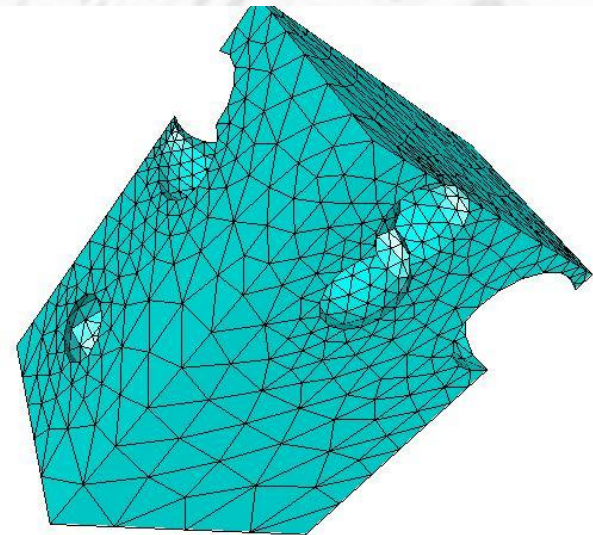
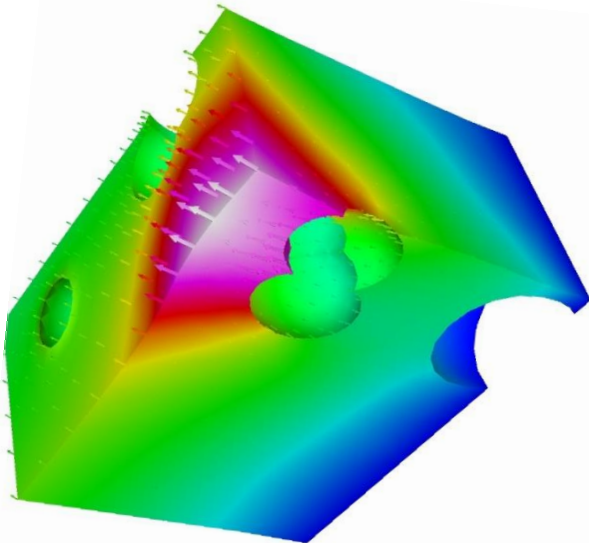




ElmerGUI demo example

# How does your Swiss cheese deform?

A walk-through demonstration

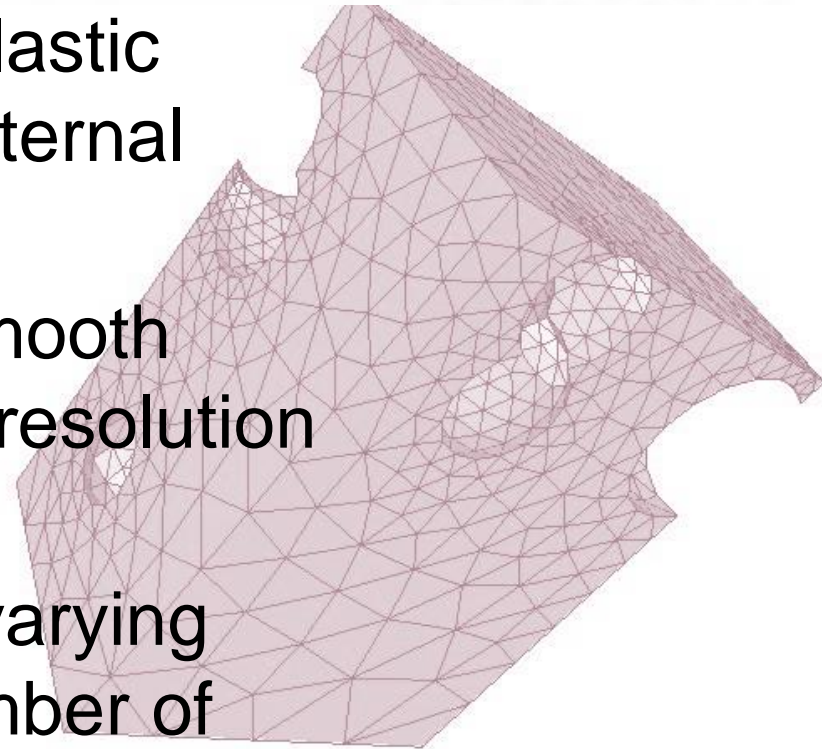


# ElmerGUI Demo

- The material for this demo is on the USB stick
- If you have a working Elmer/ElmerGUI environment at hand, you can do the example at the same time, as we will go in a reasonably slow, yet steady pace
- BUT: we are not able to stop if you got seriously stuck
  - But, there is always time for short questions
- The example can be revisited (including our support) in the last hour of this tutorial

# The Problem/Motivation

- A 20 x 20 x 40 cm block of cheese (assumed to be a linearly elastic material) is put under an external force
- **Linear equations** rather smooth solution → increased mesh resolution only by geometry
- Testing on topologies with varying geometric complexity – number of voids may easily be altered





# Work Flow

## ➤ Pre-processing

- Create a random distribution of spherical voids (=holes) in a brick as a Netgen geometry file using Octave/Matlab script
- Meshing geometry using Netgen
- Checking mesh using ElmerGrid/Post

## ➤ Set-up

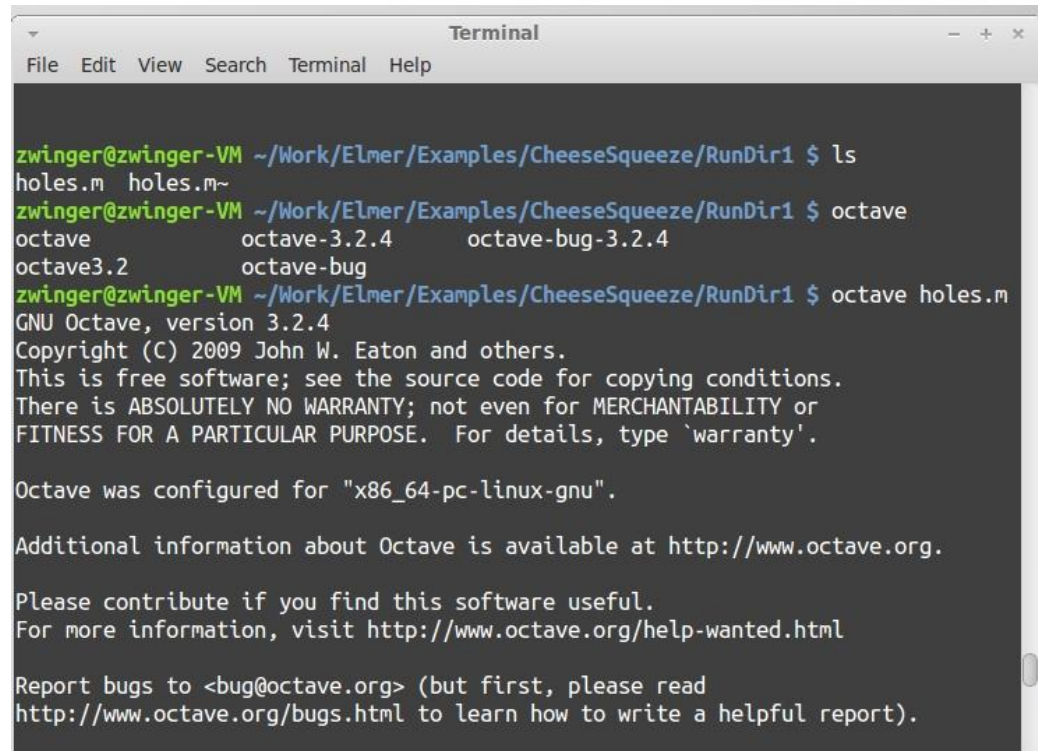
- Importing mesh into ElmerGUI
- Defining the case/project

## ➤ Run

## ➤ Post-processing

# Pre-processing

- ➡ Creating the geometry:
  - ➡ Copy file `holes.m` into a new directory
  - ➡ Apply the changes (change from 100 to 10 holes)
  - ➡ Run `octave holes.m`
  - ➡ That should create a file named `holes10.geo`



```
Terminal
File Edit View Search Terminal Help

zwinger@zwinger-VM ~/Work/Elmer/Examples/CheeseSqueeze/RunDir1 $ ls
holes.m holes.m~
zwinger@zwinger-VM ~/Work/Elmer/Examples/CheeseSqueeze/RunDir1 $ octave
octave octave-3.2.4 octave-bug-3.2.4
octave3.2 octave-bug
zwinger@zwinger-VM ~/Work/Elmer/Examples/CheeseSqueeze/RunDir1 $ octave holes.m
GNU Octave, version 3.2.4
Copyright (C) 2009 John W. Eaton and others.
This is free software; see the source code for copying conditions.
There is ABSOLUTELY NO WARRANTY; not even for MERCHANTABILITY or
FITNESS FOR A PARTICULAR PURPOSE. For details, type `warranty'.

Octave was configured for "x86_64-pc-linux-gnu".

Additional information about Octave is available at http://www.octave.org.

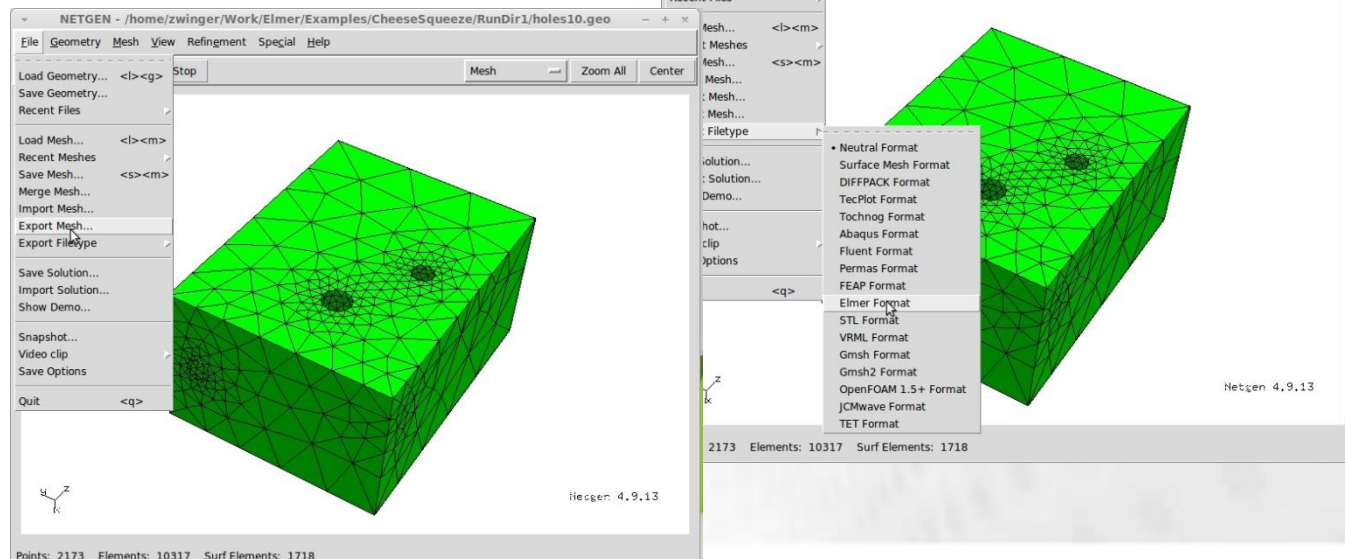
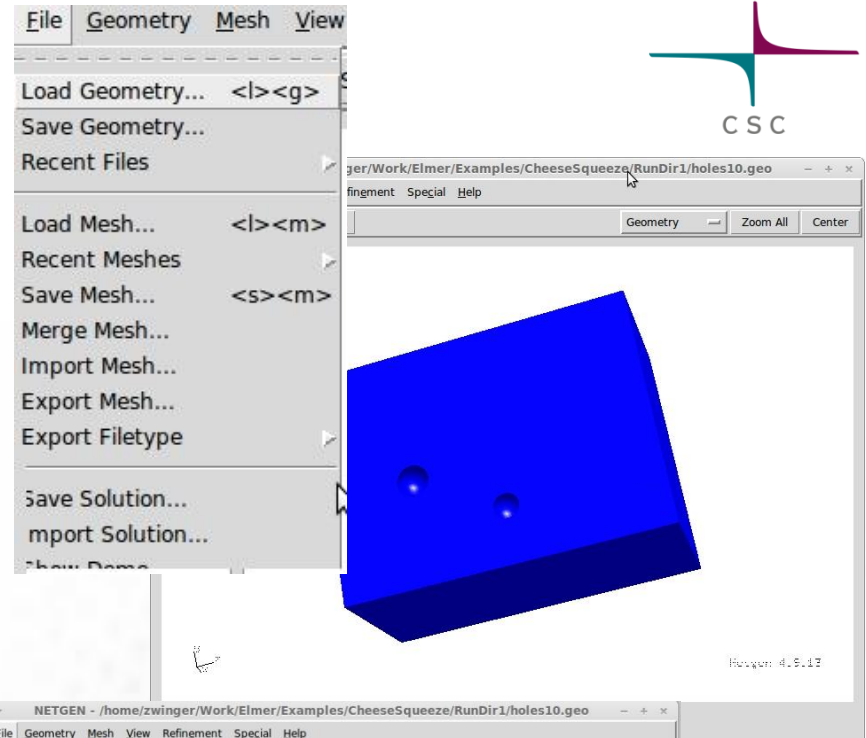
Please contribute if you find this software useful.
For more information, visit http://www.octave.org/help-wanted.html

Report bugs to <bug@octave.org> (but first, please read
http://www.octave.org/bugs.html to learn how to write a helpful report).
```

# Pre-processing

## Creating the geometry:

- Start **netgen**
- Load **holes10.geo**
- Click on **Generate Mesh**
- Choose *Elmer* under **Export Filetype**
- Export Mesh**



# Pre-processing

- The mesh is stored in the files

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

- Copy them into a sub-directory `holes10`

- Create an ElmerPost output-file:

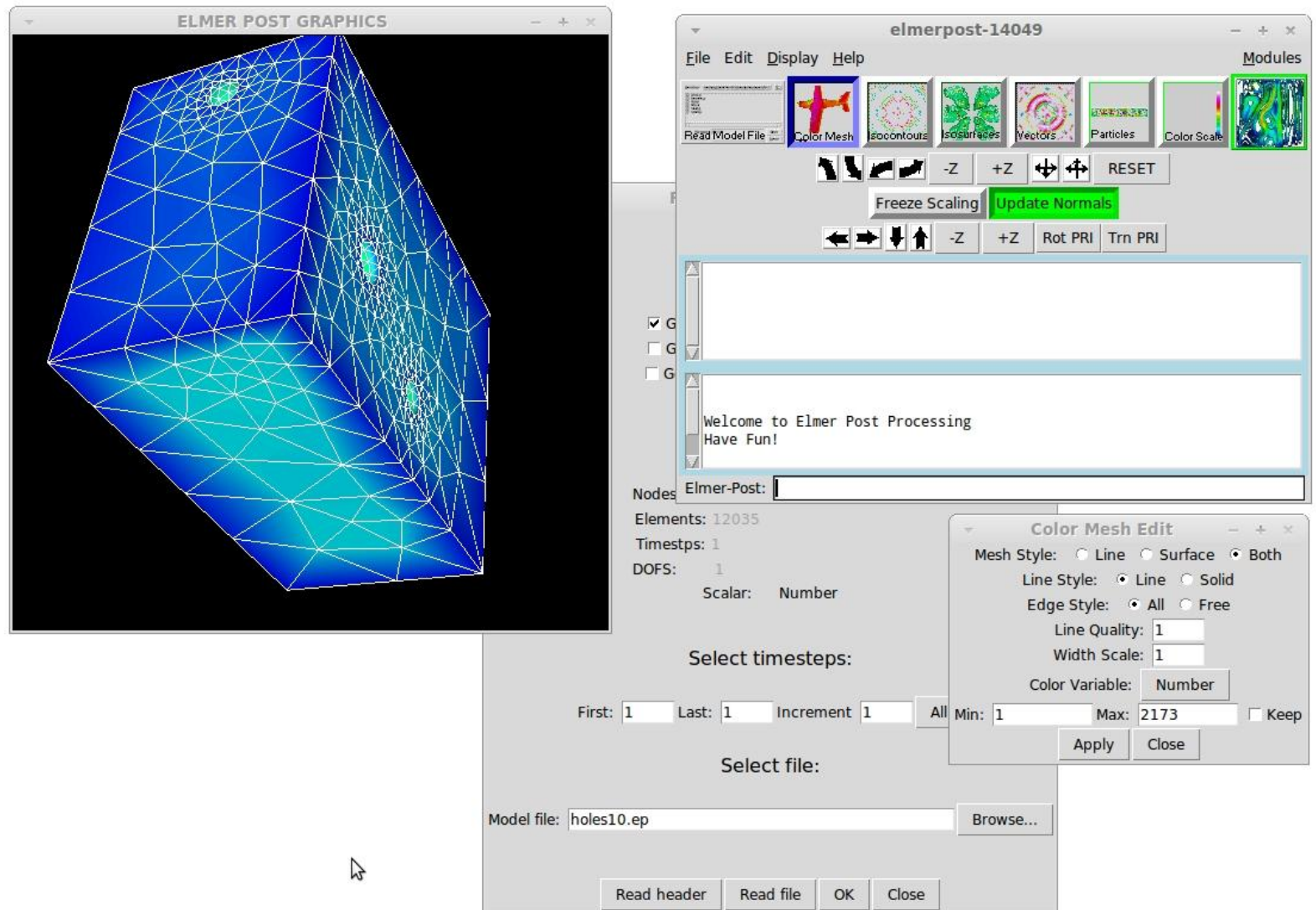
`ElmerGrid 2 3 holes10`

- Launch **ElmerPost**

- and load `holes10.ep`
- Open Edit → Grouping ... and check the different boundaries
- Our relevant boundaries are 1-6 (the large sides)
- Check the dimensions 4 x 4 x 2
  - Too large a piece of cheese, if meters



# Pre-processing

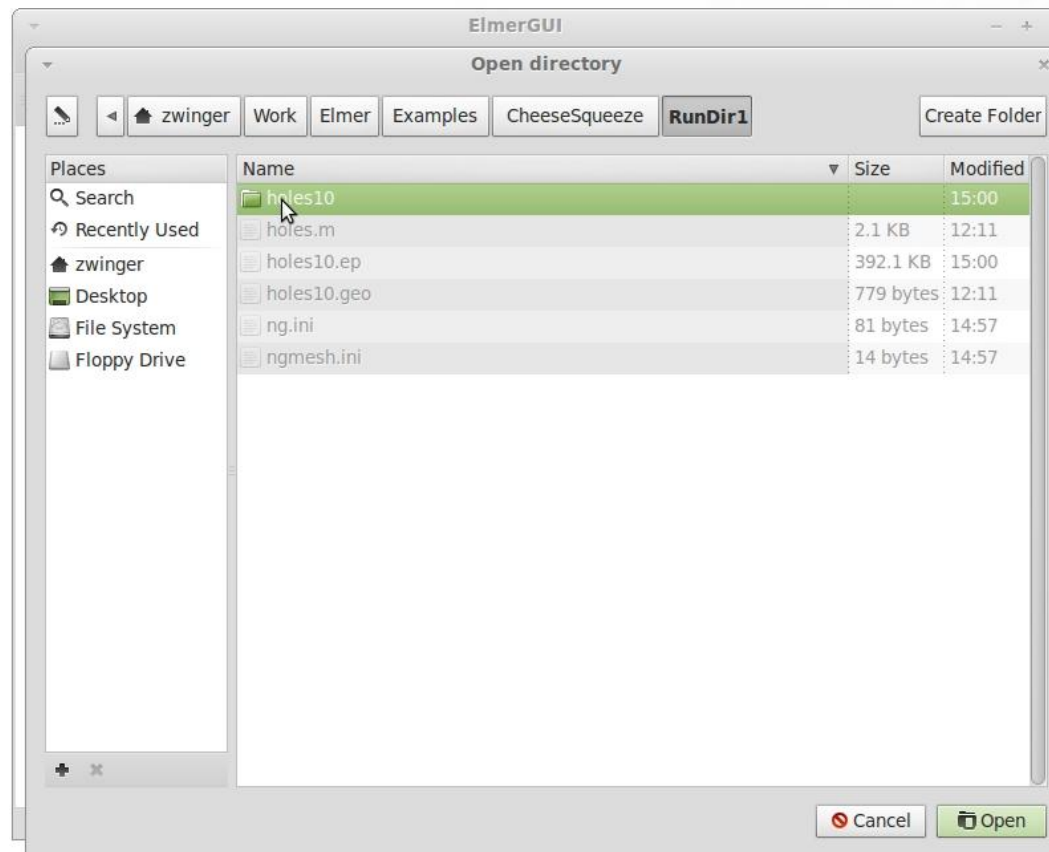


# Pre-processing

- Elmer does not assume any unit system
- Built-in material library parameters are in SI units
- User has to guarantee consistency between geometry and physical constants
- One possibility: Scale the mesh
  - unit length was 10 cm = 0.1 m
  - ```
ElmerGrid 2 2 holes10 -scale 0.1 0.1 0.1 -out  
holes10_scale
```
  - Now the mesh is in SI-units (meters) and the built-in material library (in SI units) could be used
- Here, we are using **internal scaling** provided by Elmer (see later in this tutorial)

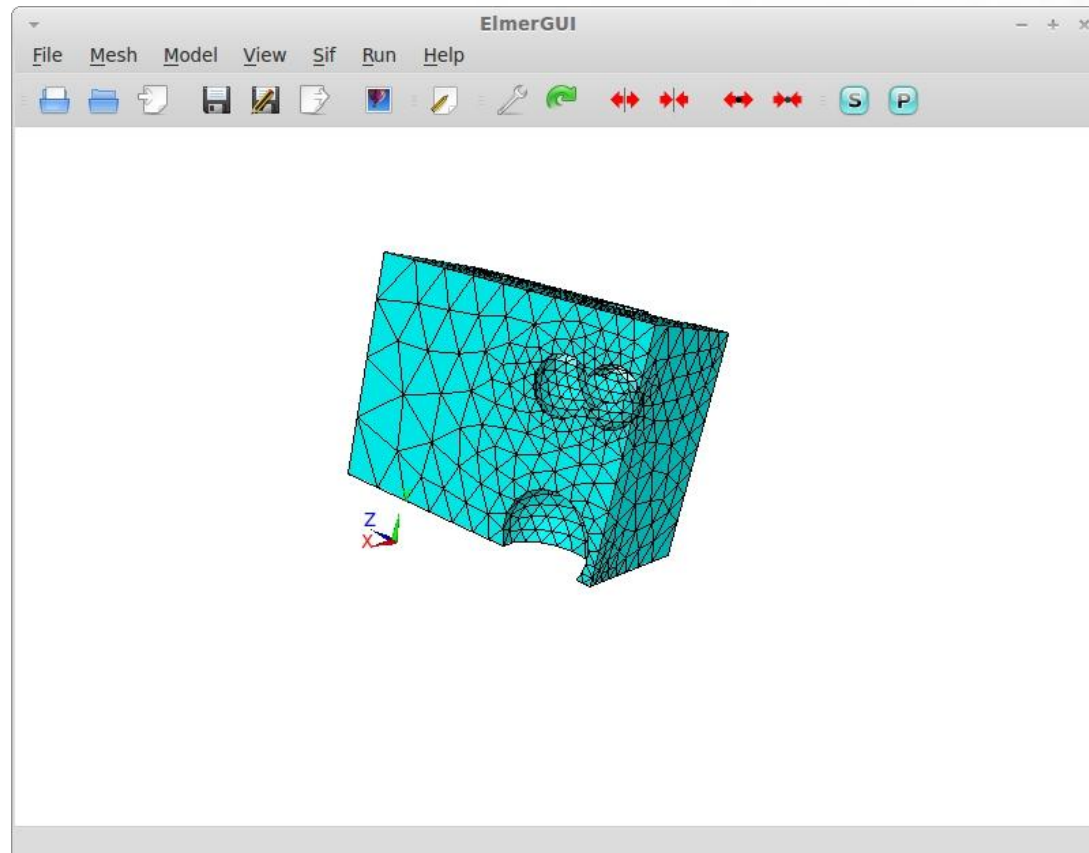
# Set-up

- ➡ Launch ElmerGUI
- ➡ **File** → **Load Mesh: holes10**



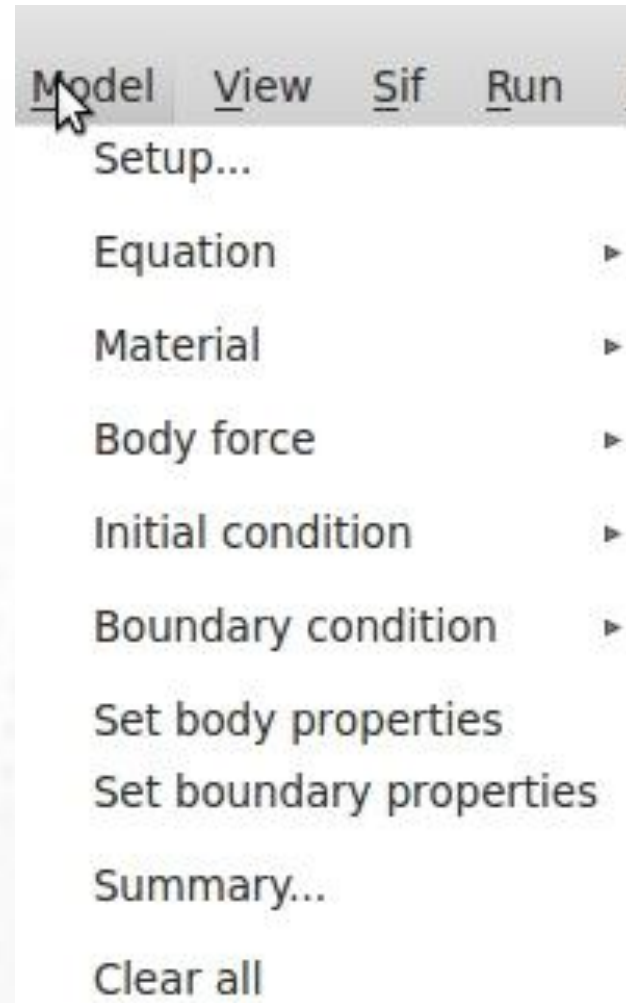
# Set-up

- ➡ Launch ElmerGUI
- ➡ **File** → **Load Mesh: holes10**



# Set-up

- All following steps are sequentially ordered in the menu **Model**
- They reflect the different section in the solver input file that is created for the solver step





# Set-up

## ➤ Model → Setup

– Basic setup of the simulation

➤ Header for mesh

➤ Simulation

➤ Constants

– Possibility to scale input mesh

Setup

Header

☒ Check keywords warn

MeshDB

Include path

Results directory

Free text

Simulation

Max. output level

Steady state max. iter

Coordinate system

Timestepping method

Coordinate mapping

BDF order

Simulation type

Timestep intervals

Output intervals

Timestep sizes

Solver input file

Post file

Free text

Constants

Gravity

Boltzmann

Stefan Boltzmann

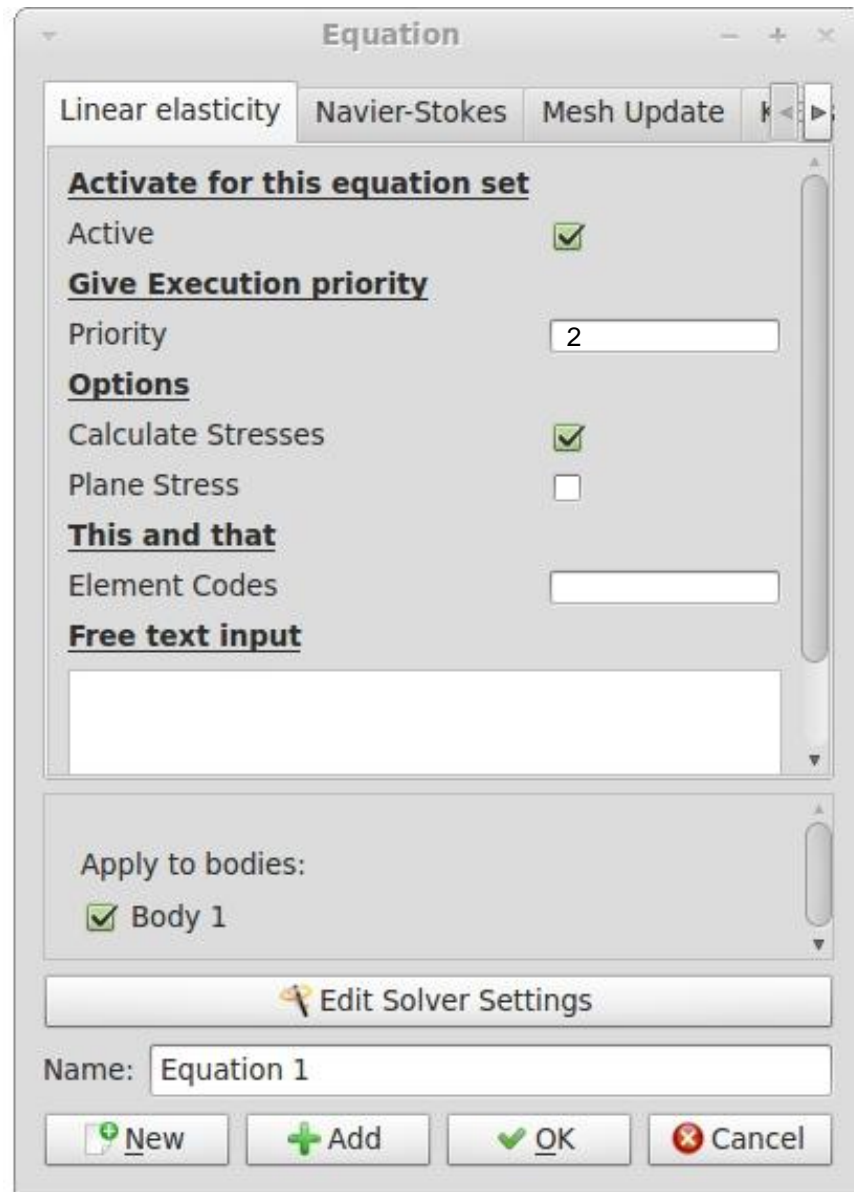
Unit charge

Vacuum permittivity

Free text

# Set-up

- ➔ **Model → Equation**
  - definition of physical models (=Solvers) for the simulation
- ➔ **Toggle Linear elasticity**
  - **Priority to 2**
  - **Edit Solver Settings**



The image shows a software dialog box titled "Equation". It has three tabs: "Linear elasticity" (selected), "Navier-Stokes", and "Mesh Update".

**Activate for this equation set**

Active ☒

**Give Execution priority**

Priority

**Options**

Calculate Stresses ☒

Plane Stress ☐


**This and that**

Element Codes

**Free text input**

**Apply to bodies:**

☒ Body 1

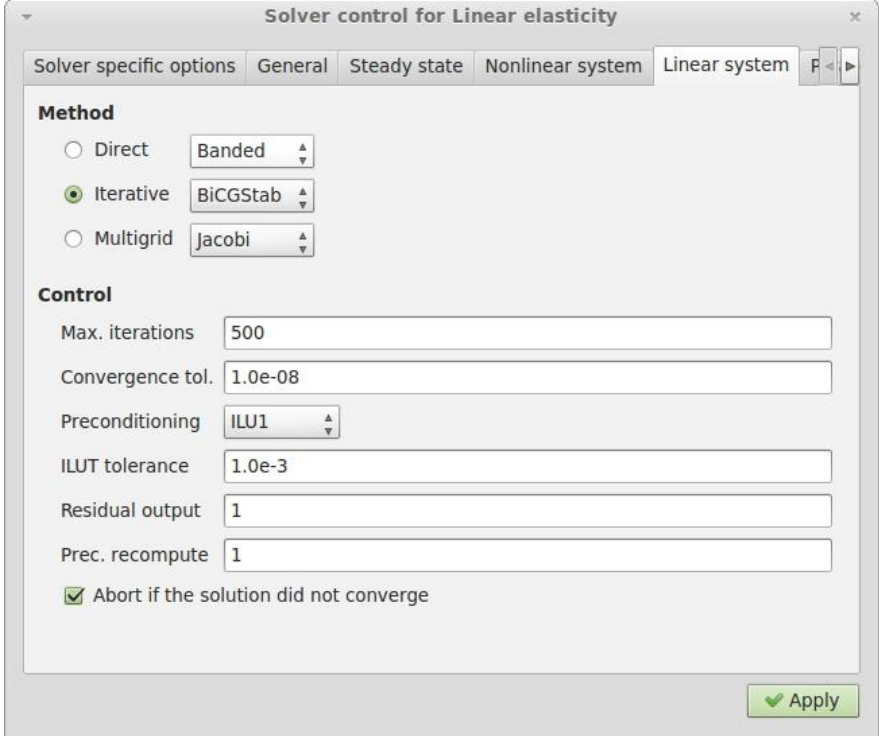
 Edit Solver Settings

Name:

Buttons:

# Set-up

- ➔ **Linear system**
- ➔ **Nonlinear system:**
  - Reduce **Max. Iterations** to 1 (linear problem!)



Solver control for Linear elasticity

Solver specific options General Steady state Nonlinear system Linear system

**Method**

☐ Direct Banded

☒ Iterative BiCGStab

☐ Multigrid Jacobi

**Control**

Max. iterations 500

Convergence tol. 1.0e-08

Preconditioning ILU1

ILUT tolerance 1.0e-3

Residual output 1

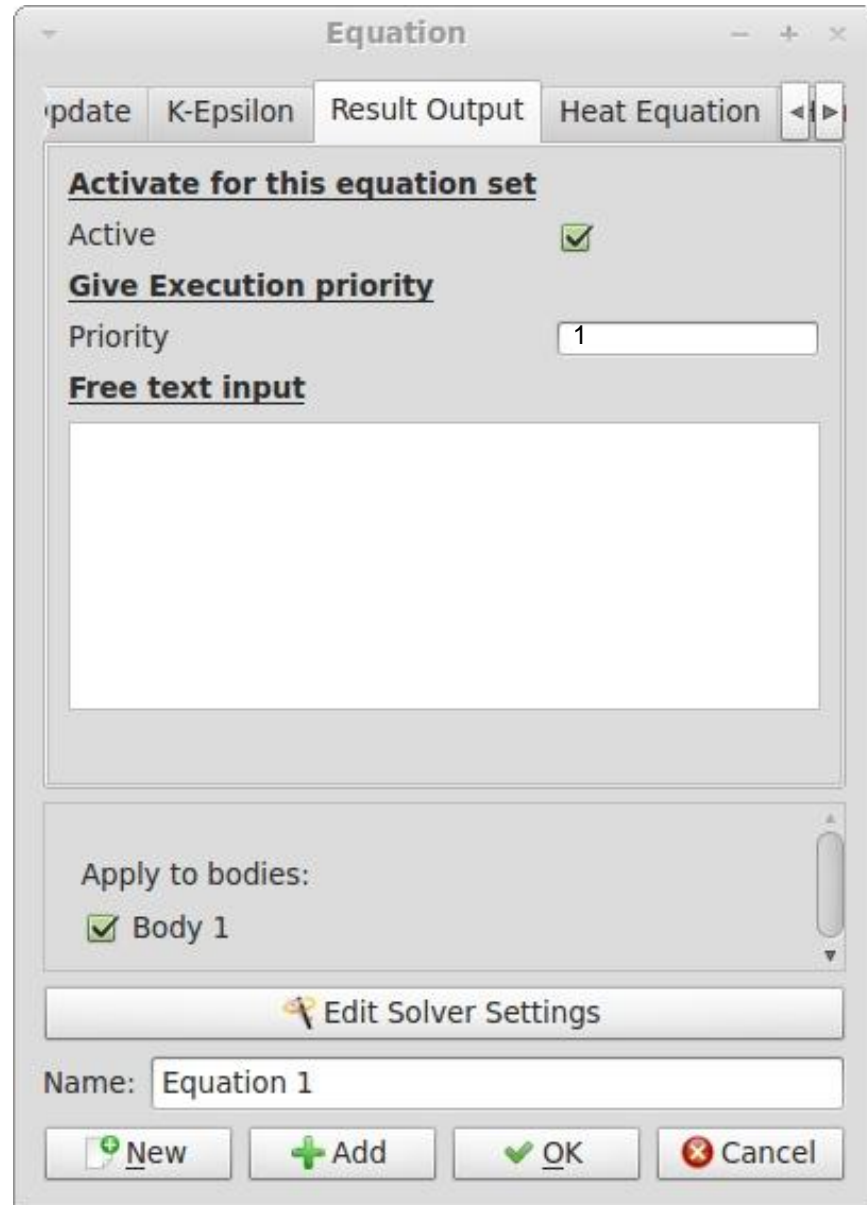
Prec. recompute 1

☒ Abort if the solution did not converge

Apply

# Set-up

- **Toggle Result Output**
  - **Priority to 1**
  - **Edit Solver Settings**



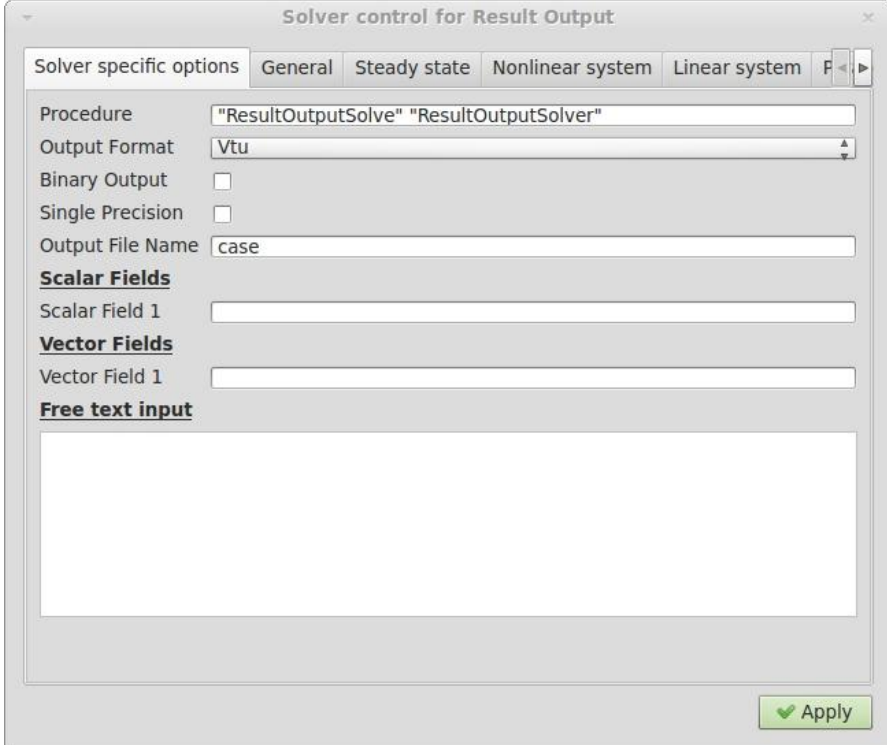
The image shows a software dialog box titled "Equation". It has four tabs: "update", "K-Epsilon", "Result Output", and "Heat Equation". The "Result Output" tab is currently selected. Inside this tab, there are three sections:
 

- Activate for this equation set**: The "Active" checkbox is checked.
- Give Execution priority**: The "Priority" is set to "1" in a text field.
- Free text input**: A large empty text area.

 Below these sections, there is a section labeled "Apply to bodies:" with a list containing "Body 1" which has a checked checkbox. At the bottom of the dialog, there is a button labeled "Edit Solver Settings" with a wrench icon. Below that is a "Name:" field containing the text "Equation 1". At the very bottom are four buttons: "New" (with a plus icon), "Add" (with a plus icon), "OK" (with a checkmark icon), and "Cancel" (with a red X icon).

# Set-up

- ➔ **Linear system**
- ➔ **Solver specific options**



Solver control for Result Output

Solver specific options General Steady state Nonlinear system Linear system

Procedure "ResultOutputSolve" "ResultOutputSolver"

Output Format Vtu

Binary Output ☐

Single Precision ☐

Output File Name case

**Scalar Fields**

Scalar Field 1

**Vector Fields**

Vector Field 1

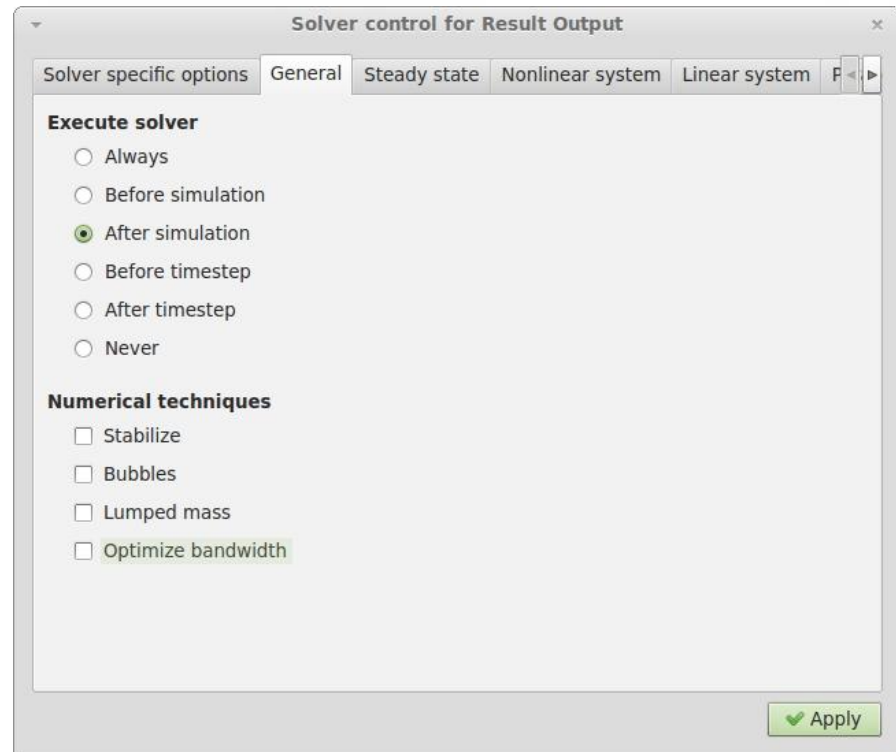
**Free text input**

Apply



# Set-up

- ➔ **Linear system**
- ➔ **Solver specific options**
- ➔ **General:**
  - Toggle **after simulation**
  - Why?: Results to be saved when converged



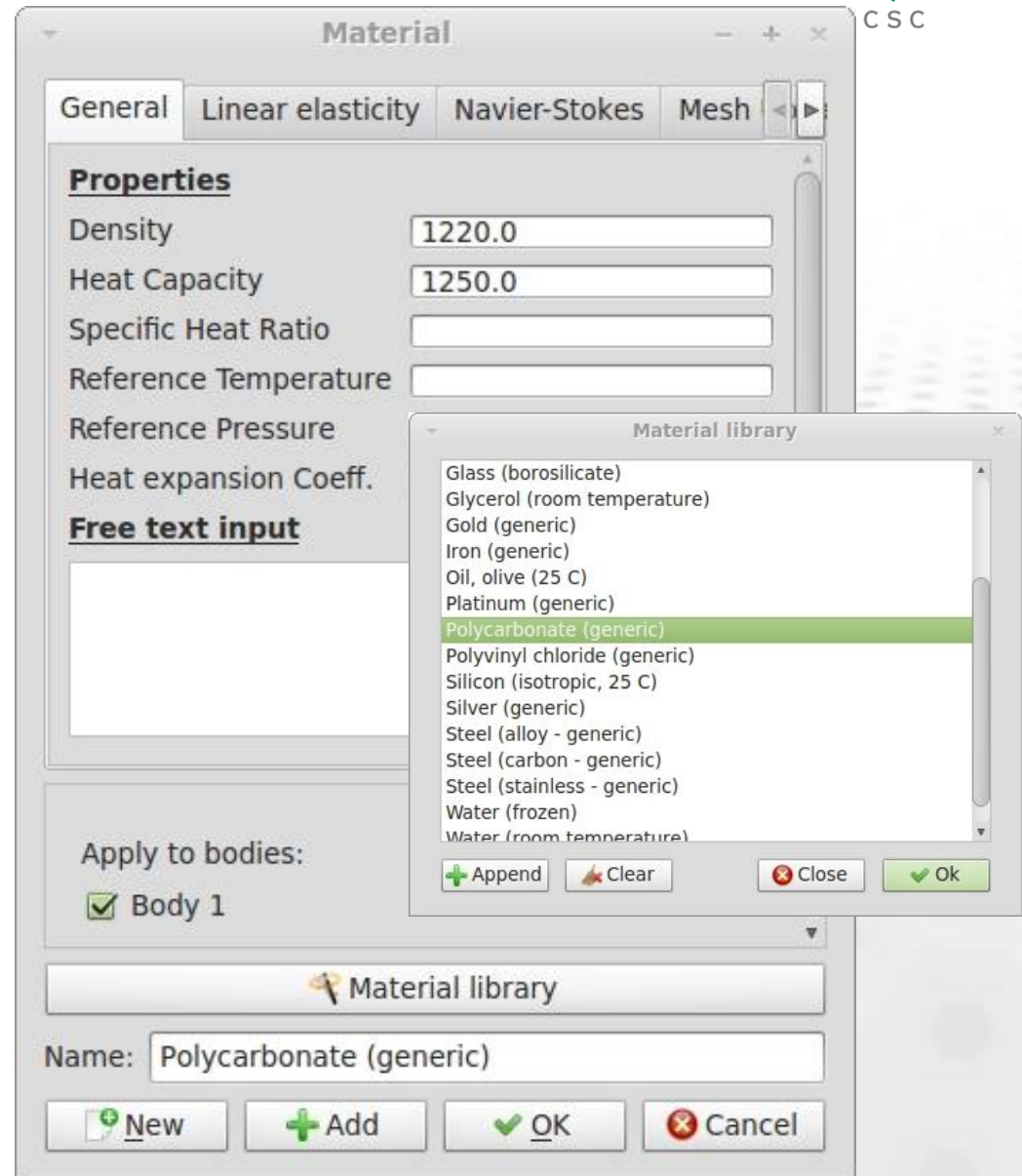
# Set-up

## ➤ Model → Material

- definition of physical properties for the simulation

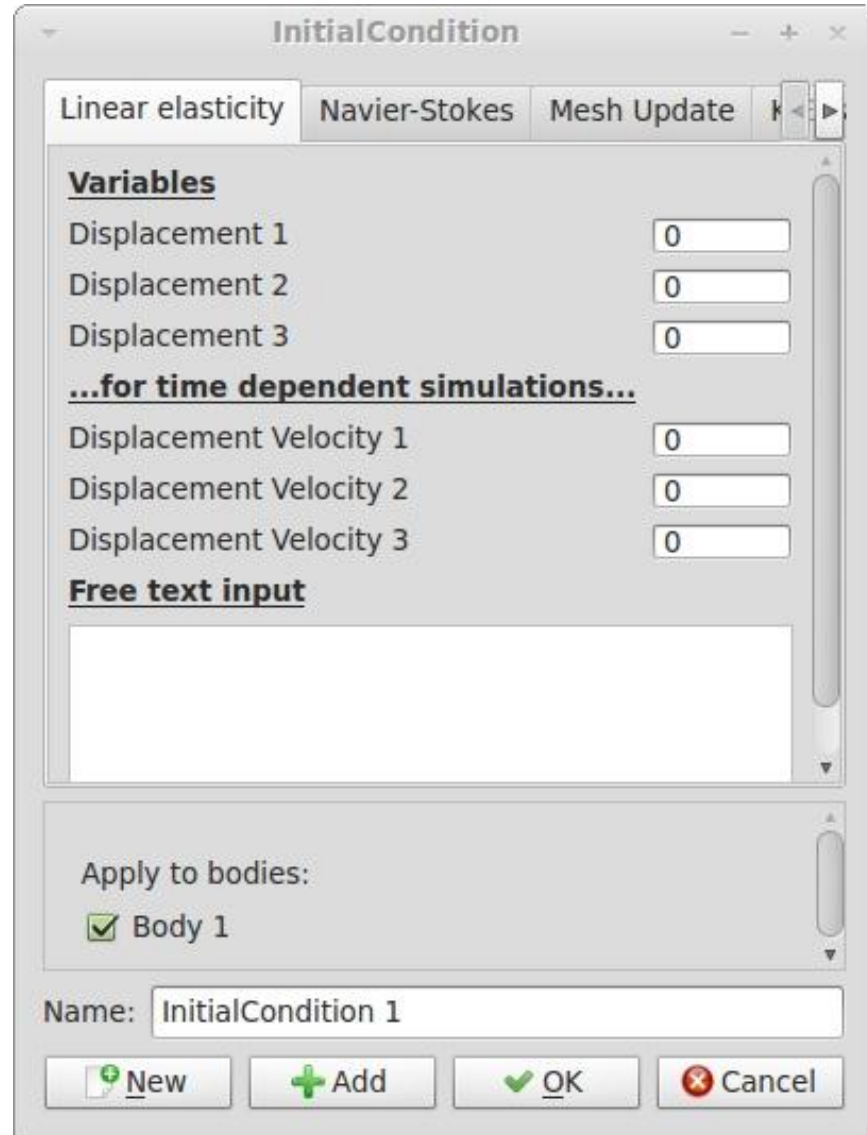
## ➤ Material Library

- Polycarbonate (generic)



# Set-up

- ➔ **Model → Initial Condition**
  - initialization of variable values for the simulation
- ➔ **Linear elasticity**
  - All variables to zero



The image shows a software dialog box titled "InitialCondition". It has three tabs: "Linear elasticity", "Navier-Stokes", and "Mesh Update". The "Linear elasticity" tab is selected. Inside the dialog, there are sections for "Variables" and "Free text input". The "Variables" section lists "Displacement 1", "Displacement 2", and "Displacement 3", each with a text input field containing the value "0". Below this is a section for "Displacement Velocity 1", "Displacement Velocity 2", and "Displacement Velocity 3", each also with a text input field containing "0". The "Free text input" section is a large empty text area. At the bottom, there is a section "Apply to bodies:" with a checked checkbox next to "Body 1". Below that is a "Name:" label followed by a text input field containing "InitialCondition 1". At the very bottom are four buttons: "New" (with a plus icon), "Add" (with a plus icon), "OK" (with a checkmark icon), and "Cancel" (with a red X icon).

InitialCondition

Linear elasticity   Navier-Stokes   Mesh Update

**Variables**

Displacement 1   0

Displacement 2   0

Displacement 3   0

**...for time dependent simulations...**

Displacement Velocity 1   0

Displacement Velocity 2   0

Displacement Velocity 3   0

**Free text input**

Apply to bodies:

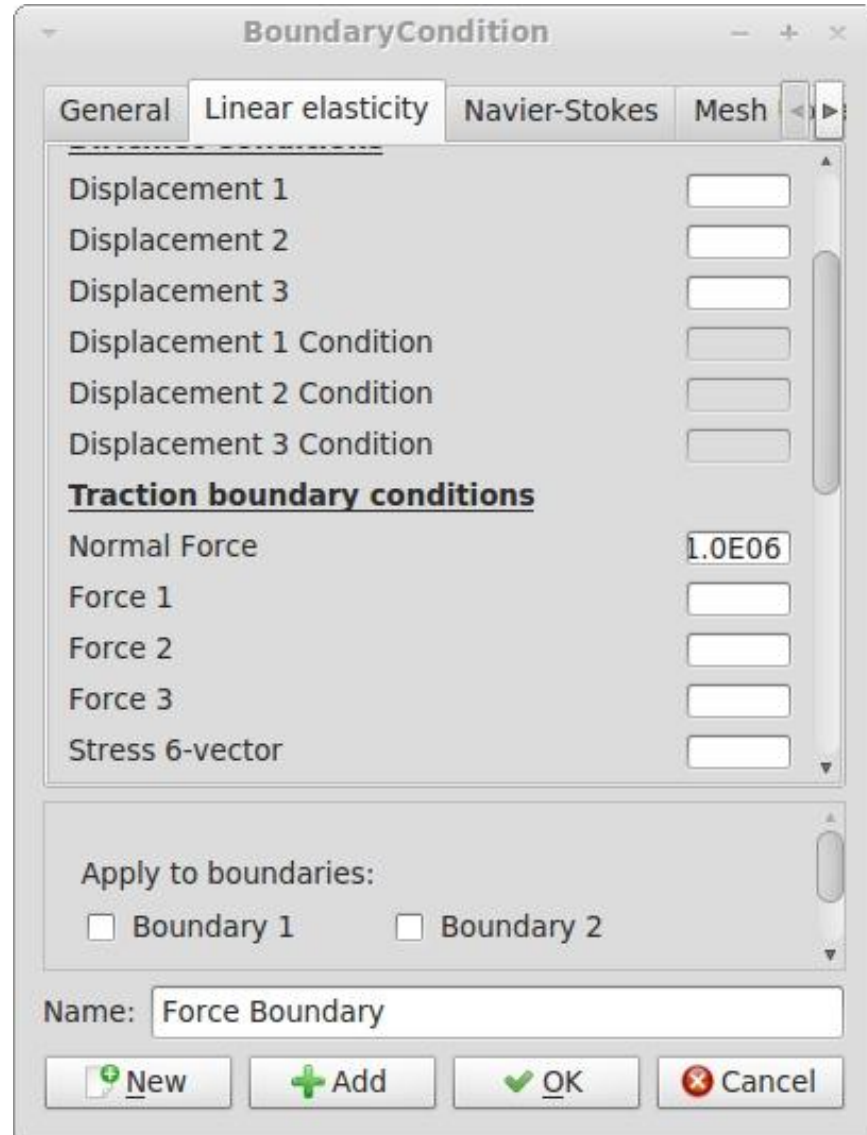
☒ Body 1

Name: InitialCondition 1

New   Add   OK   Cancel

# Set-up

- **Model** → **Boundary Condition**
  - definition of variable values at boundaries for the simulation
  - Usually multiple
  - Different names
- **Linear elasticity**
  - Force boundary
  - **Add + New**



BoundaryCondition

General Linear elasticity Navier-Stokes Mesh

Displacement 1  
Displacement 2  
Displacement 3  
Displacement 1 Condition  
Displacement 2 Condition  
Displacement 3 Condition

**Traction boundary conditions**

Normal Force 1.0E06  
Force 1  
Force 2  
Force 3  
Stress 6-vector

Apply to boundaries:  
☐ Boundary 1 ☐ Boundary 2

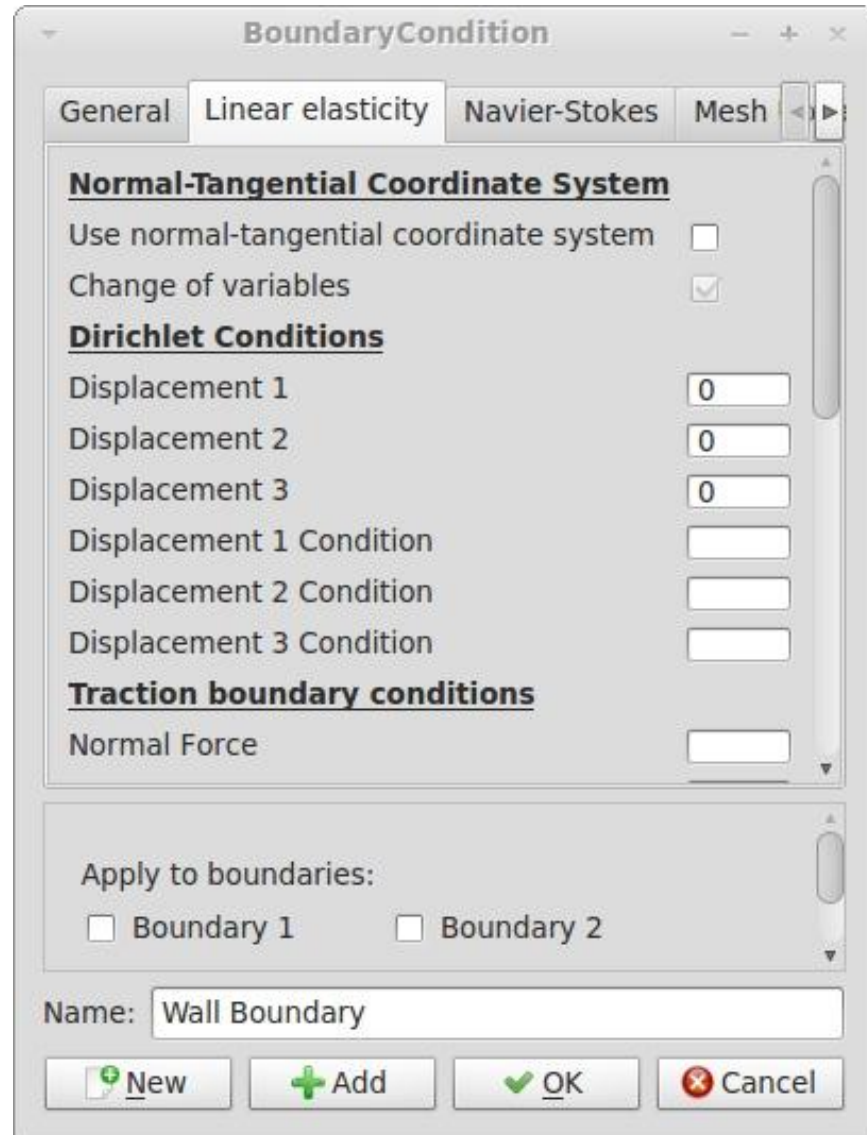
Name: Force Boundary

New Add OK Cancel

# Set-up

## ➡ Linear elasticity

- Wall boundary
- **Add + New**



The image shows a software dialog box titled "BoundaryCondition". It has four tabs: "General", "Linear elasticity" (which is selected), "Navier-Stokes", and "Mesh".

Under the "Linear elasticity" tab, there are three main sections:

- Normal-Tangential Coordinate System**:
  - "Use normal-tangential coordinate system" is unchecked.
  - "Change of variables" is checked.
- Dirichlet Conditions**:
  - "Displacement 1" is set to 0.
  - "Displacement 2" is set to 0.
  - "Displacement 3" is set to 0.
  - "Displacement 1 Condition" is an empty text box.
  - "Displacement 2 Condition" is an empty text box.
  - "Displacement 3 Condition" is an empty text box.
- Traction boundary conditions**:
  - "Normal Force" is an empty text box.

At the bottom of the dialog, there is a section "Apply to boundaries:" with two checkboxes: "Boundary 1" (unchecked) and "Boundary 2" (unchecked).

Below this is a "Name:" label followed by a text box containing "Wall Boundary".

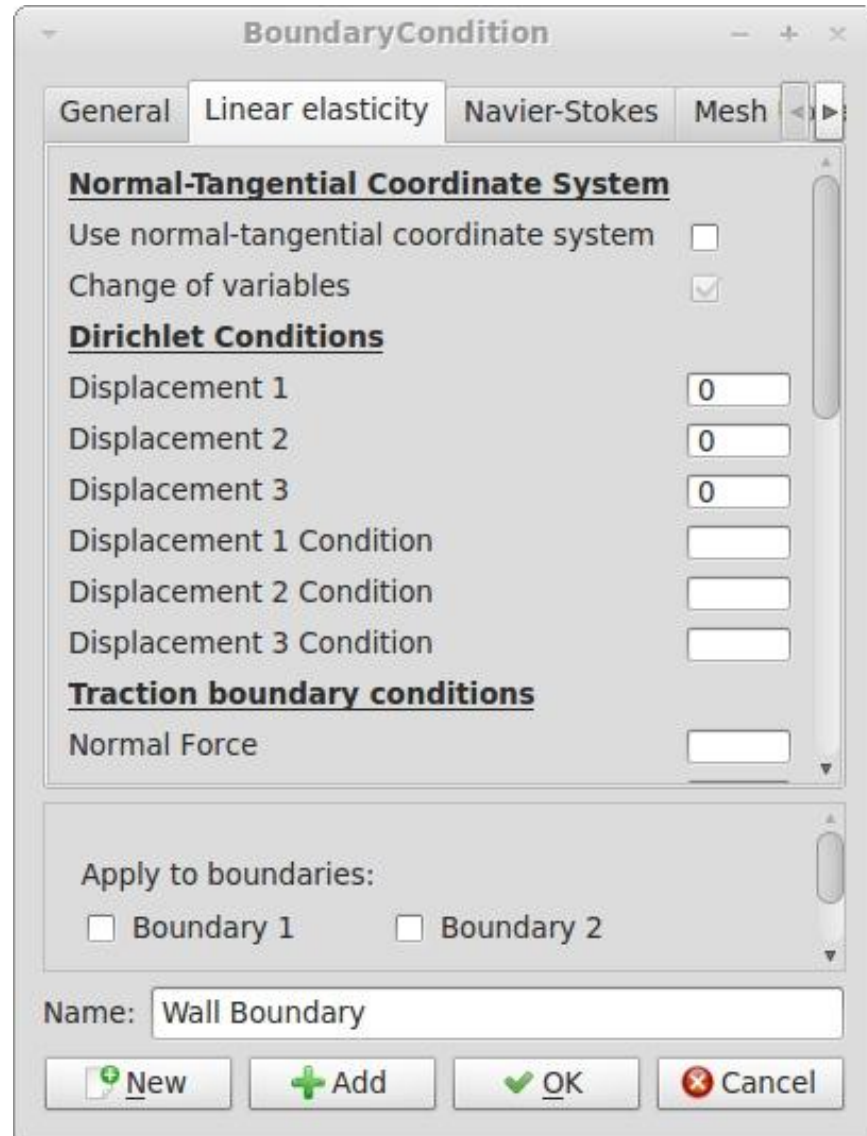
At the very bottom are four buttons: "New" (with a plus icon), "Add" (with a green plus icon), "OK" (with a green checkmark icon), and "Cancel" (with a red X icon).



# Set-up

## ➡ Linear elasticity

- Wall Boundary
- **Add + OK**



The image shows a software dialog box titled "BoundaryCondition". It has four tabs: "General", "Linear elasticity" (which is selected), "Navier-Stokes", and "Mesh".

Under the "Linear elasticity" tab, there are three main sections:

- Normal-Tangential Coordinate System**:
  - "Use normal-tangential coordinate system" is unchecked.
  - "Change of variables" is checked.
- Dirichlet Conditions**:
  - "Displacement 1" is set to 0.
  - "Displacement 2" is set to 0.
  - "Displacement 3" is set to 0.
  - "Displacement 1 Condition" is an empty text box.
  - "Displacement 2 Condition" is an empty text box.
  - "Displacement 3 Condition" is an empty text box.
- Traction boundary conditions**:
  - "Normal Force" is an empty text box.

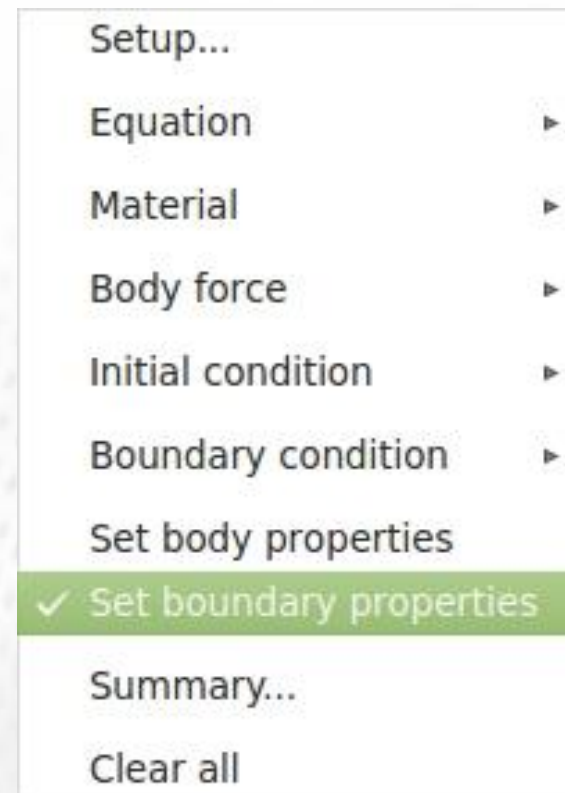
At the bottom of the dialog, there is a section "Apply to boundaries:" with two checkboxes: "Boundary 1" (unchecked) and "Boundary 2" (unchecked).

Below this, there is a "Name:" label followed by a text box containing "Wall Boundary".

At the very bottom, there are four buttons: "New" (with a plus icon), "Add" (with a green plus icon), "OK" (with a green checkmark icon), and "Cancel" (with a red X icon).

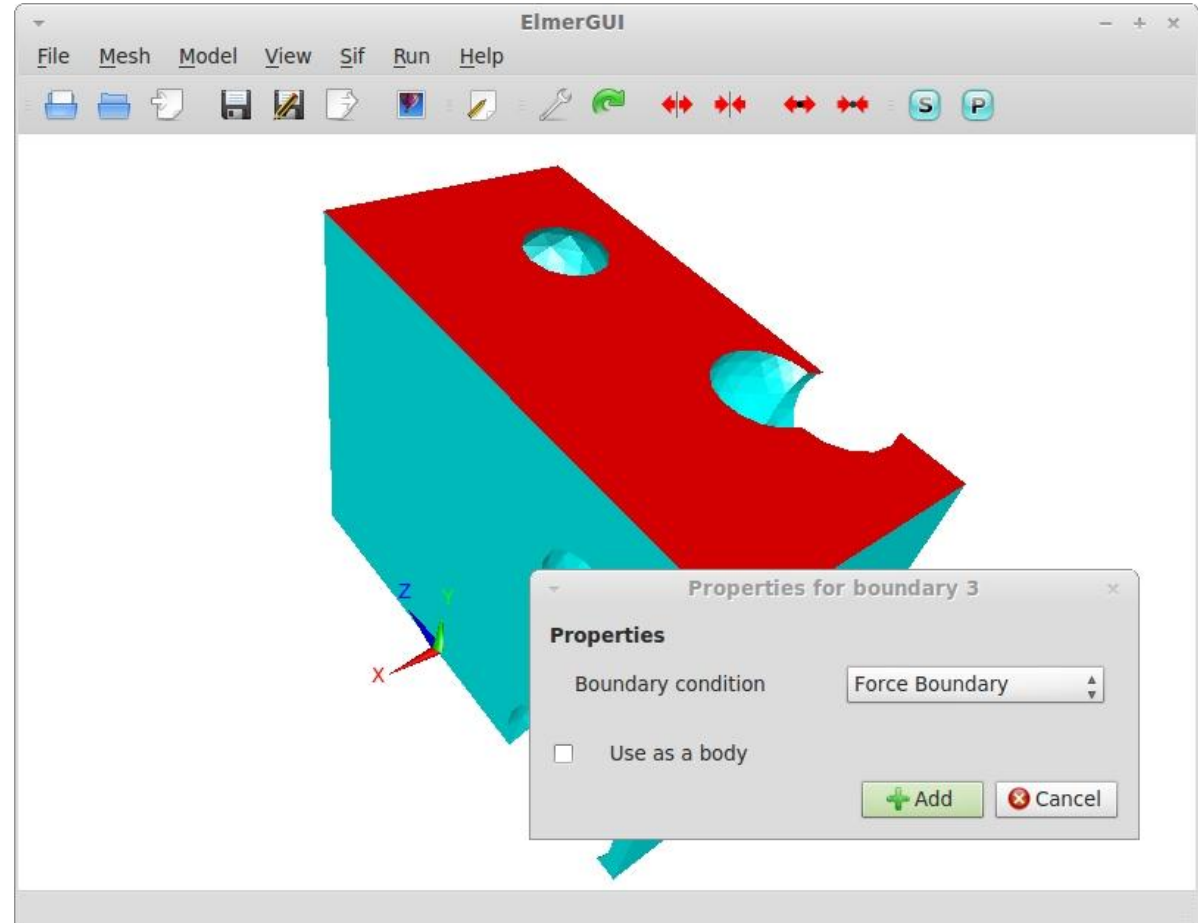
# Set-up

- ➡ In general it is difficult to know the boundary number in the mesh → assign manually
- ➡ **Model → Set boundary properties**
- ➡ Then double-click on specific boundary (gets highlighted)



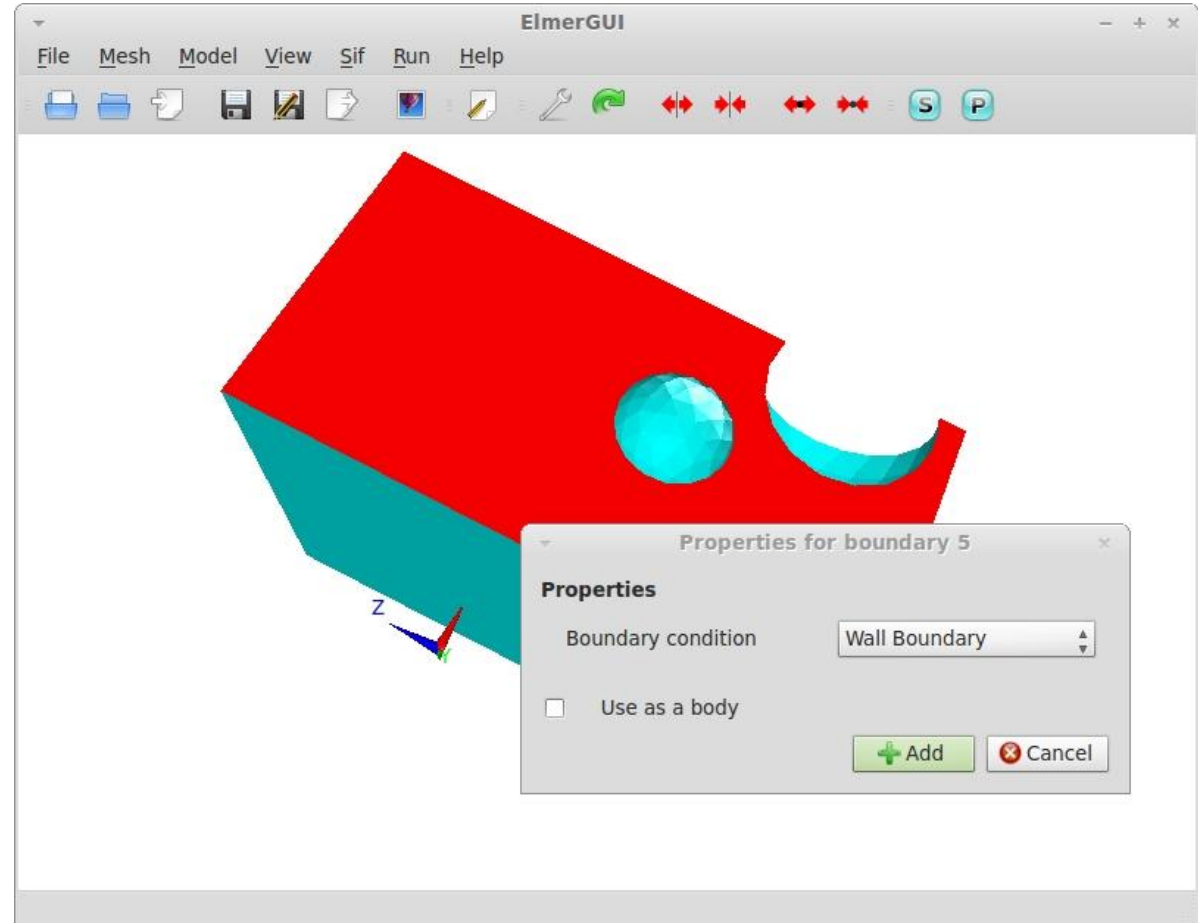
# Set-up

- ➡ Assigning force boundary
- ➡ Press **+Add**
- ➡ Rotate to lower boundary



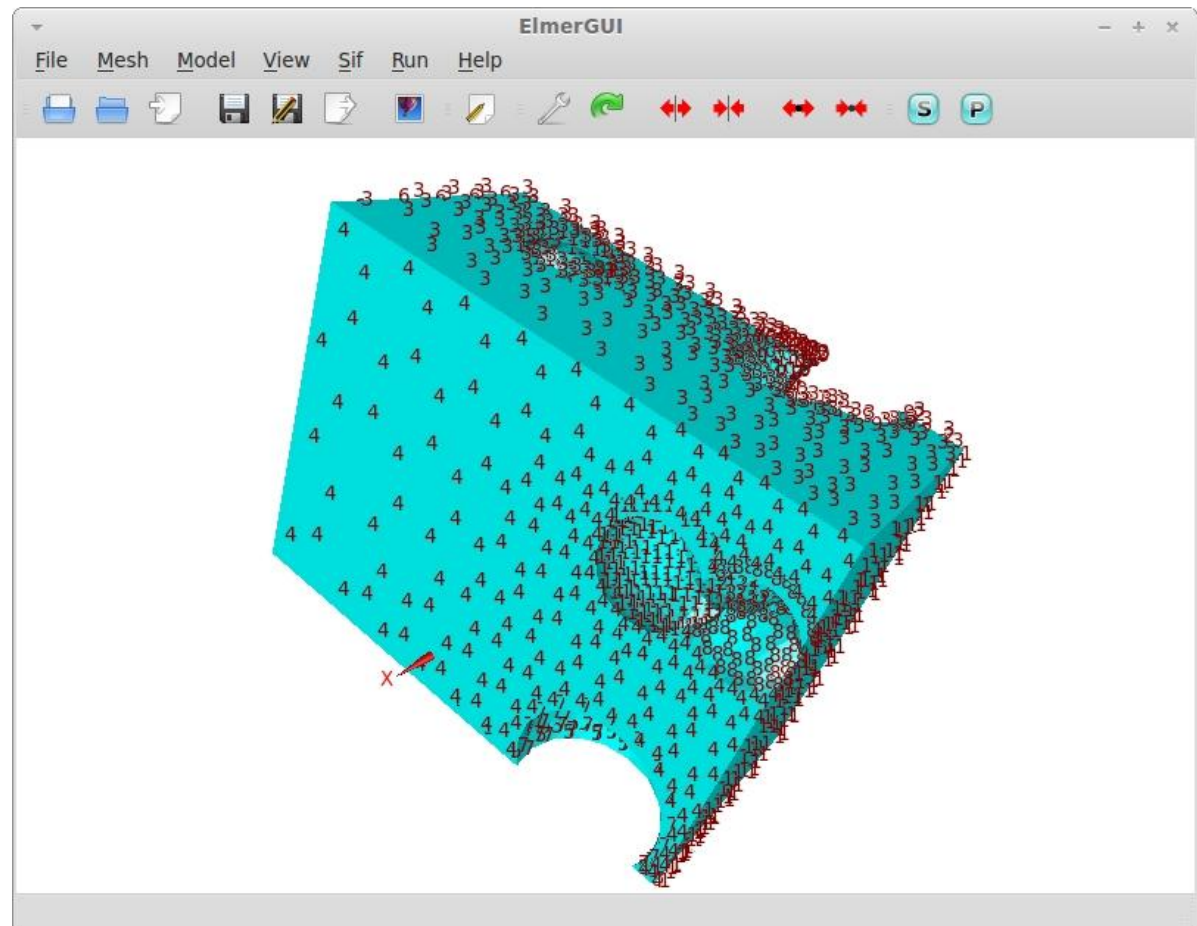
# Set-up

- ➡ Assigning wall boundary
- ➡ Press **+Add**
- ➡ Toggle off **Model** → **Set boundary properties**



# Set-up

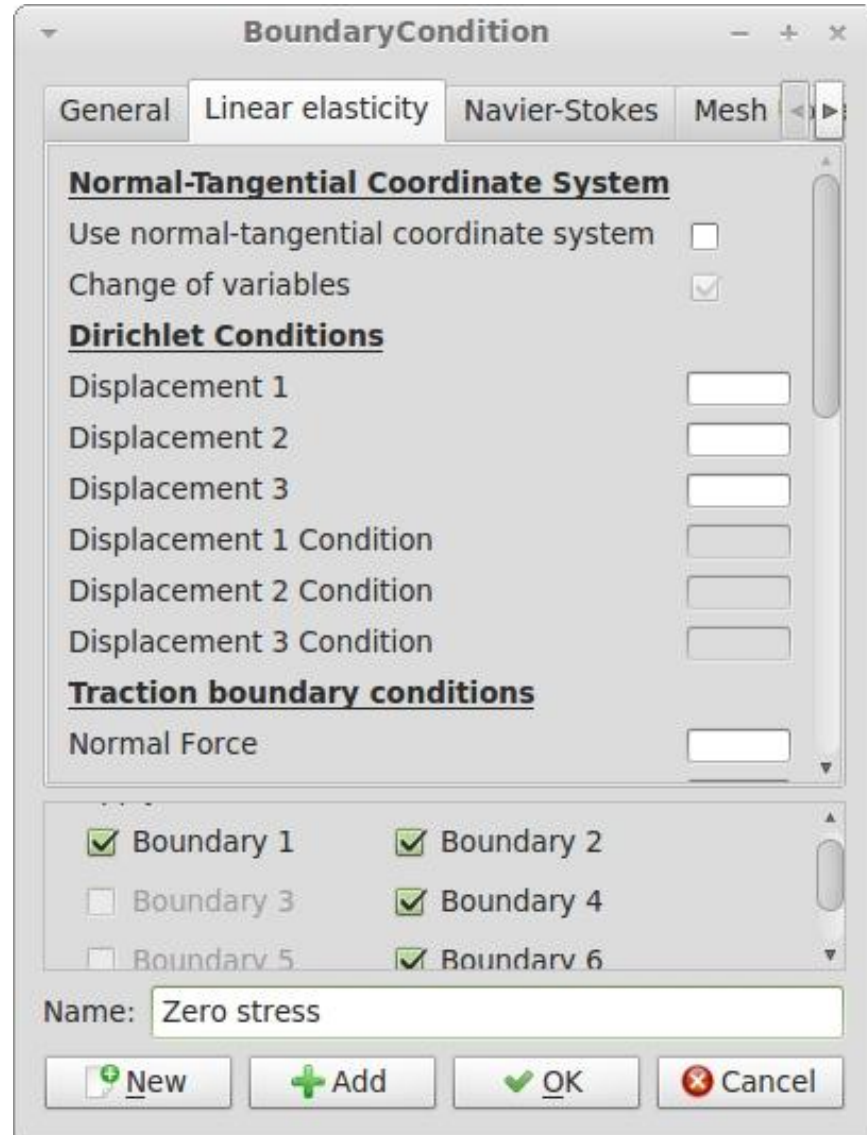
- Visualization of boundary ID in ElmerGUI:
  - **View** → **Numbering** → **Boundary index**





# Set-up

- ➔ **Model**
  - **Boundary condition** → **Add...**
- ➔ **Linear elasticity**
  - Zero stress
  - Toggle remaining boundaries with  $ID \leq 6$
  - **Add + OK**



The image shows a software window titled "BoundaryCondition" with four tabs: "General", "Linear elasticity", "Navier-Stokes", and "Mesh". The "Linear elasticity" tab is selected. Inside this tab, there are three main sections: "Normal-Tangential Coordinate System", "Dirichlet Conditions", and "Traction boundary conditions".

**Normal-Tangential Coordinate System**

- Use normal-tangential coordinate system: ☐
- Change of variables: ☒

**Dirichlet Conditions**

- Displacement 1:
- Displacement 2:
- Displacement 3:
- Displacement 1 Condition:
- Displacement 2 Condition:
- Displacement 3 Condition:

**Traction boundary conditions**

- Normal Force:



At the bottom, there is a list of boundaries with checkboxes:

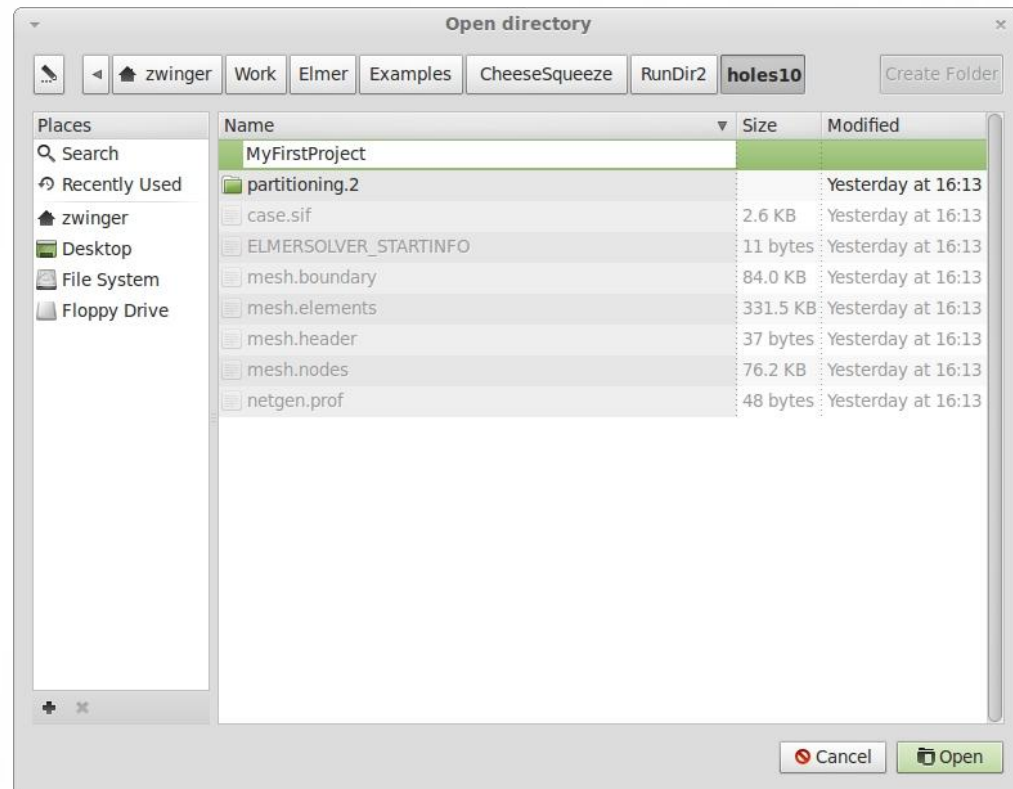
|                                                |                                                |
|------------------------------------------------|------------------------------------------------|
| <input checked="" type="checkbox"/> Boundary 1 | <input checked="" type="checkbox"/> Boundary 2 |
| <input type="checkbox"/> Boundary 3            | <input checked="" type="checkbox"/> Boundary 4 |
| <input type="checkbox"/> Boundary 5            | <input checked="" type="checkbox"/> Boundary 6 |

Below the list, there is a "Name:" field containing the text "Zero stress". At the bottom of the window, there are four buttons: "New" (with a plus icon), "Add" (with a green plus icon), "OK" (with a green checkmark icon), and "Cancel" (with a red X icon).

# Set-up



## • Finishing the setup:

- **SIF** → **generate**
- Save the project:
  - Either by **File** → **Save project**
  - Or by the symbol  in the task bar
  - Create new folder
- Save the files:
  - Either by **File** → **Save**
  - Or by the symbol  in the task bar




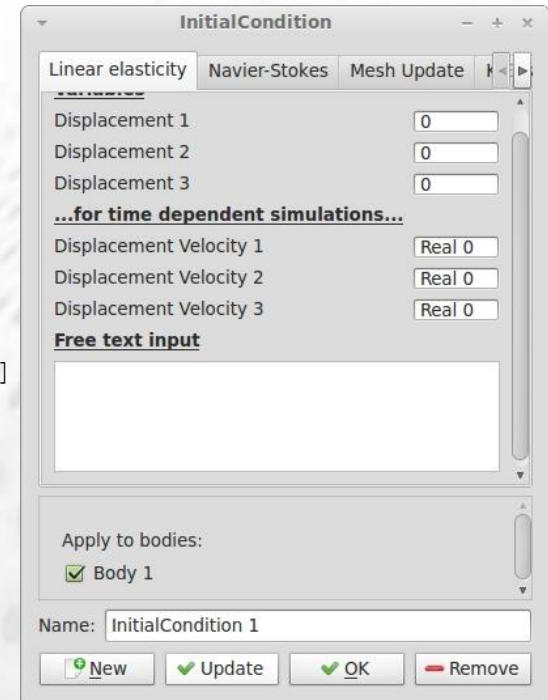
# Run

## ➤ Run the case:

- Either **Run**→**start solver** or 
- The symbol then will change color 
- A log window will occur
- ... and display an error message:



```
ERROR:: Model Input: Unknown specifier: [0]
ERROR:: Model Input: In section: [initial condition 1]
ERROR:: Model Input: For property name:[displacement velocity 3]
```

- Problem: *Displacement Velocity* not in Keyword Database
- Re-open Initial Condition 1
- Cast the values with *Real*
- **Update** and **OK**
- **SIF** →**generate** and 



# Post-processing

## ➤ Two post processors:

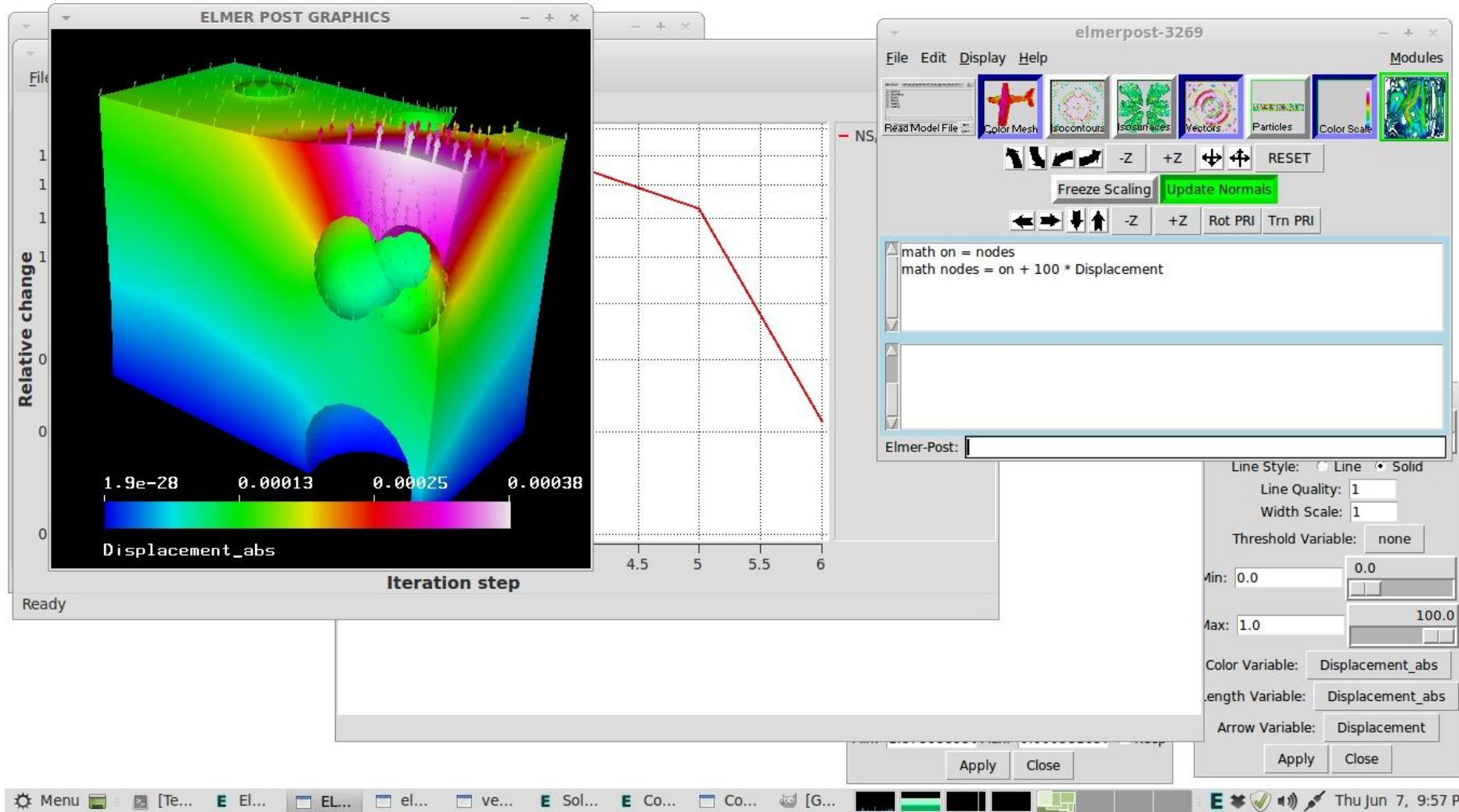
1. Internal VTK based: **Run** → **Postprocessor (VTK)**
2. Externally linked ElmerPost (legacy postprocessor of Elmer):
  - Either **Run** → **Start postprocessor** or 
  - Changes color if active 

## ➤ ElmerPost manipulations:

`math on = nodes`

`math nodes = on + 100 * Displacement`

# Post-processing



## Further steps

- Why is the cheese not squeezed, but pulled?
  - Mind, that surface normal (that defines the direction of normal force) by definition is pointing outwards.
- Task: change force to correct sign and re-run case
  - Remember to create the SIF and save it, before re-running