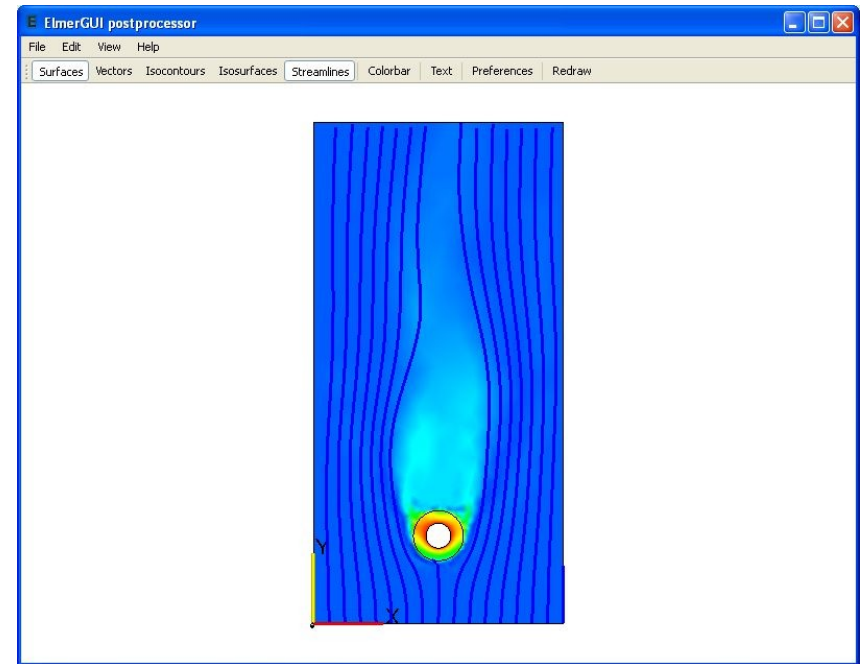
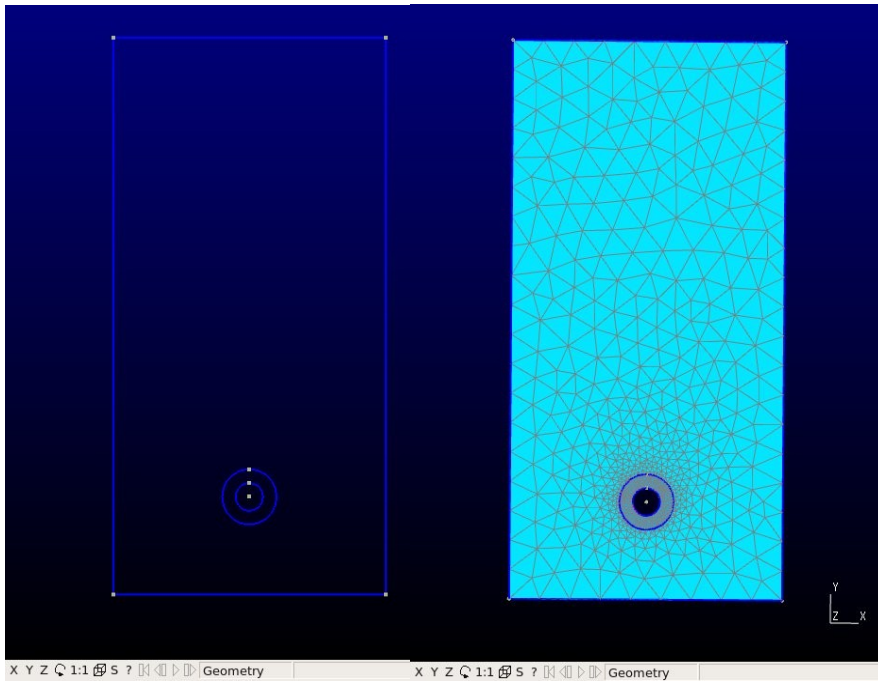
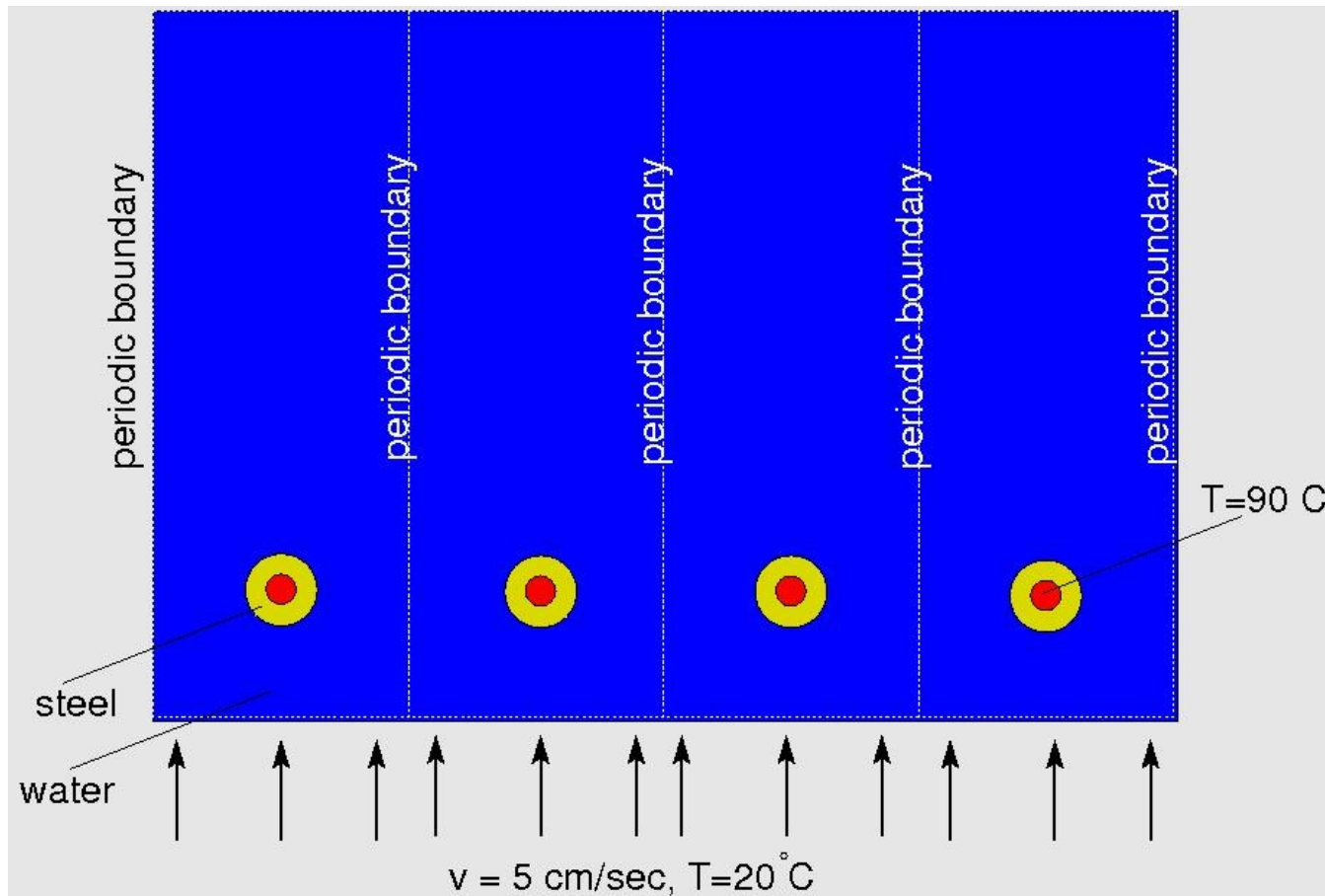


# Introductory Example

Example of coupled flow and heat transfer problem using ElmerGUI



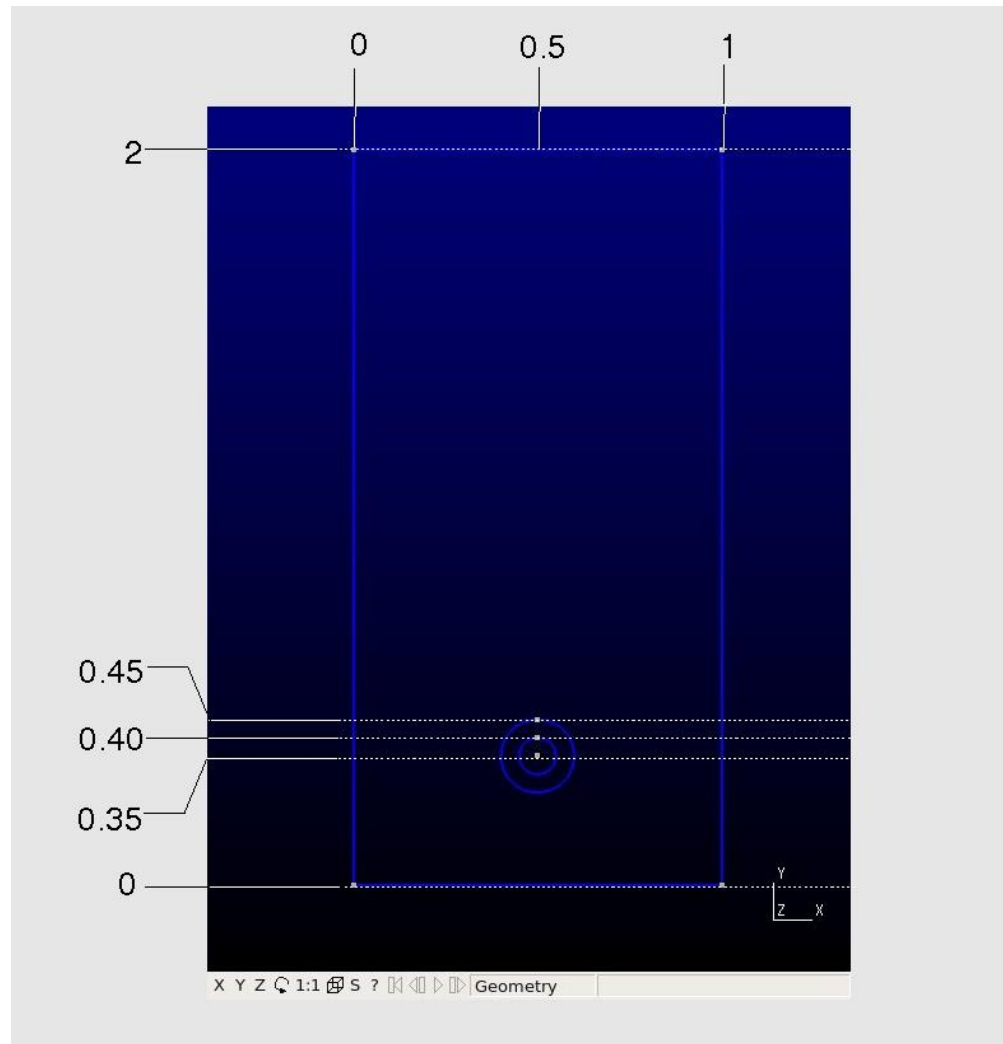
# Problem Outline



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

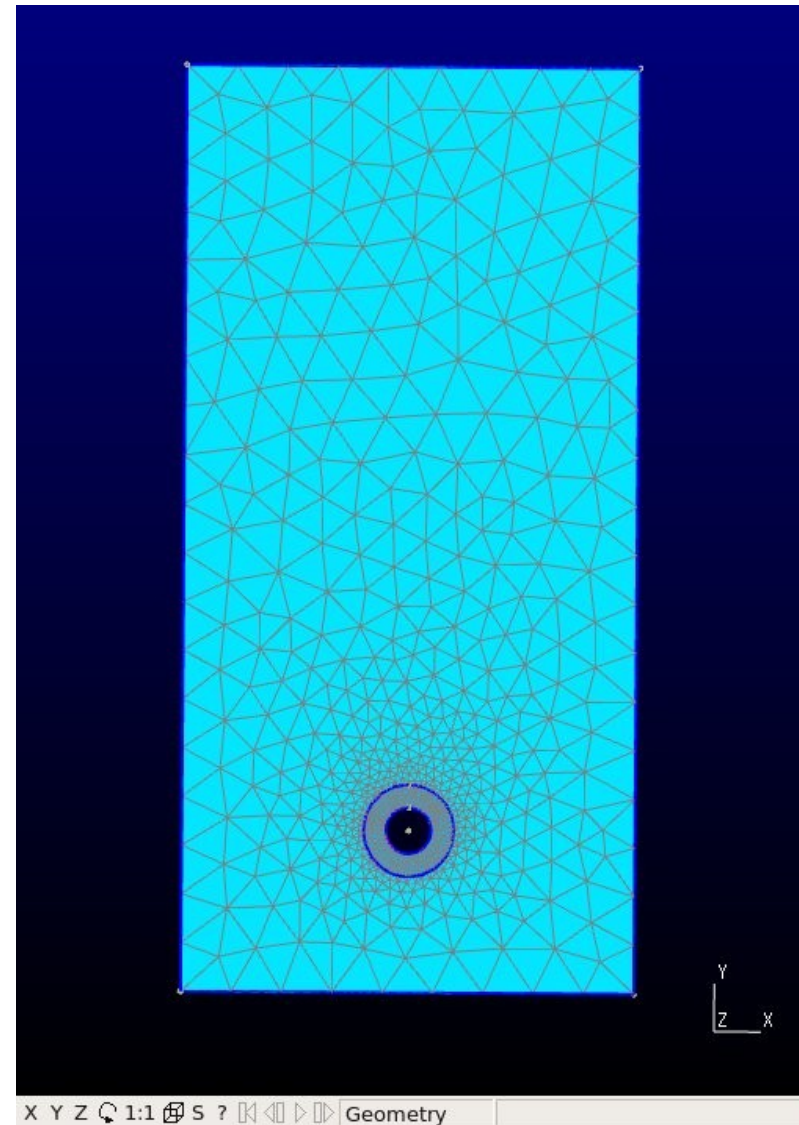
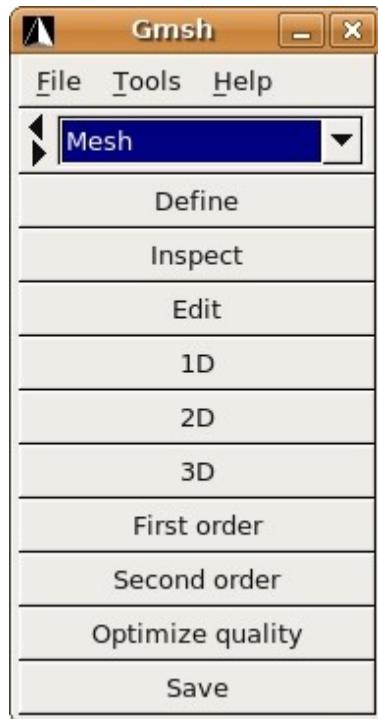
# Mesh

- 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**



# 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:  

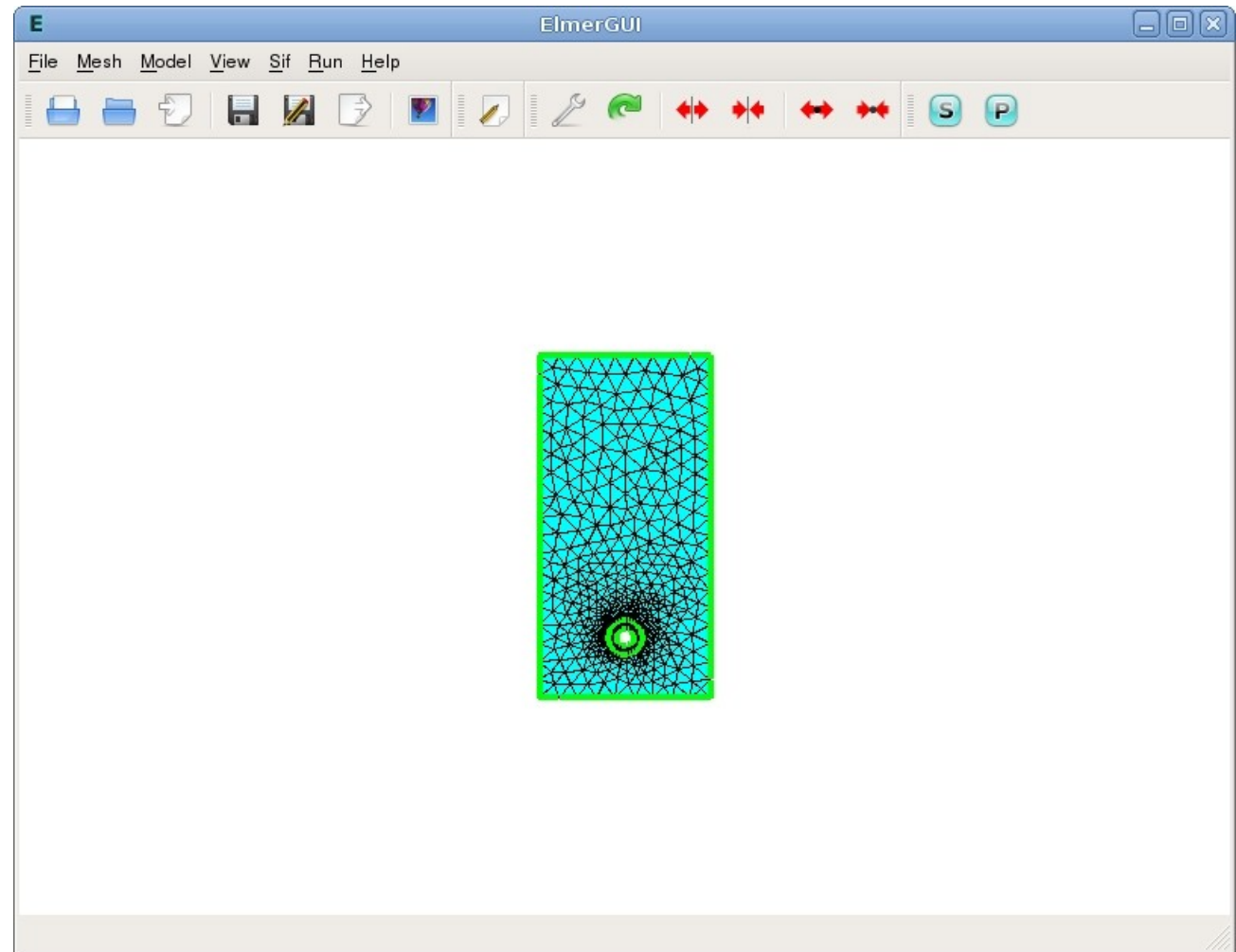
```
ElmerGrid 14 3 periodiccyl.msh -autoclean -scale 0.1 0.1 1.0
```
- This directly creates the mesh in ElmerSolver format:  

```
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**

**Setup**

Header

Check keywords warn

MeshDB

Include path

Results directory

Simulation

Max. output level: 4    Steady state max. iter: 1

Coordinate system: Cartesian    Timestepping method: BDF

Coordinate mapping: 1 2 3    BDF order: 1

Simulation type: Transient    Timestep intervals: 100

Output intervals: 1    Timestep sizes: 0.1

Solver input file: case.sif

Post file: case.ep

Constants

Gravity: 0 -1 0 9.82

Stefan Boltzmann: 5.67e-08

Vacuum permittivity: 8.8542e-12

Boltzmann: 1.3807e-23

Unit charge: 1.602e-19

Apply



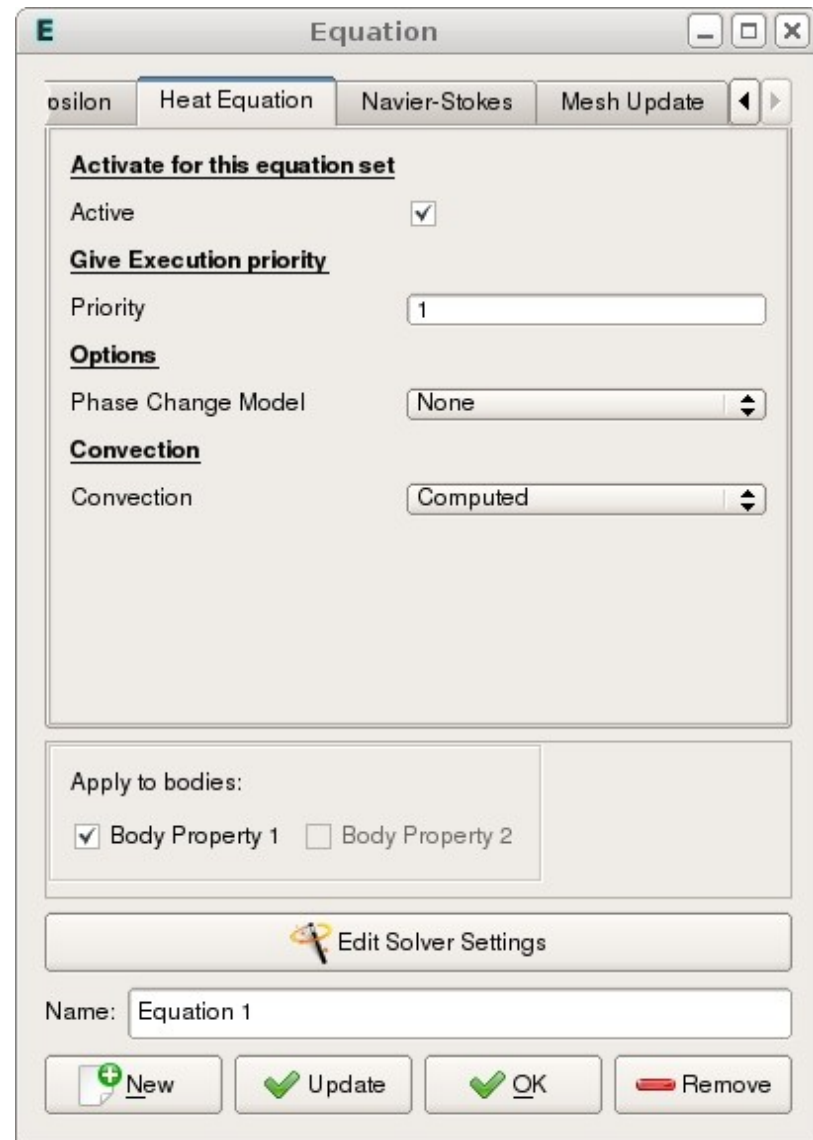
# ElmerGUI Equation

- 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



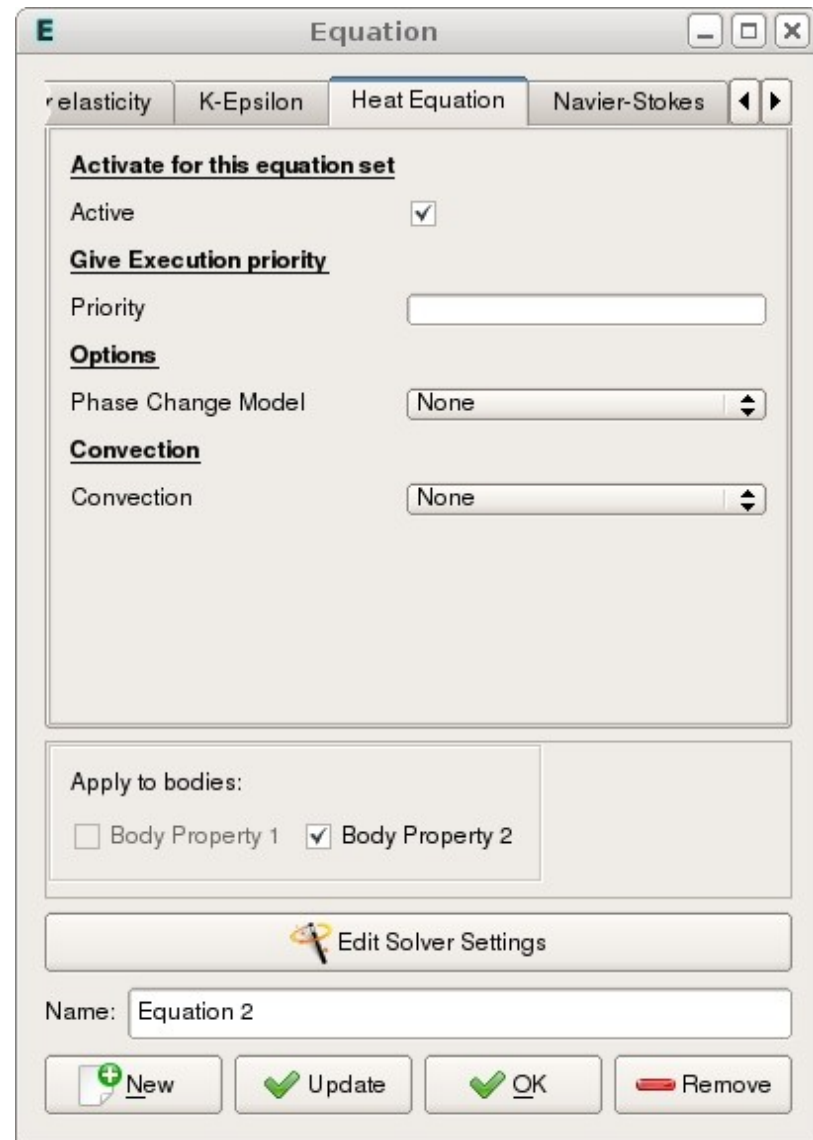
# ElmerGUI Equation ctd.

- 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)**



# 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)



# 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**

The screenshot shows the 'Material' dialog box in ElmerGUI. The 'General' tab is selected, displaying the following properties:

Property	Value
Density	998.3
Heat Capacity	4183.0
Specific Heat Ratio	
Reference Temperature	
Reference Pressure	
Heat expansion Coeff.	0.207e-3

Below the properties, the 'Apply to bodies' section is checked for 'Body Property 1'. The 'Material library' button is visible, and the 'Name' field contains 'Water (room temperature)'. The bottom of the dialog features buttons for 'New', 'Update', 'OK', and 'Remove'.

# ElmerGUI Material ctd.

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

The screenshot shows the 'Material' dialog box in ElmerGUI. The 'General' tab is selected, displaying the following properties:

Property	Value
Density	7925.0
Heat Capacity	460.0
Specific Heat Ratio	
Reference Temperature	
Reference Pressure	
Heat expansion Coeff.	14.9e-6

Below the properties, the 'Apply to bodies:' section shows that 'Body Property 2' is selected (checked).

The 'Material library' button is visible, and the 'Name:' field contains 'Steel (stainless - generic)'. The bottom of the dialog features buttons for 'New', 'Update', 'OK', and 'Remove'.

# 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**




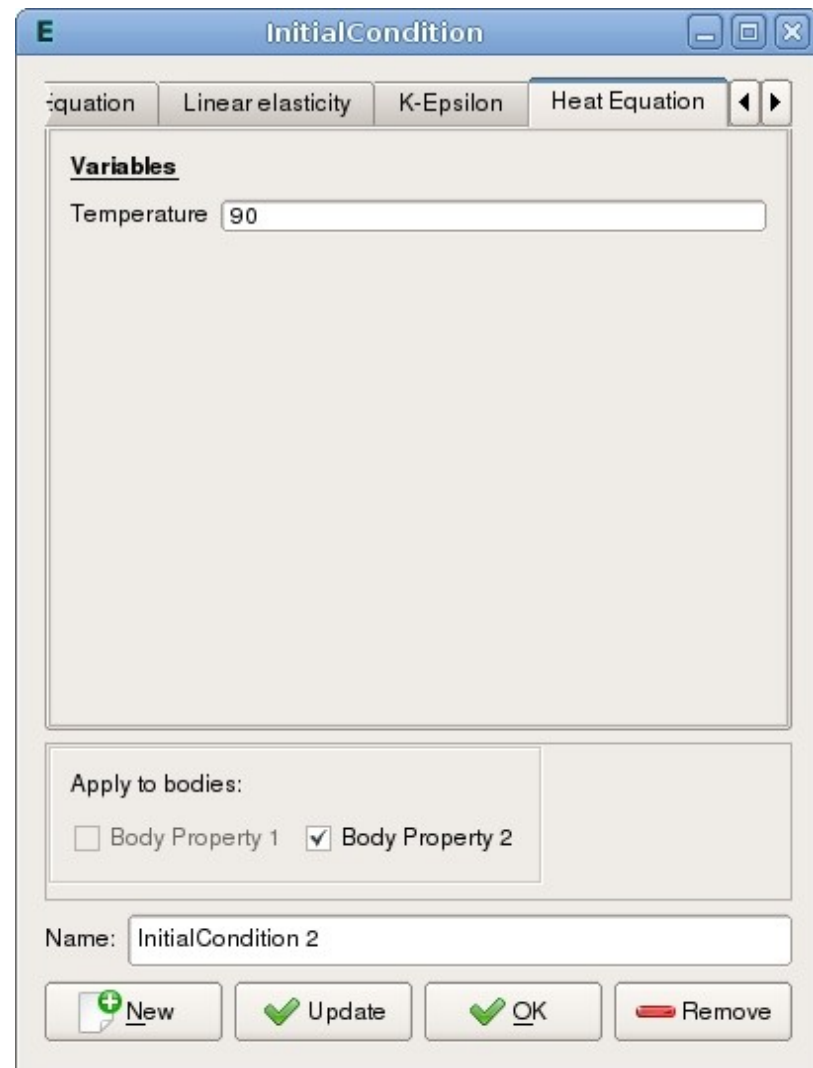
# ElmerGUI Initial Condition ctd

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



# ElmerGUI Initial Condition ctd.

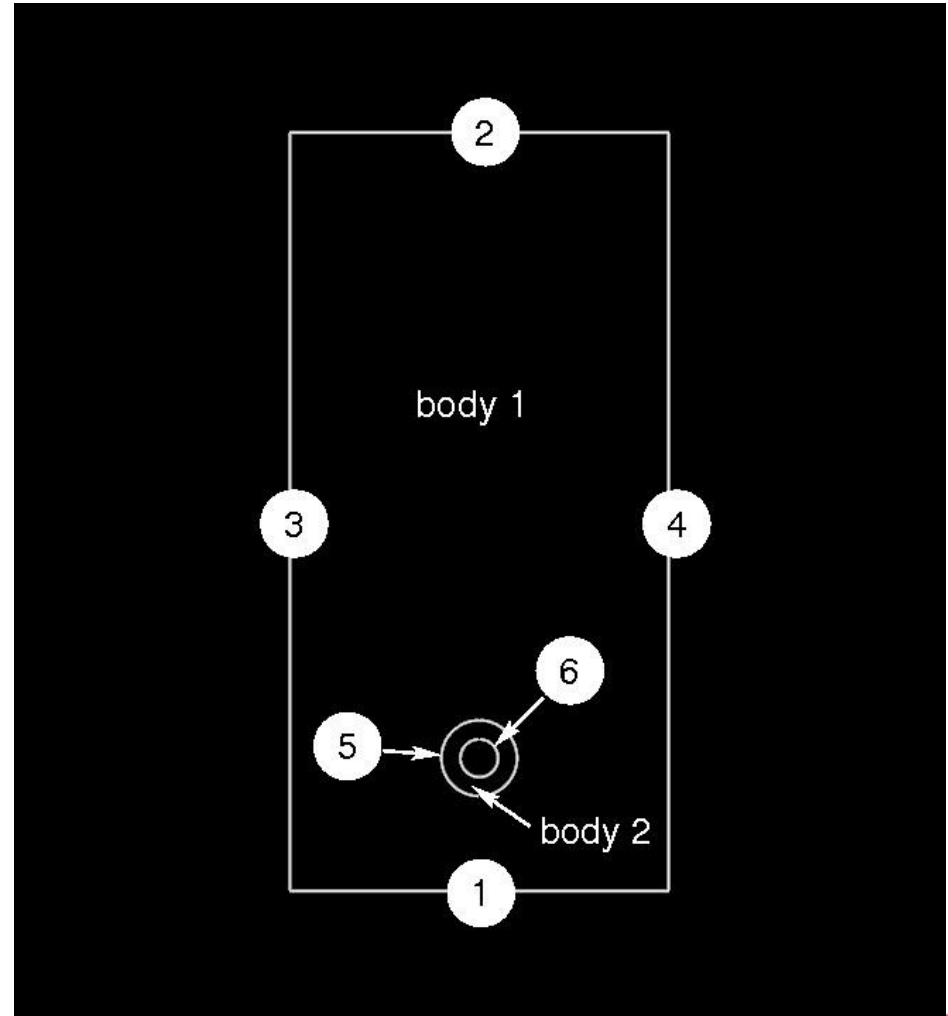
- Toggle **Body Property 2** (the solid part of the domain)
- Select Tab **Heat Equation**:
  - Set value according to r.h.s. picture
- Click **OK** (=Update + Close)
  
- NB.: You might want to save the stuff from time to time (you never know ...). Use this button  and choose a new folder for your project





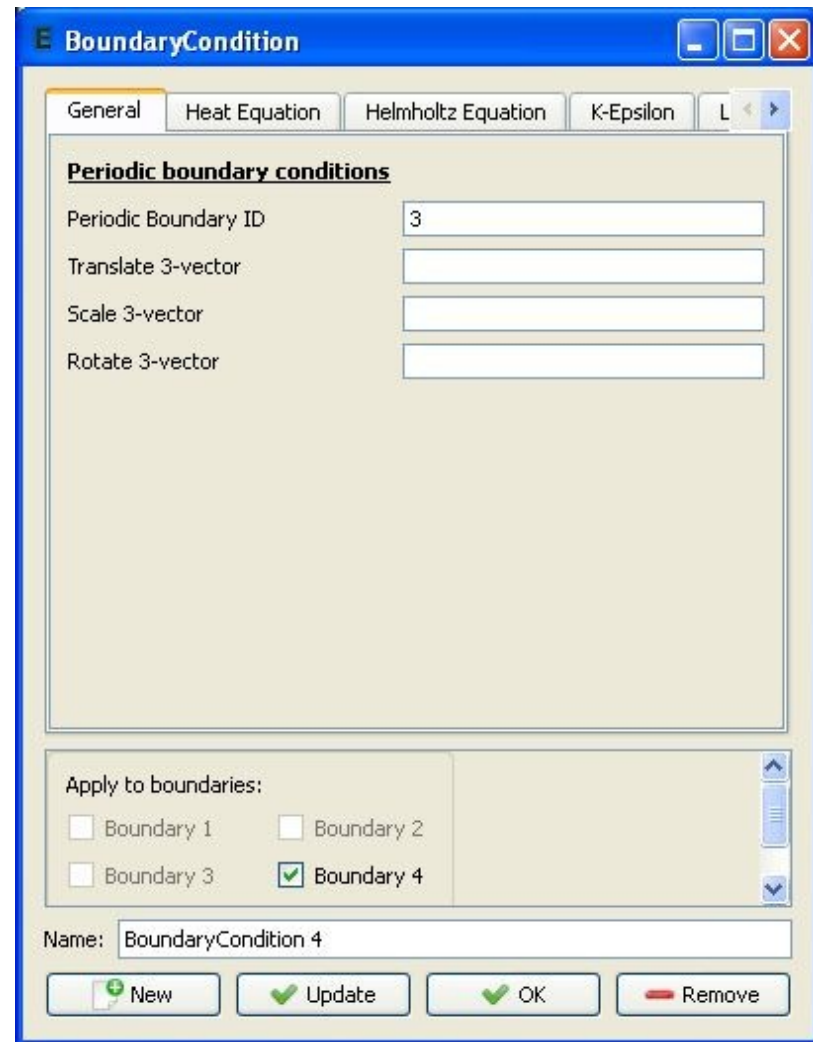
# ElmerGUI Boundary Condition

- 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)



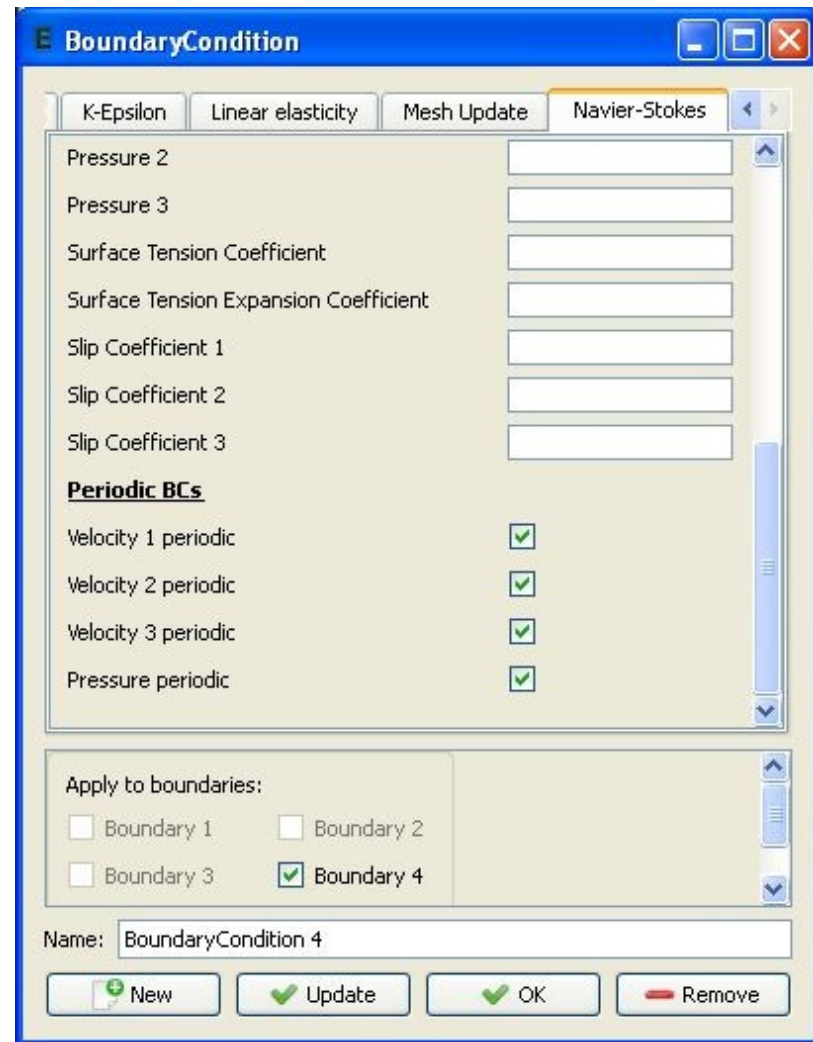
# ElmerGUI Boundary Condition ctd.

- 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



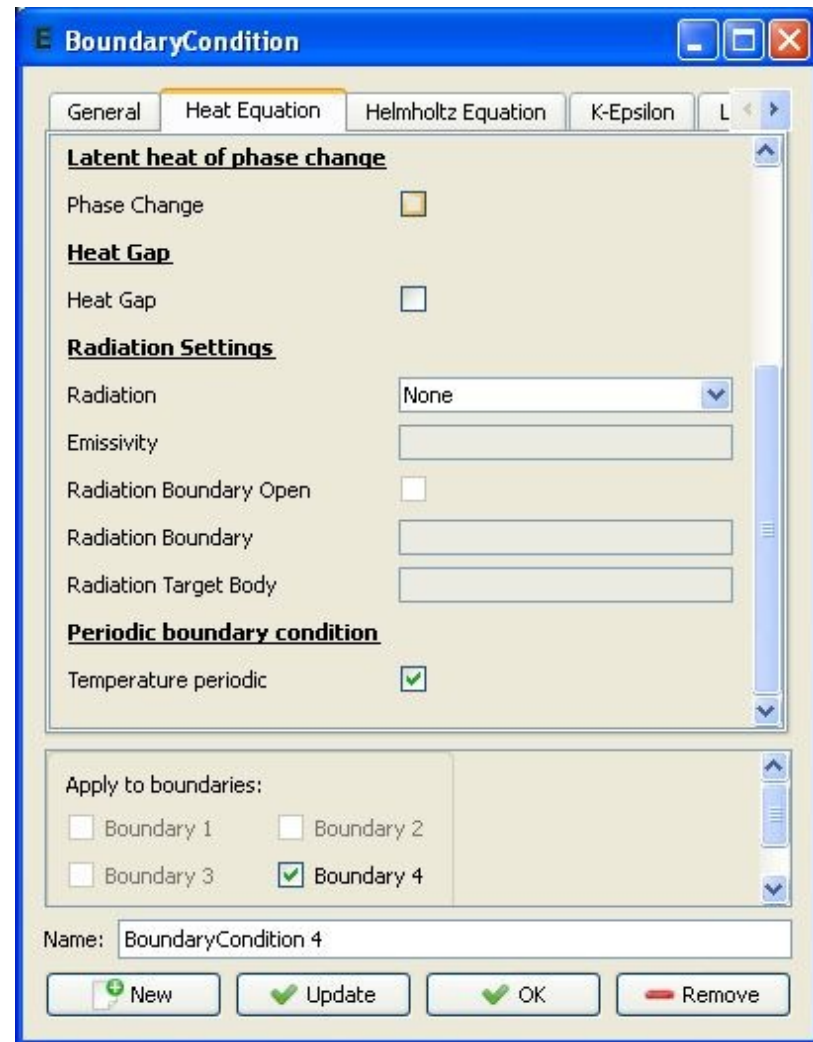
# ElmerGUI Boundary Condition ctd.

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



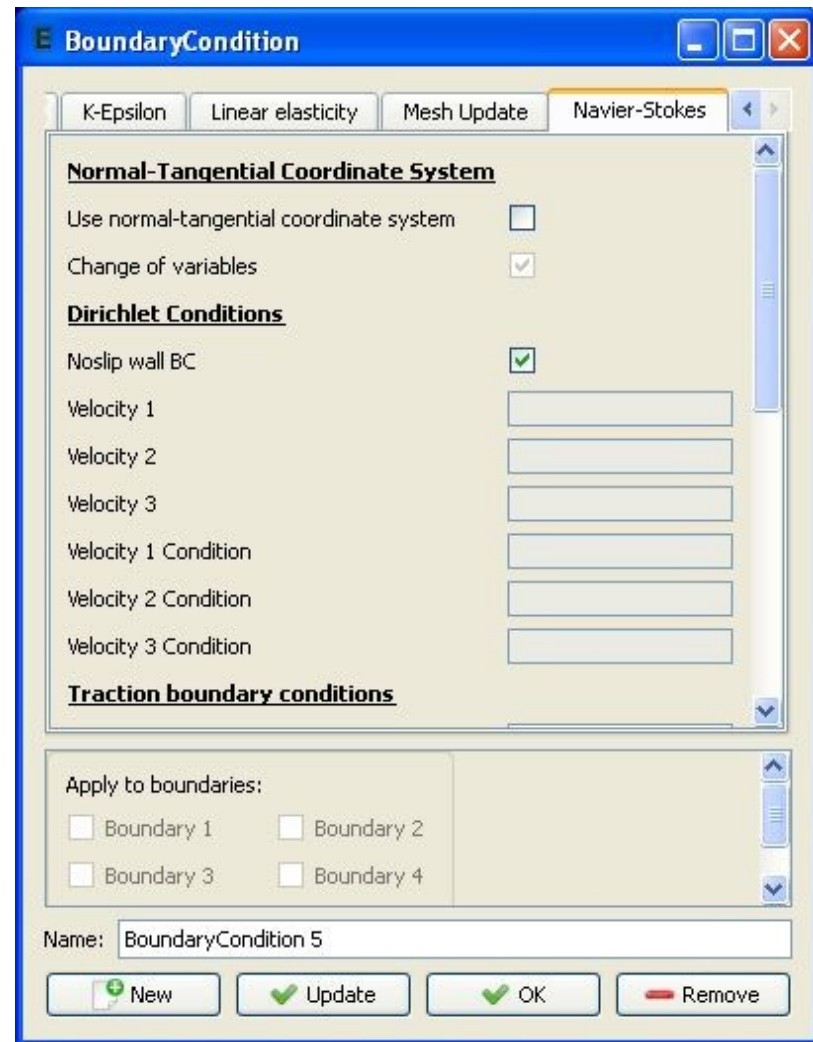
# ElmerGUI Boundary Condition ctd.

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



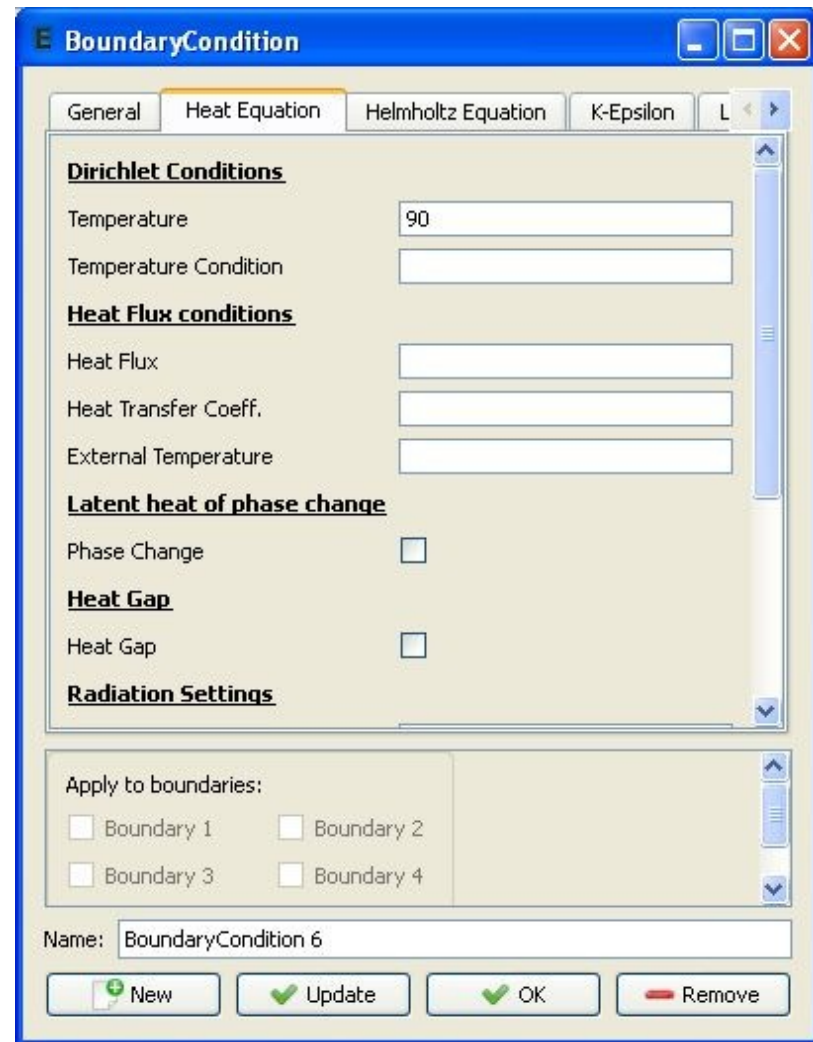
# ElmerGUI Boundary Condition ctd.

- 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**



# ElmerGUI Starting the run

- Do a **Sif -> Generate**
- Save the project first!
  - 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 

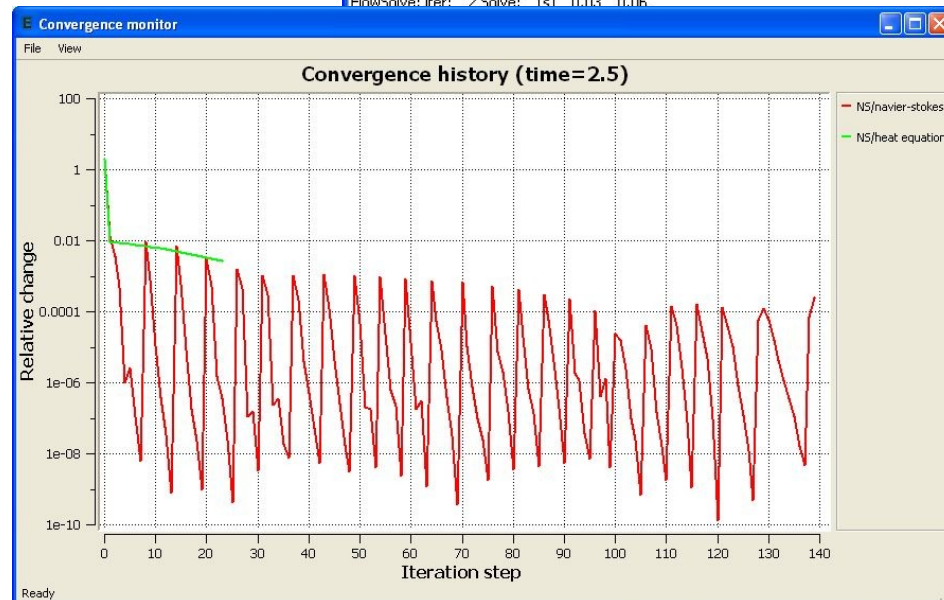
# ElmerGUI Starting the run ctd.

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




```
Solver log
File Edit
FlowSolve: Assembly done
FlowSolve: Dirichlet conditions done
ComputeChange: NS (ITER=1) (NRM,REL): ( 0.35711352E-01 0.73369060E-04 ) :: navier-stokes
FlowSolve: iter: 1 Assembly: (s) 0.09 0.09
FlowSolve: iter: 1 Solve: (s) 0.03 0.03
FlowSolve: Result Norm : 3.57113523814792666E-002
FlowSolve: Relative Change : 7.33690595206891265E-005
FlowSolve:
FlowSolve:
FlowSolve: -----
FlowSolve: NAVIER-STOKES ITERATION      2
FlowSolve: -----
FlowSolve:
FlowSolve: Starting Assembly...

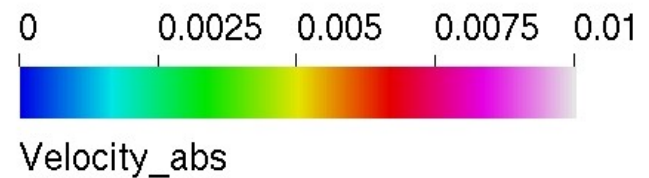
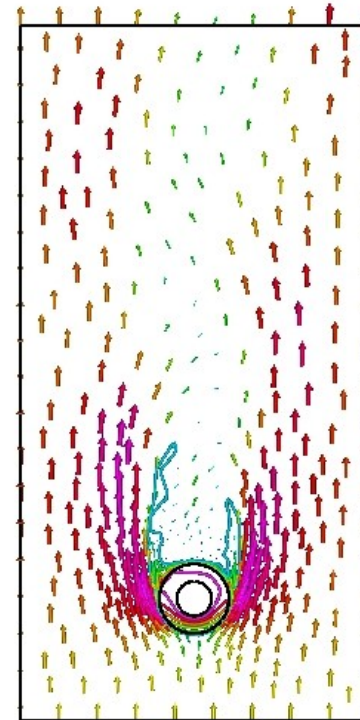
FlowSolve: Assembly done
FlowSolve: Dirichlet conditions done
ComputeChange: NS (ITER=2) (NRM,REL): ( 0.35720259E-01 0.24936739E-03 ) :: navier-stokes
FlowSolve: iter: 2 Assembly: (s) 0.11 0.20
FlowSolve: iter: 2 Solve: (s) 0.03 0.06
```



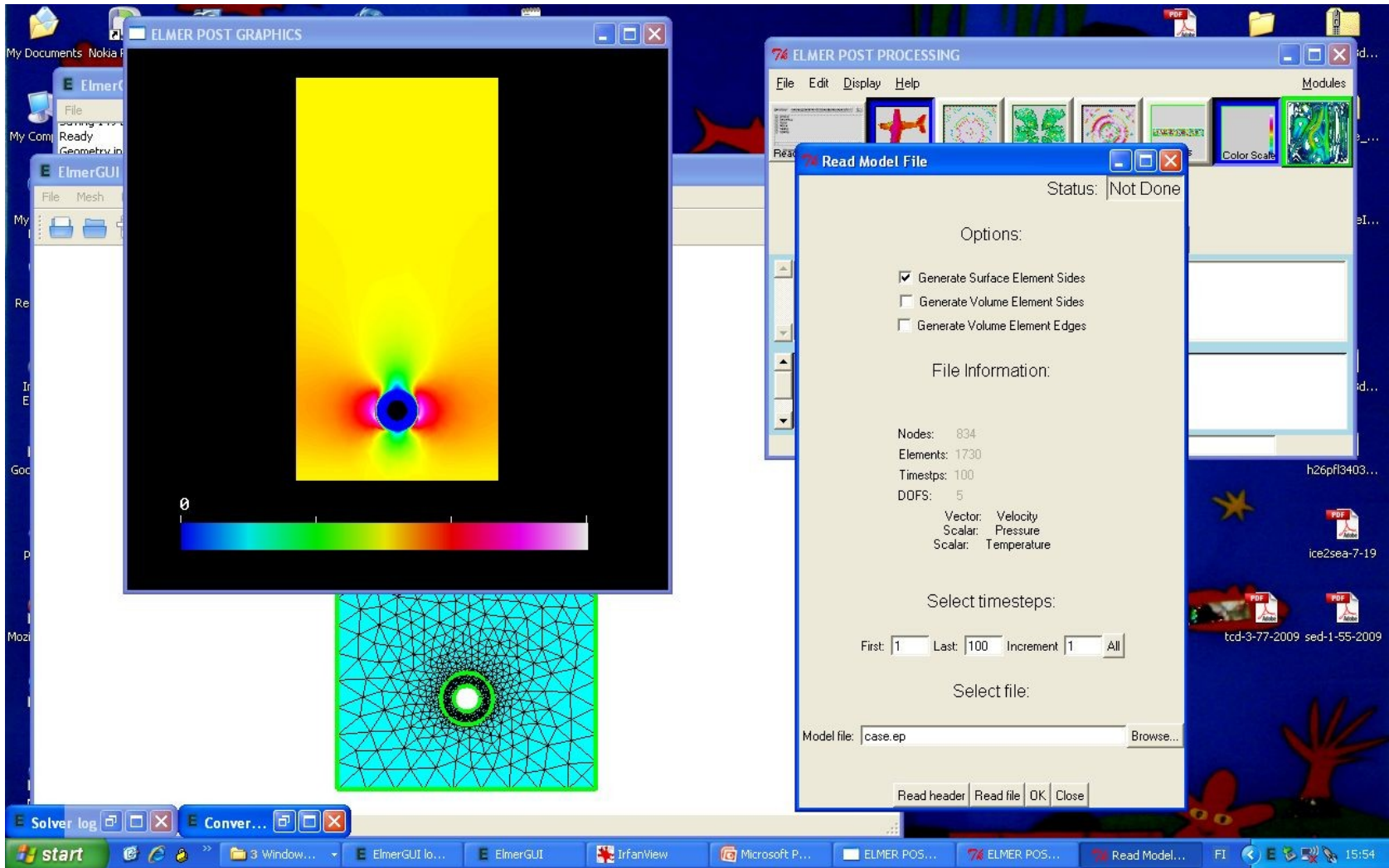


# ElmerGUI Postprocessing

- Upon getting a converged result
  - Either launch ElmerPost by clicking the symbol
    - Or **Run -> Start Postprocessor**
- Then ElmerPost launches 
  - ElmerPost will render the absolute 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** button + **OK** thereafter

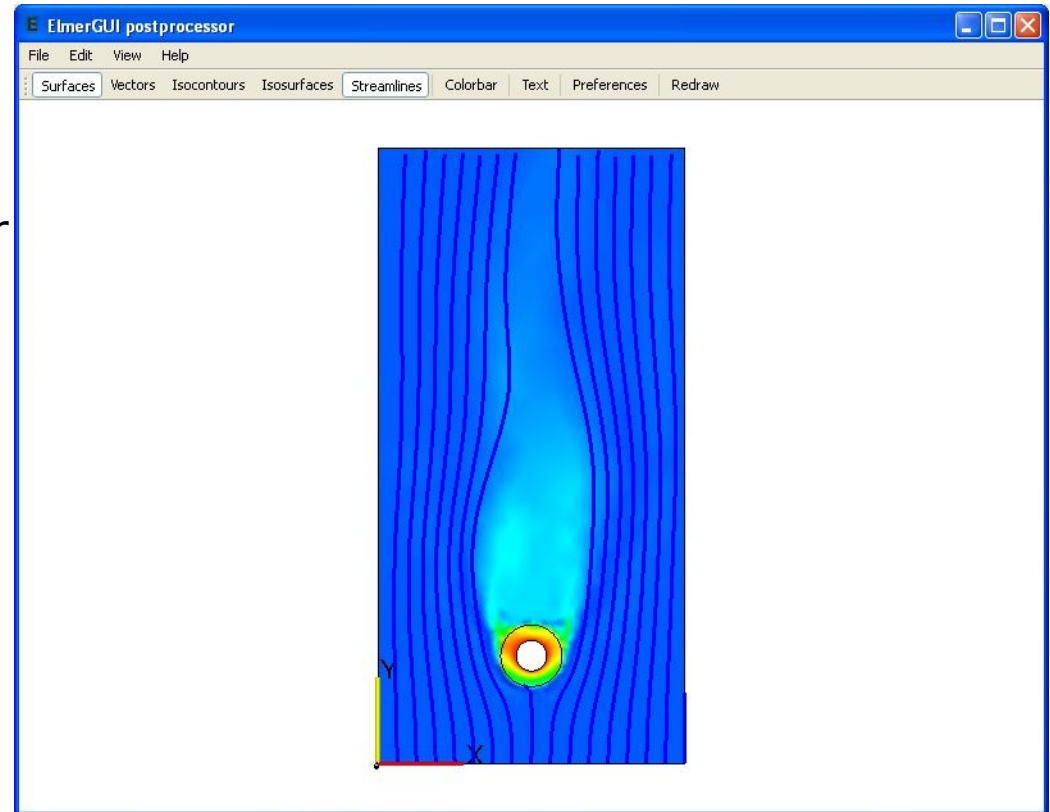


# ElmerGUI Postprocessing ctd.



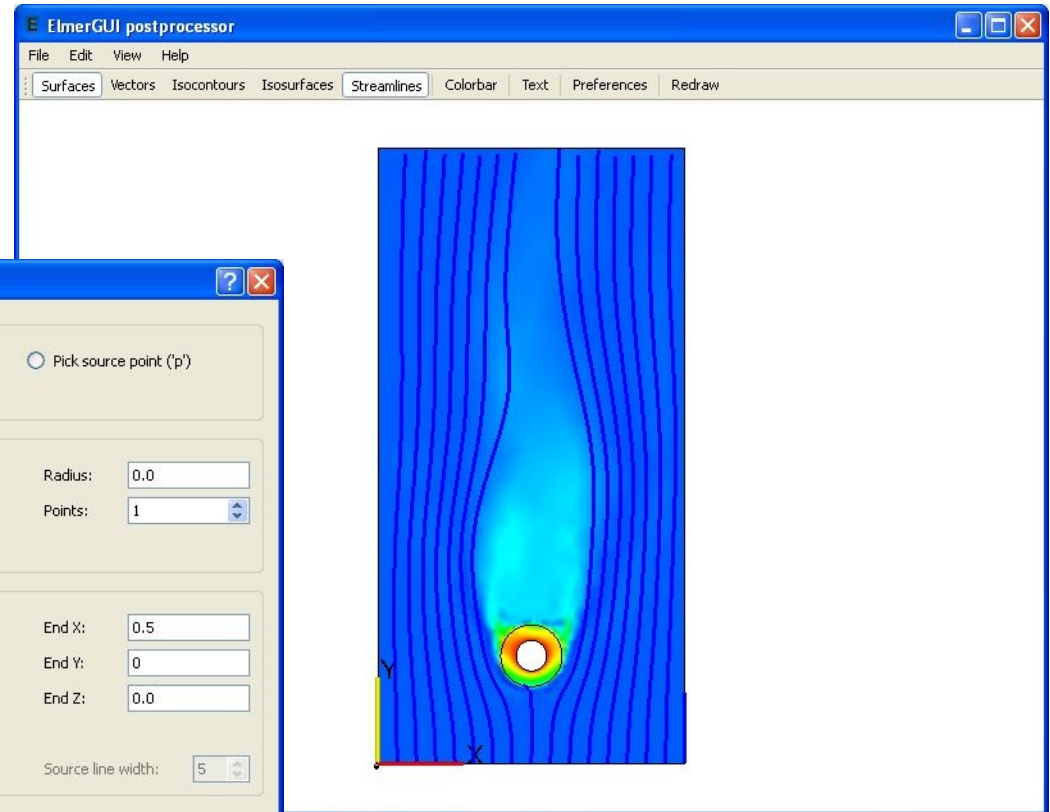
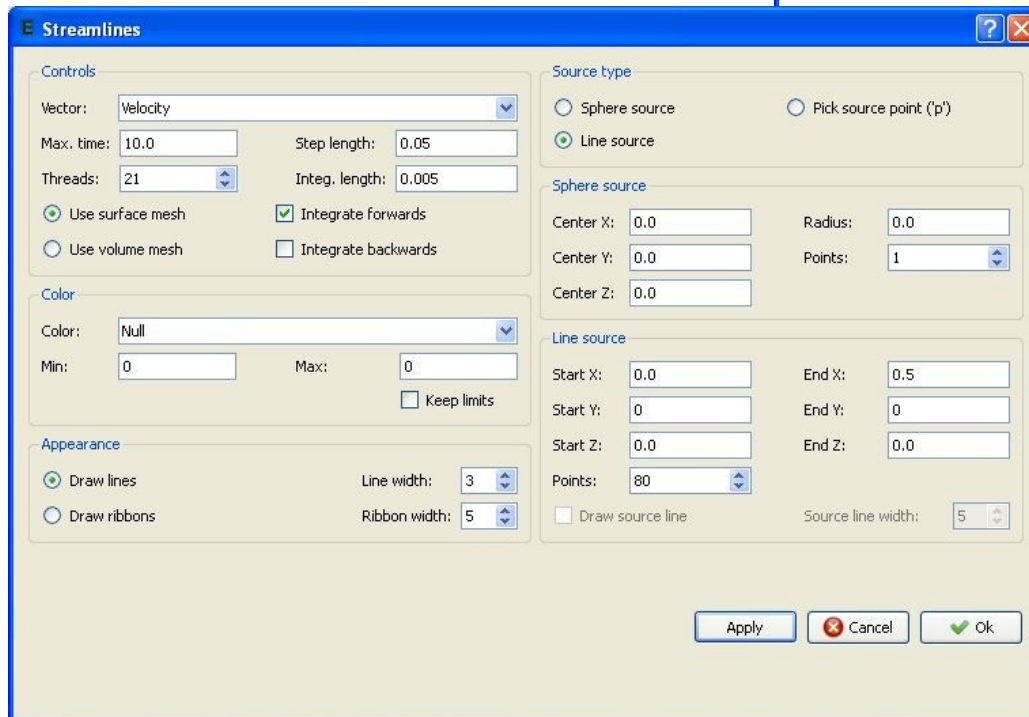
# ElmerGUI Postprocessing VTK

- 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** button + **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