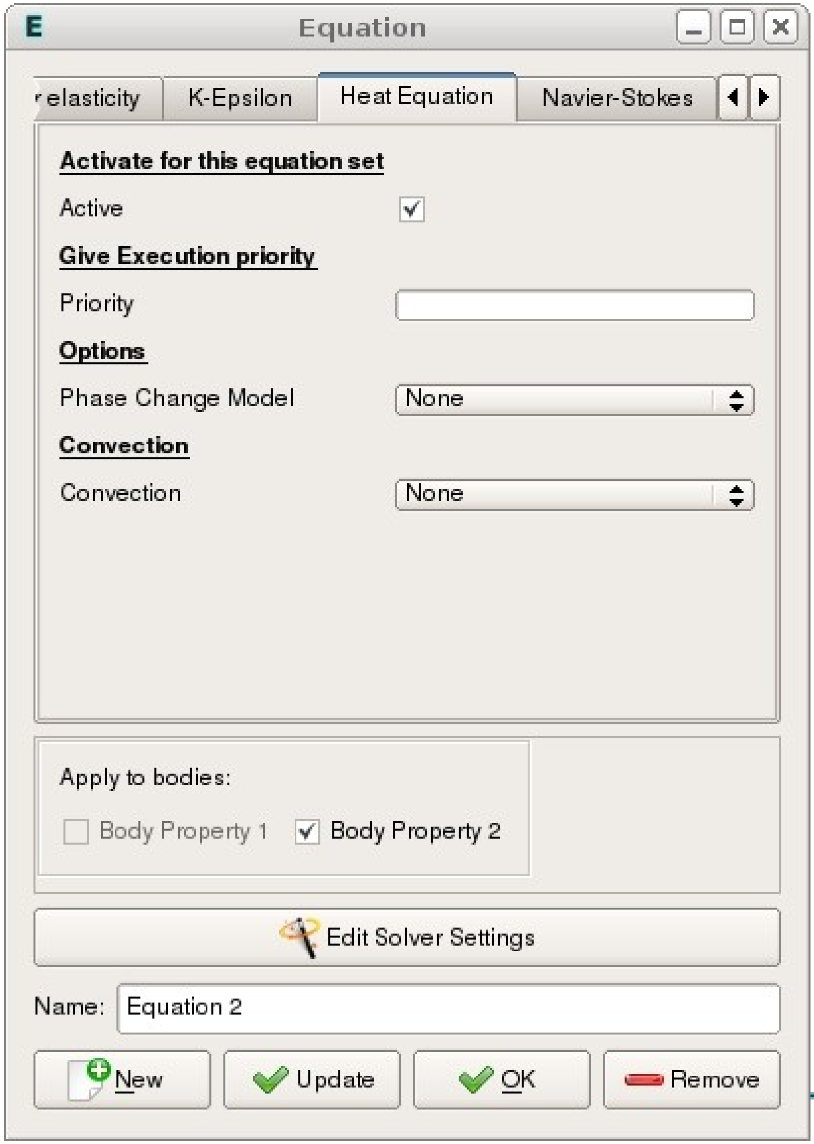
- Activate Tab **Heat Equation**
- Toggle **Active**
- Insert *1* in **Priority** (lower than 100 for Navier-Stokes!)
- Select **Convection** *Computed*
- Toggle **Body Property 1** (the part of the domain filled with fluid)
- Then press **Edit Solver Settings:**
  - We only have a linear problem, thus setting **Nonlinear system** -> **Max. Iterations** to *1*
- Click **Update (Add+** in later ElmerGUI versions)




CSC

# ElmerGUI Equation ctd.

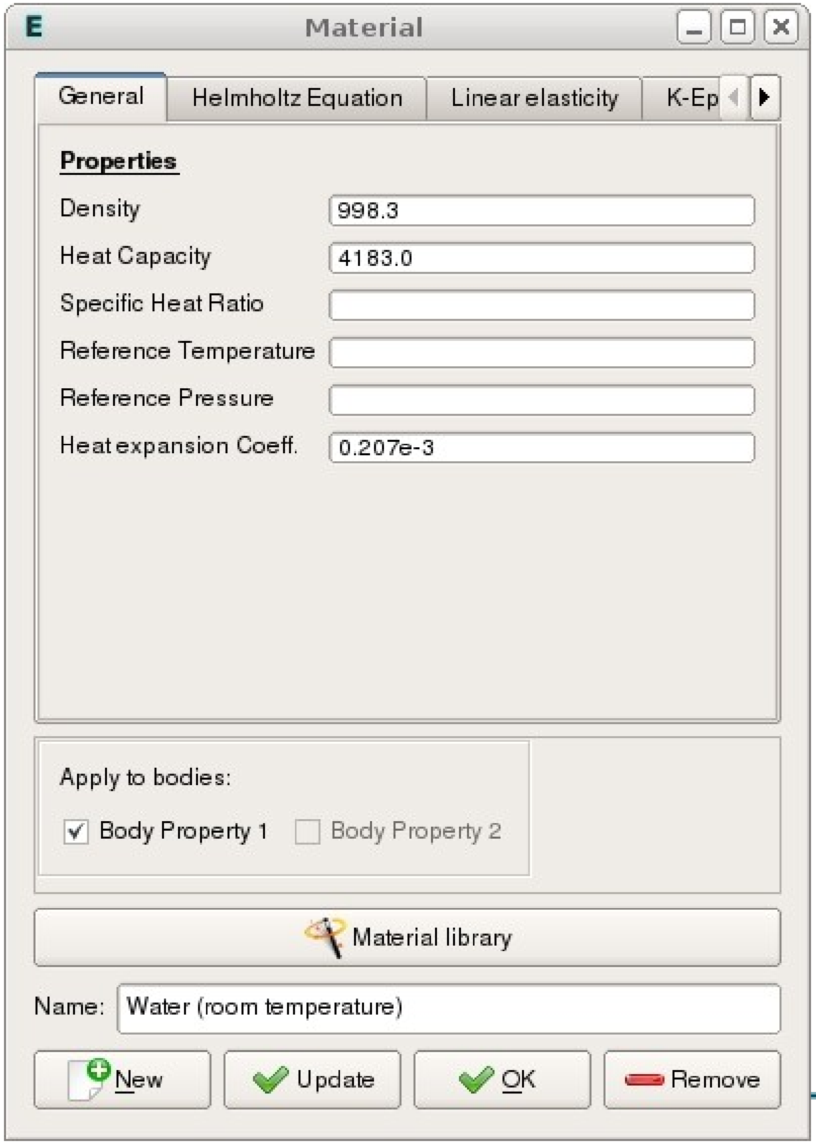
- Click **New** (to get a 2nd set of Equations)
- Toggle **Body Property 2** (the solid part of the domain)
- Activate Tab **Heat Equation**
- Toggle **Active**
- No **Priority** needed **(as only equation)**
- Select **Convection** to *None*
- Then press **Edit Solver Settings:**
  - Same settings as before
  - NB.: settings apply to all equation sets (=bodies) the solver is part of
- Click **OK** (also in still open **Equation** windows)




CSC

# ElmerGUI Material

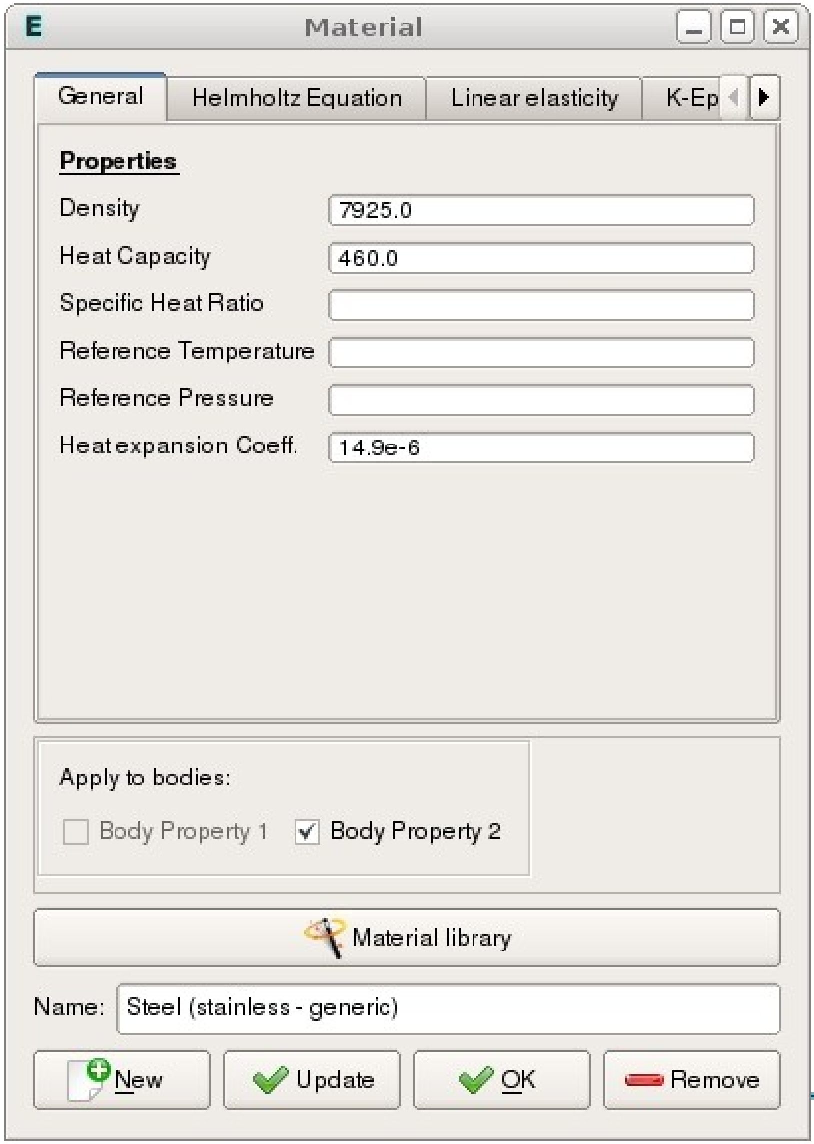
- In **Model** -> **Material**
- Press **Add**
- Toggle **Body Property 1** (the part of the domain filled with fluid)
- Then press **Material library:**
  - Choose from list: *Water (room temperature)*
- Click **Update**
- Click **New**




CSC

# ElmerGUI Material ctd.

- Toggle **Body Property 2** (the solid part of the)
- Then press **Material library:**
  - Choose from list: *Steel (stainless - generic)*
- Click **OK**

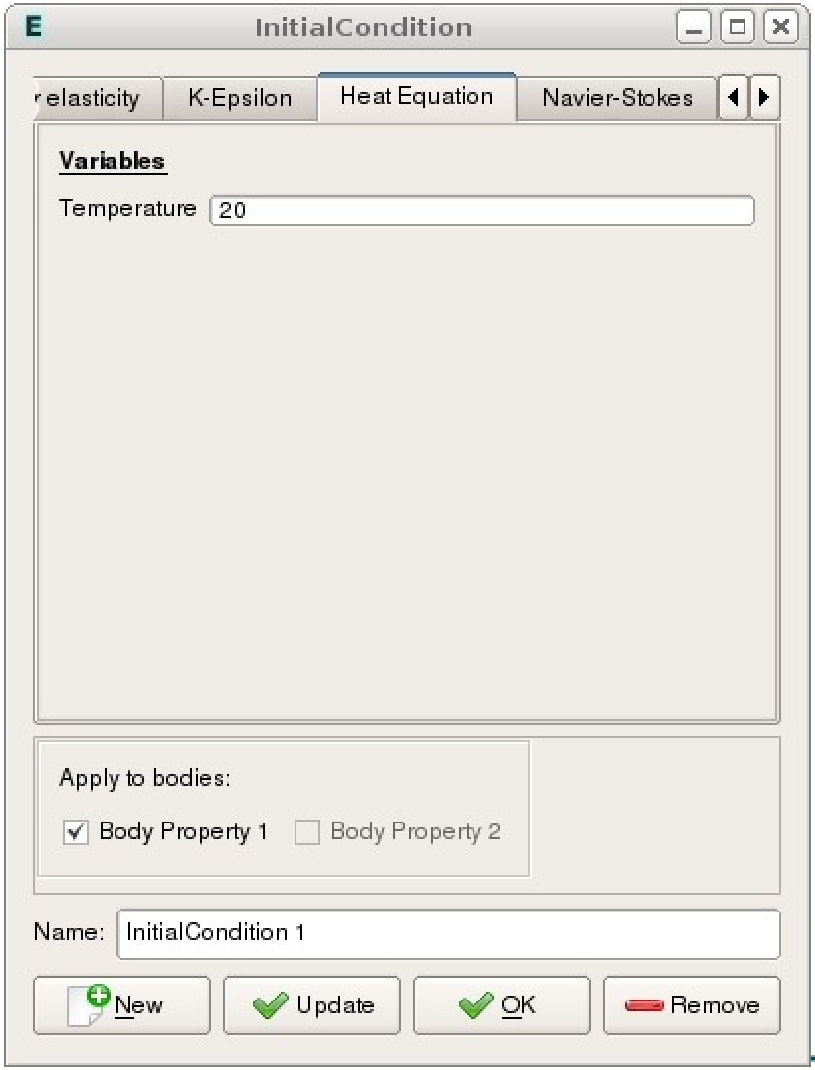


# ElmerGUI Initial Condition

- In **Model -> Initial Condition**
- Press **Add**
- Toggle **Body Property 1** (the part of the domain filled with fluid)
- Select Tab **Navier-Stokes:**
  - Set value according to r.h.s. picture
- Click **Update**

# ElmerGUI Initial Condition ctd

- Select Tab **Heat Equation:**
  - Set value according to r.h.s. picture
- Click **Update**
- Click **New**



# ElmerGUI Initial Condition ctd.

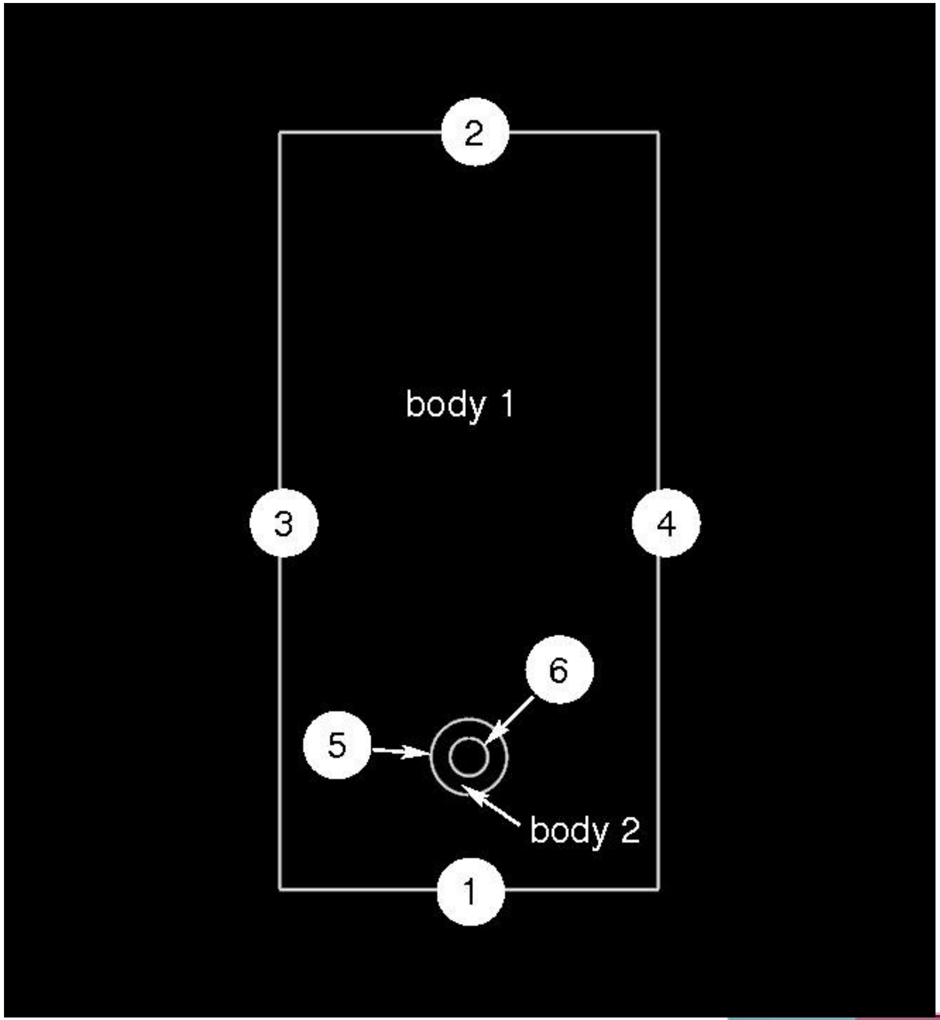
- Toggle **Body Property 2** (the solid part of the domain)
- Select Tab **Heat Equation:**
  - Set value according to r.h.s. picture
- Click **OK** (=Update + Close)


- NB.: You might want to save the stuff from time to time (you never know ...). Use this button and choose a new folder for your project

# ElmerGUI Boundary Condition

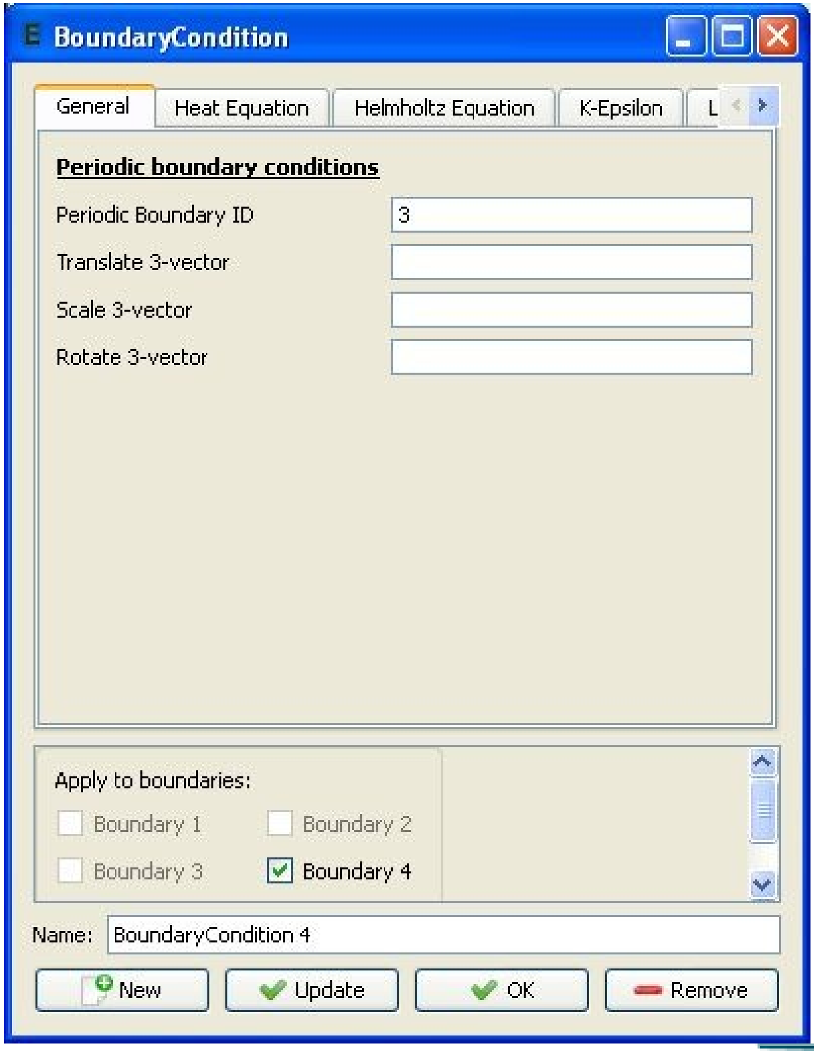
- Numbering of boundaries (see r.h.s. picture)
- First time **Add**, thereafter **Update** + **New**
- Boundary 1-5: Navier-Stokes + Heat Equation
- Boundary 6: Heat Equation only
- Boundary:

1. ”Inflow”
   - Vityeloc 1 = 0.0
     Velocity 2 = 0.05
   - Temperature = 20
1. ”Outflow”
   - Natural boundary conditions (= set nothing)
1. ”left”: nothing
2. ”right”: periodic of left (see next slide)



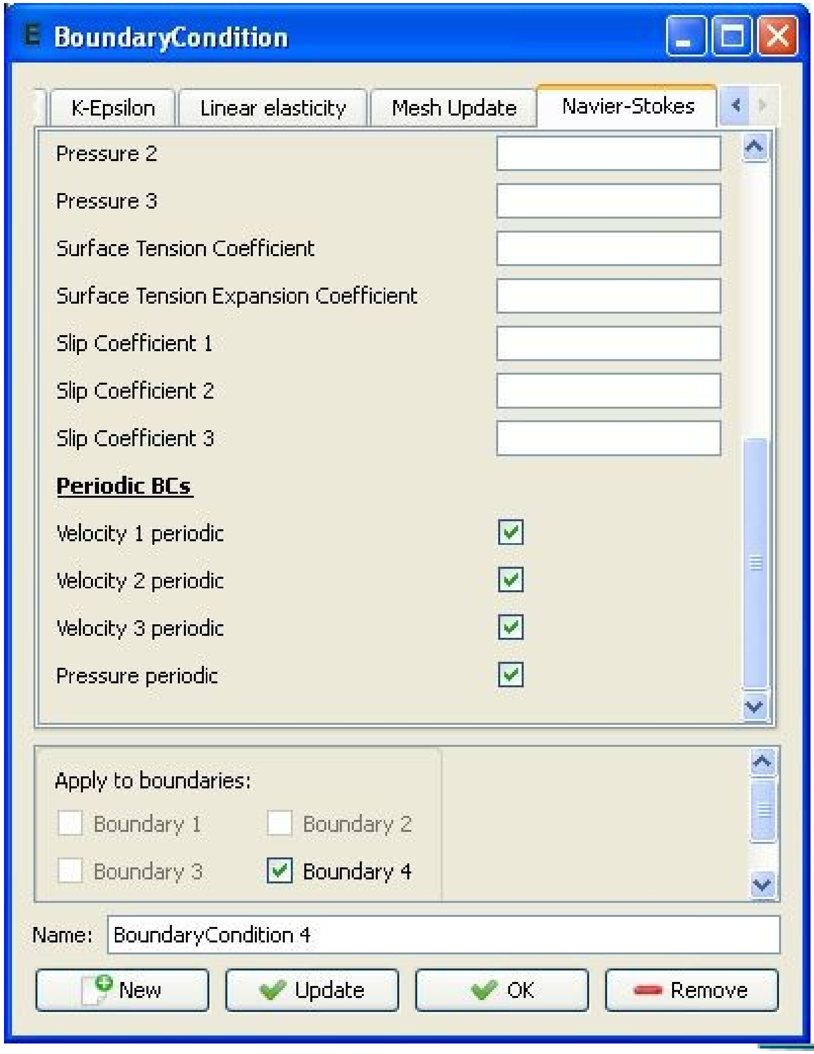
# ElmerGUI Boundary Condition ctd.

- In **Model -> Boundary Condition**
- Press **Add**
- Toggle **Boundary 4** (the right side of the domain)
- Insert **Periodic Boundary ID** *3*
  - This links boundary 4 to boundary 3



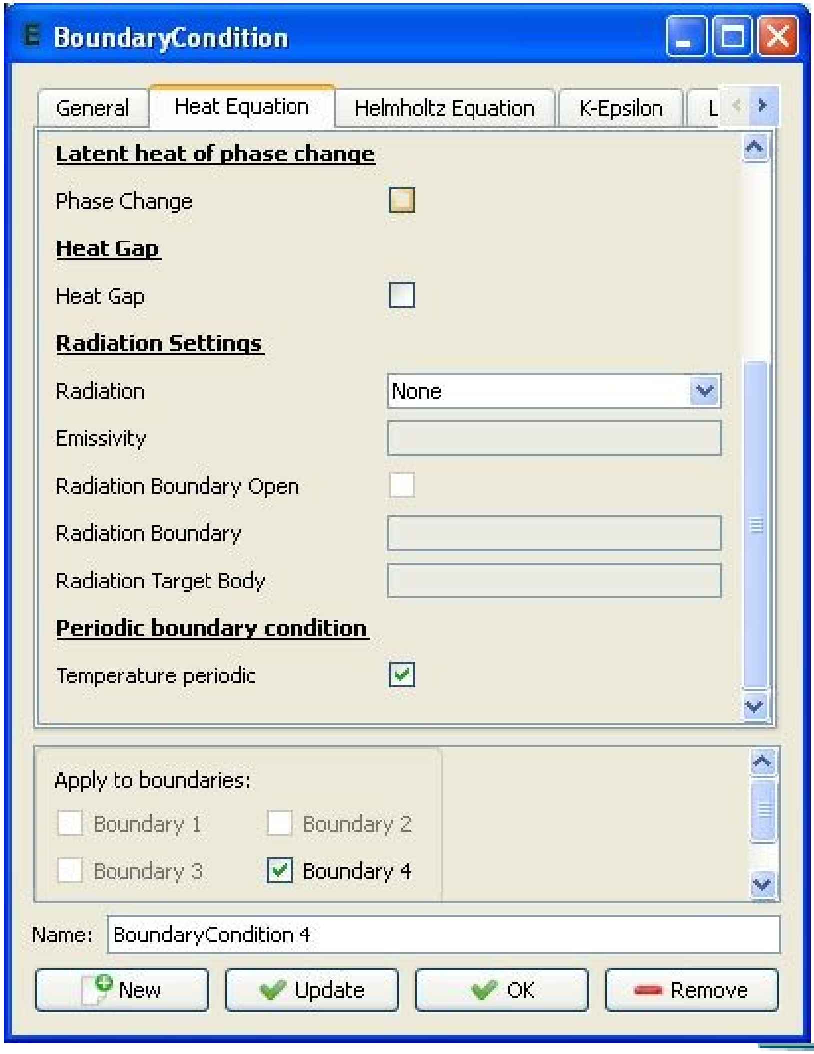
# ElmerGUI Boundary Condition ctd.

- Click Tab **Navier-Stokes**
  - Toggle all Variables in **Periodic BC's**
- Click **Update**



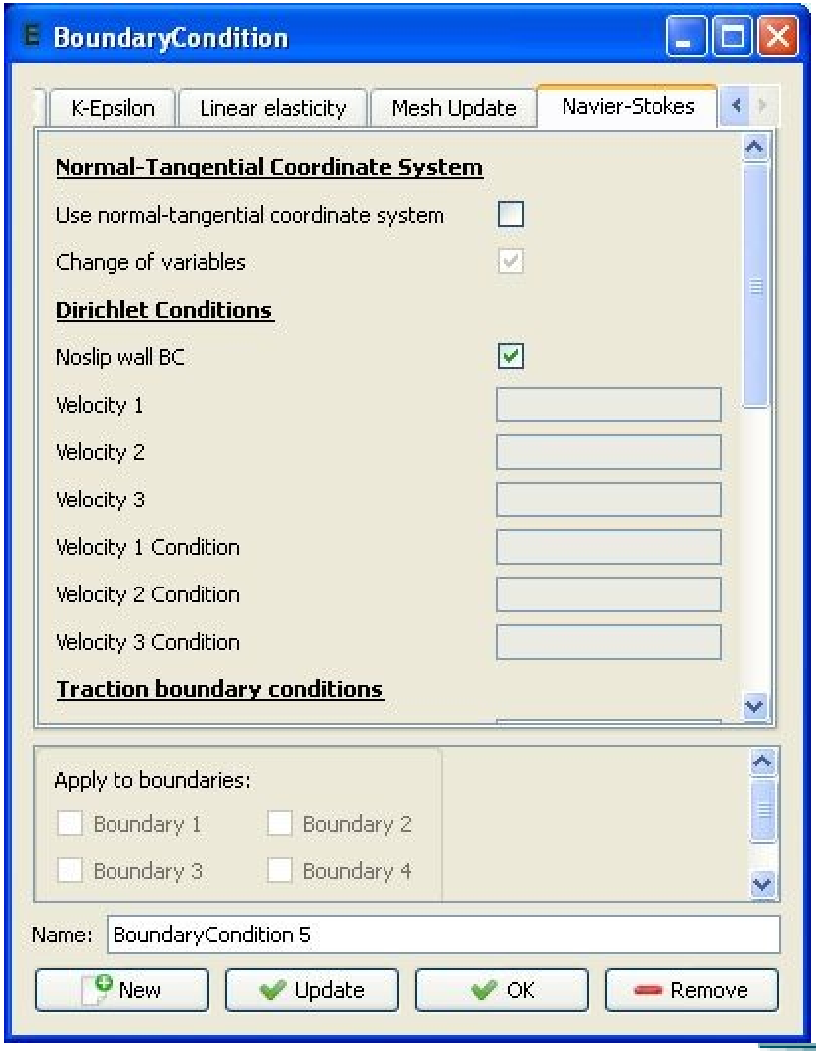
# ElmerGUI Boundary Condition ctd.

- Click Tab **Heat Equation**
  - Toggle all Variables in **Periodic BC's**
- Click **Update**
- Click **New**



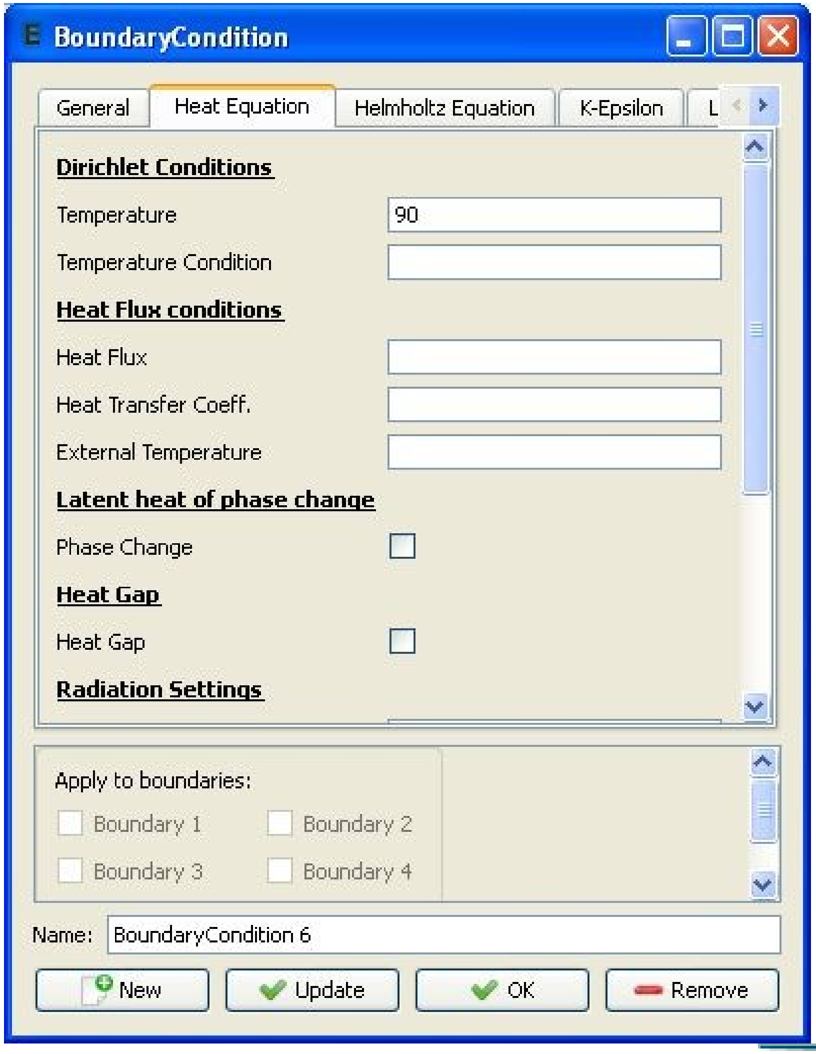
# ElmerGUI Boundary Condition ctd.

- Toggle **Boundary 5** (the outer cylinder wall)
- Activate Tab **Navier-Stokes**
- Toggle **Noslip wall BC**
  - Heat Equation: natural BC
- Click **Update**
- Click **New**



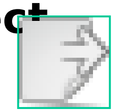
# ElmerGUI Boundary Condition ctd.

- Toggle **Boundary 6** (the inner cylinder wall)
- Activate Tab **Heat Equation**
- Temperature = *90*
- Nothing for Navier-Stokes (solid body)
- Click **OK**



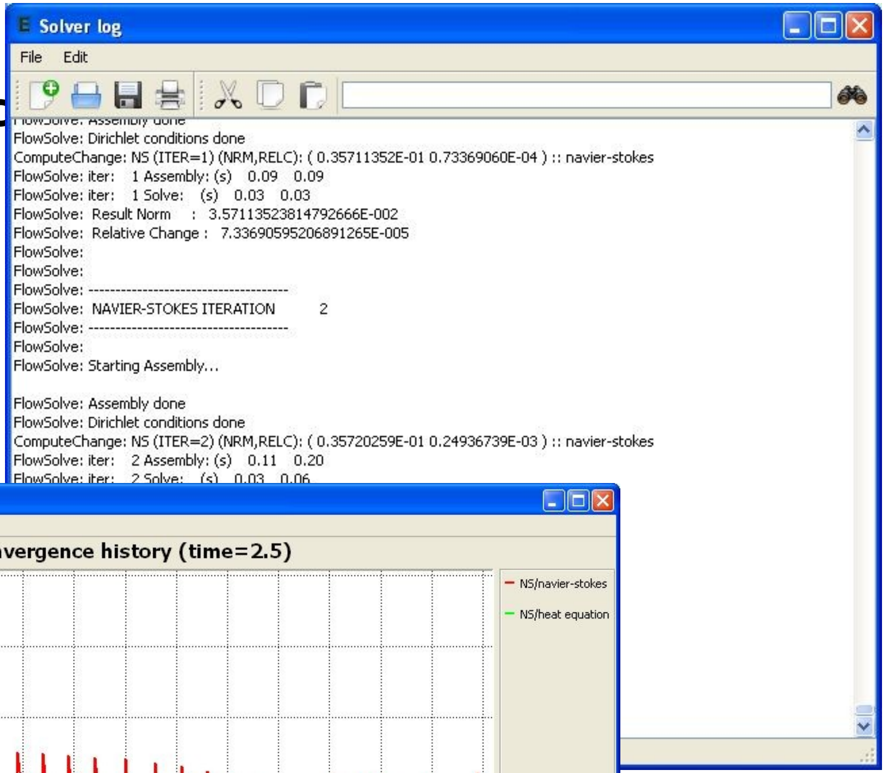

CSC

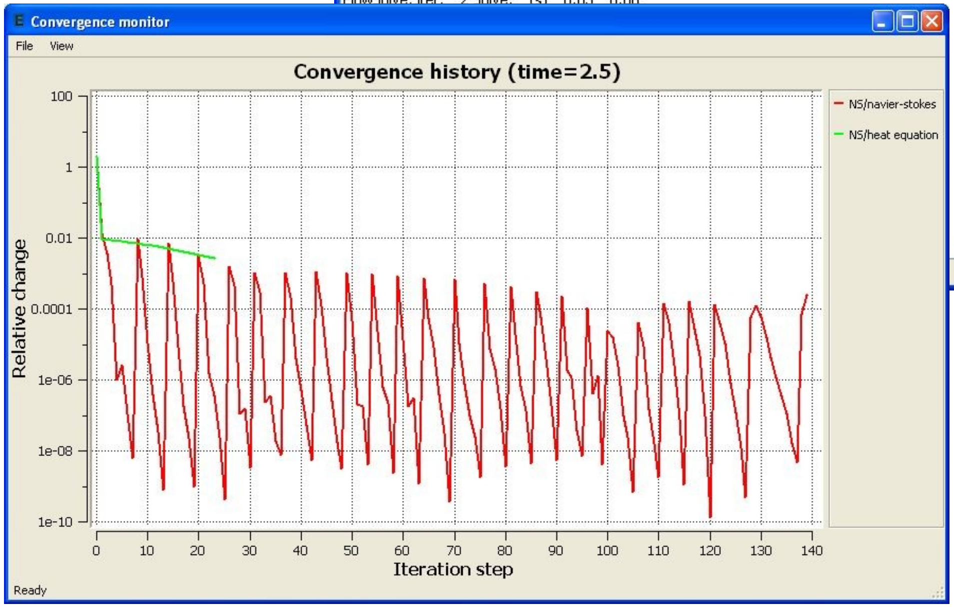
# ElmerGUI Starting the run

- Do a **Sif -> Generate**
- <u>Save the project first</u>!
  - either by **File** -> **Save Project**
  - **or** by pressing the symbol 
  - The earlier given location (folder) will be suggested: confirm with **OK**
- Check your settings: **Model** -> **Summary ...**
- Launch the run by 
  - Either pressing the symbol
  - Or by **Run** -> **Start Solver**
  - The symbol should change its color 
  

# ElmerGUI Starting the run ctd.

- A **Solver log** and a **Convergence history** window will occur
- If something apparently goes wrong: **Run -> Kill Solver**

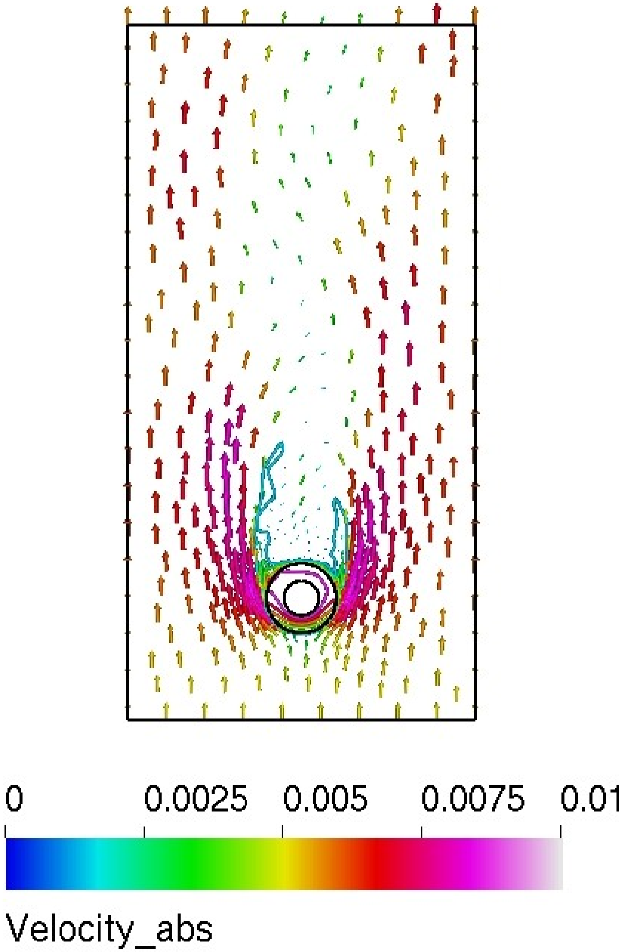


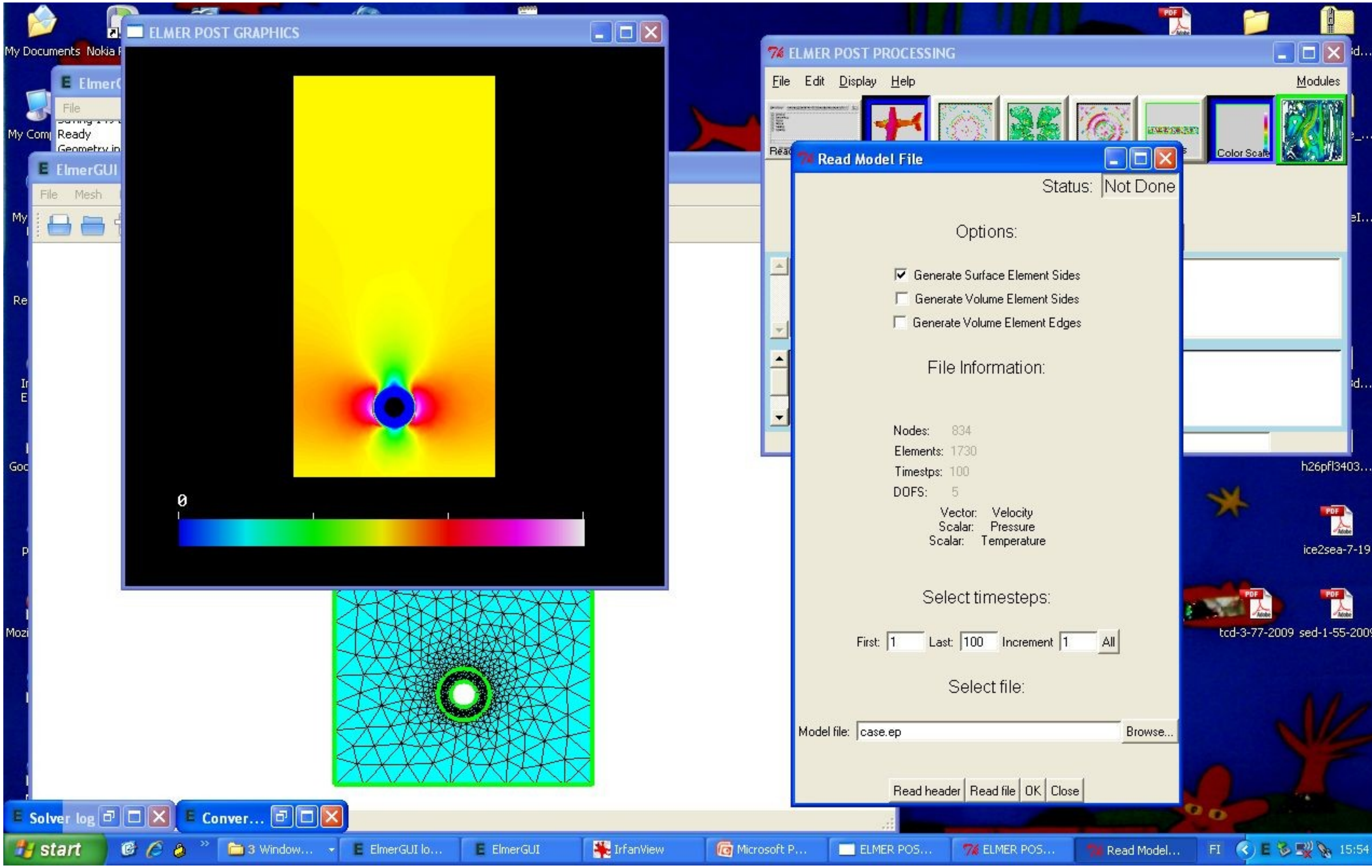
# ElmerGUI Postprocessing

- Upon getting a converged result
  - Either launch ElmerPost by clicking the symbol
  - Or **Run** -> **Start Postprocessor**

- Then ElmerPost launches
  - ElmerPost will render the absolut value of the velocity by default
  - NB: in case of transient runs, unfortunately ElmerGUI lets ElmerPost only load the first timestep
  - Remedy: reload with (in ElmerPost) **File** -> **Read** and press the **All** buton + **OK** thereafter

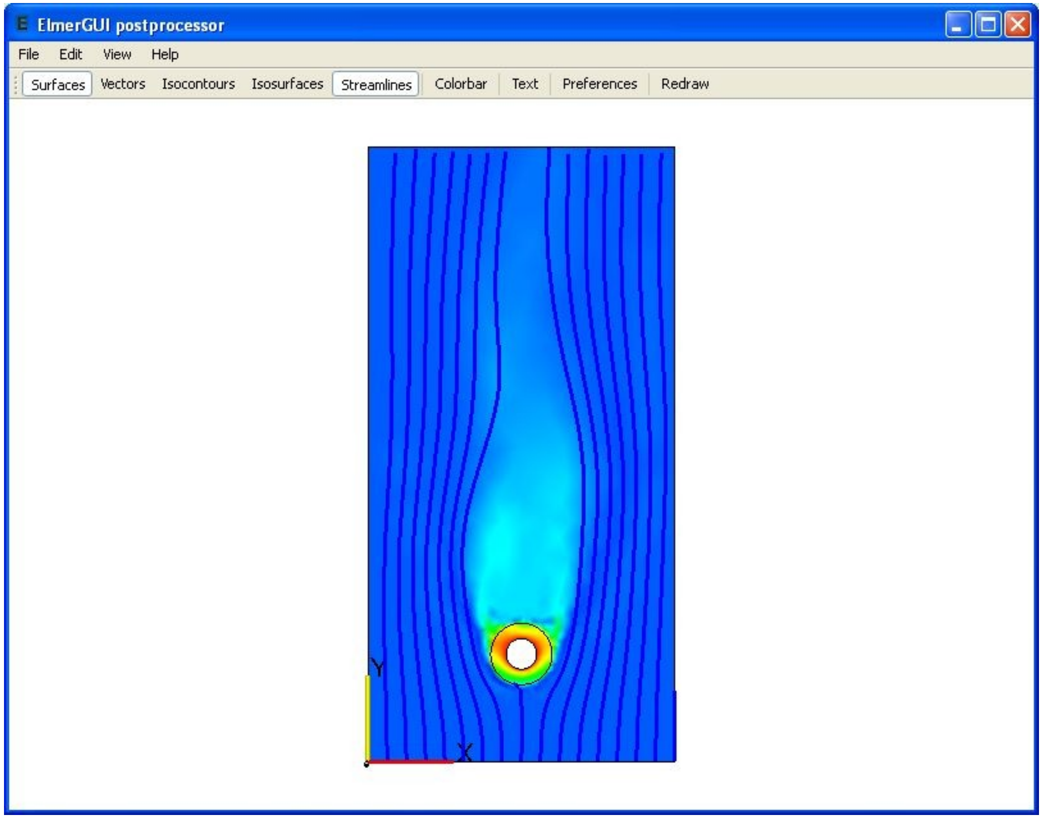


# ElmerGUI Postprocessing ctd.



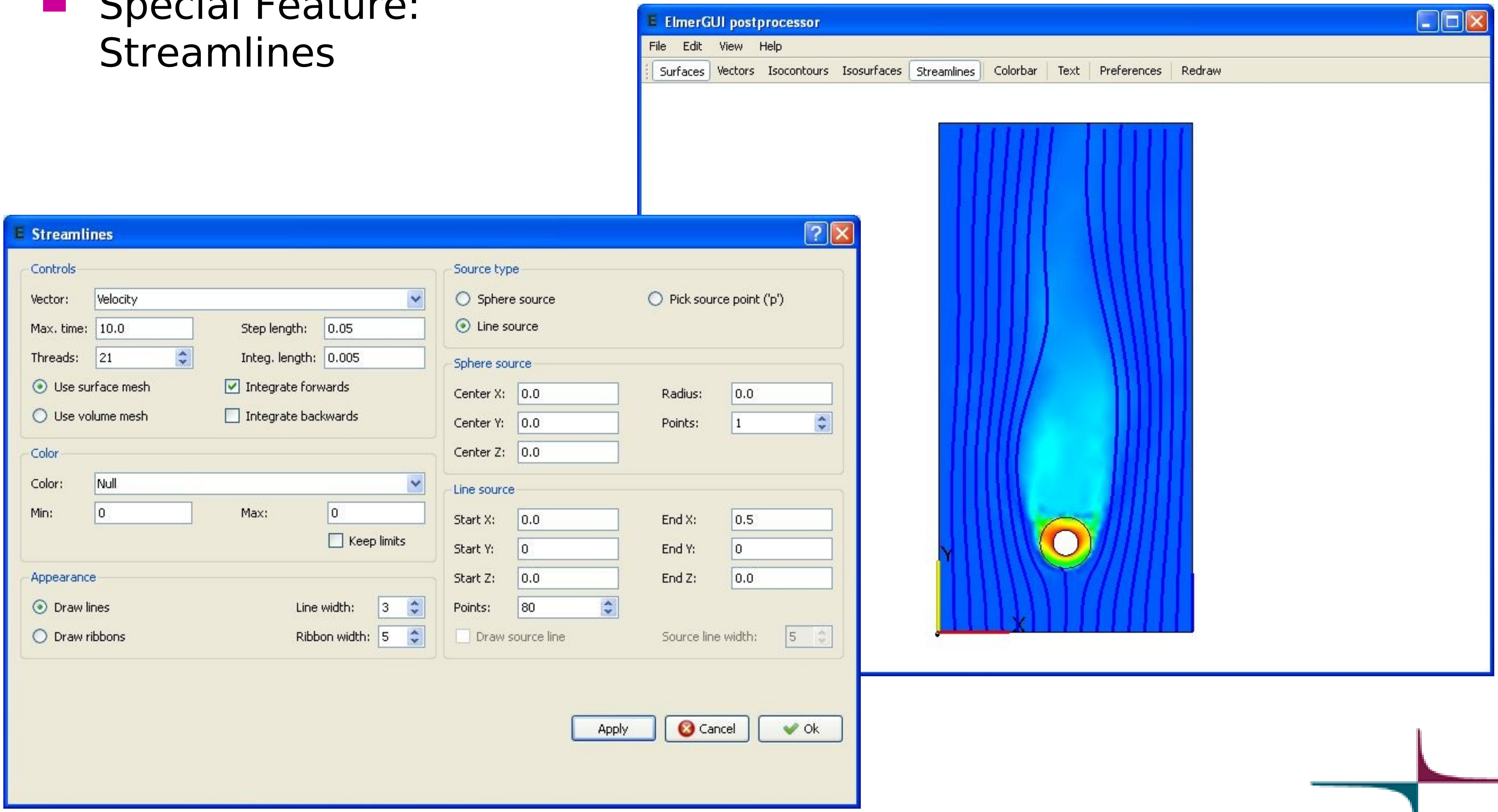
# ElmerGUI Postprocessing VTK

- ElmerGUI VTK based built-in postprocessor
  - **Run Postprocessor (VTK)**
- Then the VTK postprocessor launches
  - No variable by default
  - NB: in case of transient runs, unfortunately ElmerGUI lets the VTK postprocessor only load the first timestep
  - Remedy: reload with (in ElmerPost) **File -> Read** and press the **All** buton + **OK** thereafter



# ElmerGUI Postprocessing VTK

- Special Feature: Streamlines



# Exercises

- Change into steady state run
- Change the heat transfer through boundary 6:
  1. Heat transfer coefficient
  2. Heat flux
- Increase inflow velocity - how far can you go? Check the Reynolds number