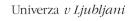


## Guided examples using ELMER FEM

#### Matic Brank,

Faculty of Mechanical Engineering, UL

06/2021







CINECA CINECA CONSOLZIO

VSB TECHNICAL

IT4INNOVATIONS NATIONAL SUPERCOMPUTING CENTER



Co-funded by the Erasmus+ Programme of the European Union This project has been funded with support from the European Commission.

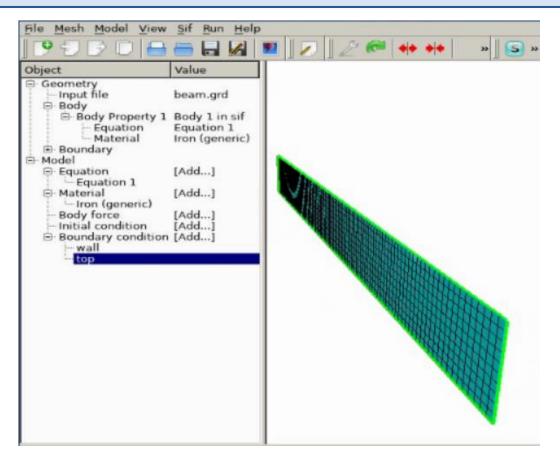
This publication [communication] reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

### Quick software overview



- Elmer FEM
  - Finite element software package for multiphysical problems
  - Multiple modules within Elmer FEM:
    - Elmer Solver: Kernel of Elmer FEM, which performs FEM computations
    - Elmer Grid: Module for mesh preparation/conversion
    - Elmer GUI: Graphical user interface built around ElmerSolver with the intention to ease the case preparation and computation for users
  - Open source
  - Source code and documentation available at

<u>GitHub - ElmerCSC/elmerfem: Official git repository of Elmer FEM</u> <u>software</u>

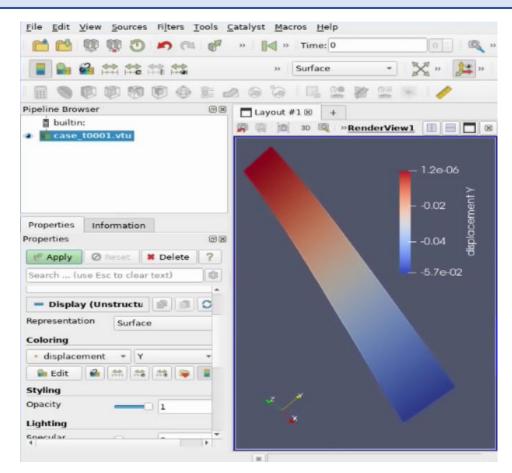


### Quick software overview

# Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

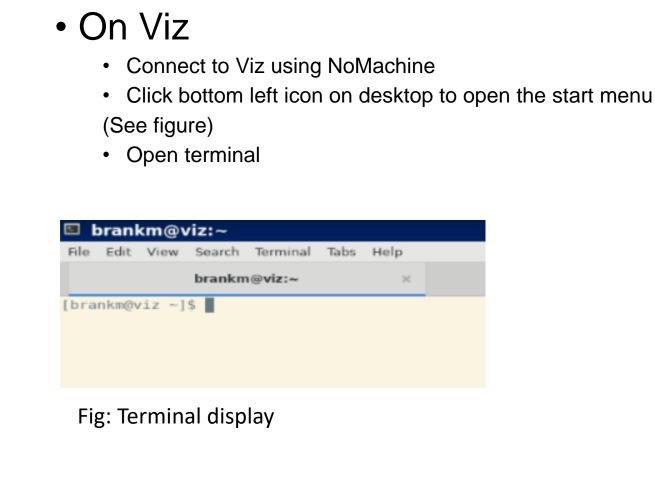
- Paraview
  - multiple-platform application for interactive, scientific visualization
  - Open source
  - Many useful visualization capabilities:
    - Contours and isosurfaces for scalar and vector fields
    - Streamlines
    - Advanced data manipulation through Python
  - Support of variety of formats
    - VTK, VTU
    - CGNS
  - Source code and documentation available at

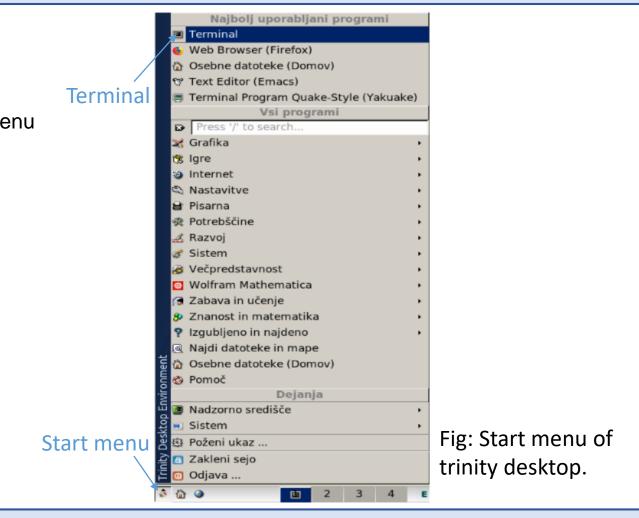
<u>GitHub - Kitware/ParaView: VTK-based Data Analysis and Visualization</u> <u>Application</u>



## Software loading and running

# Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP





Sctrain KNOWLEDGE Software loading and running

- To open Elmer GUI:
  - To load Elmer into environment, type into terminal
    - module load elmer/foss-2018b
  - To run Elmer GUI
    - ElmerGUI
- To open ParaView
  - To load ParaView into environment, type into terminal
    - module load ParaView/5.8.1-foss-2020b-mpi
  - To run ParaView
    - paraview

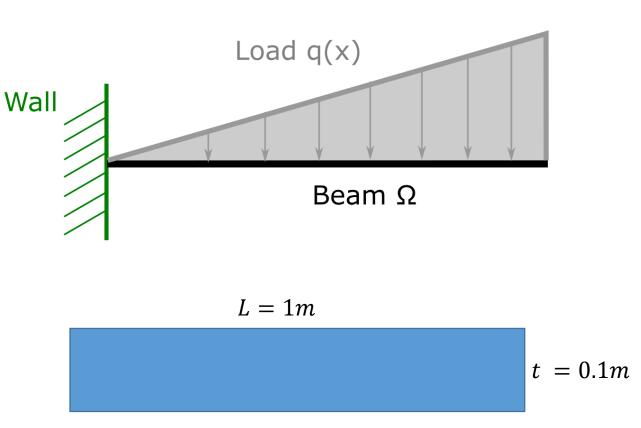
SUPERCOMPUTING

## Loaded elastic beam - 2D SCtrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

- Case definition
  - 2D case
  - Homogeneous, elastic beam, defined on x y plane

(length 1m and thickness 0.1m)

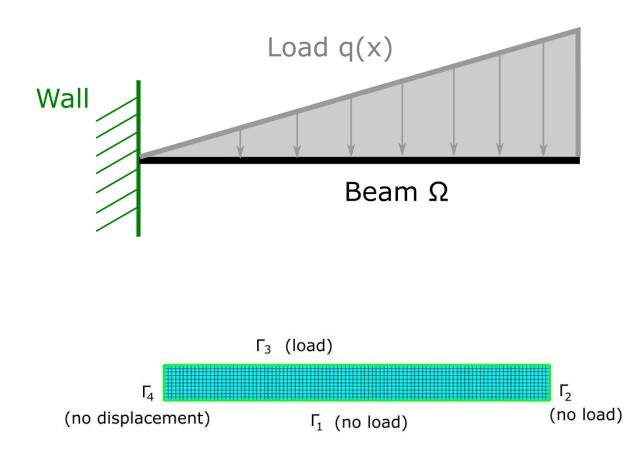
- Rigid support at the wall
- Space dependent mechanical load, which grows linearly from 0 to  $q_0 = 1e7N$
- Goal
  - Obtain the displacement of the beam



## Loaded elastic beam - 2D SCtrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

- Mathematical definition of the problem
  - Domain of computation:
    - Beam area  $\Omega$
  - Boundary conditions
    - Mechanical load  $\Gamma_3$
    - No displacement at  $\Gamma_4$
    - Zero load on  $\Gamma_{\! 1}$  and  $\Gamma_{\! 2}$
- Problem solution

$$\begin{cases} -\nabla \sigma = 0 & on \ \Omega \\ \sigma = \lambda tr[\varepsilon(u)]I + 2\mu\varepsilon(u) & on \ \Omega \\ u = 0 & on \ \Gamma_4 \\ \sigma n = 0 & on \ \Gamma_1 \ and \ \Gamma_2 \\ \sigma n = -q & on \ \Gamma_3 \end{cases}$$



### Case preparation - overview

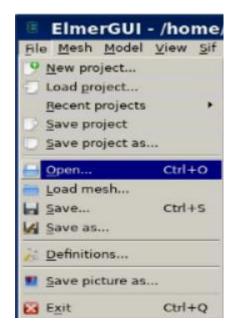


- 1. Definition of domain of computation (Elmer GUI)
  - Mesh preparation/definition
- 2. Definition of type of physical problem (Elmer GUI)
  - Definition of equation to be solved
- 3. Definition of material properties (Elmer GUI)
- 4. Definition of boundary/initial conditions (Elmer GUI)
- 5. Computation (Elmer GUI)
- 6. Postprocessing of results (ParaView)

Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

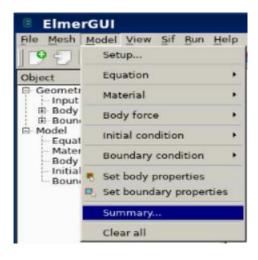
- 1. Definition of domain of computation (Elmer GUI)
  - Import mesh into Elmer GUI
  - File->Open
  - Navigate to your folder and select beam.grd
  - Use Mouse wheel/left button to rotate and zoom mesh

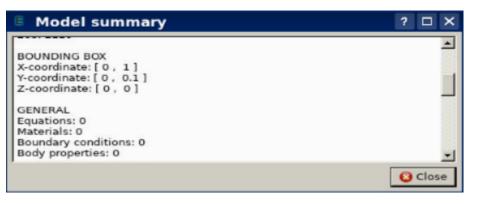
ElmerGUI		-	×
File Mesh Model View			
Object ⊖ Geometry ⊢ Input file ⊕ Body ⊕ Boundary ⊕ Model ⊢ Equation ⊷ Material ⊨ Body force ⊢ Initial condition ⊷ Boundary condition	Value beam.grd [Add] [Add] [Add]		
			1





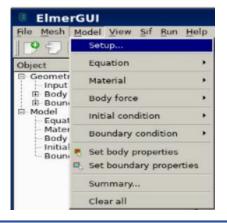
- Go to Model->summary to observe mesh information
- Check that mesh contains 3221 nodes
- How many surface elements are in the mesh?
- What is the type of the surface elements?
  - Triangles, quadrangles?
- Check Elmer documentation to see ID numbers for different elements







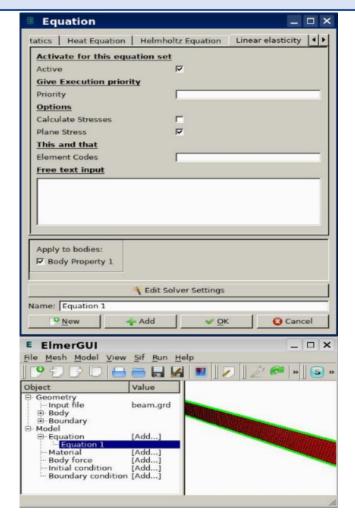
- 2. Definition of type of physical problem
  - In Model->Setup... one can define properties related to the simulation, such as results directory, input mesh directory, name of output file, constants, etc...
  - Make sure the simulation type is set to Steady state



Setup			? 🗆 :
Header Check keywords w MeshDB . Include path . Results directory . Free text	ram	Г.	
Coordinate system Coordinate mapping Simulation type Output intervals Coordinate Scaling	5 Cartesian 1 2 3 Steady state 1 Case.sif	Steady state max. iter Timestepping method BDF order Timestep intervals Timestep sizes Angular Frequency Post file	
Constants Gravity Stefan Boltzmann Vacuum permittivity	0 -1 0 9.82 5.67e-08 8.8542e-12	Boltzmann 1.380 Unit charge 1.602	
Free text			Apply

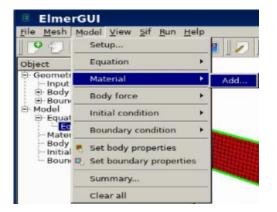


- In Model->Equation->Add... one can select equations to be solved during the computation. For the bending of beam, select Linear elasticity tab
- Tick checkboxes Active and Plane stress
- Assign the equation to the Mesh body (Tick the box in Apply to body)
- In the end, click OK to accept new changes





- 3. Definition of material properties
  - The material of the beam in our Study is iron with Poisson's ration of 0.29 and Young's modulus of  $1.93 \cdot 10^9 N/m^2$
  - Click on Model->Material->Add...
  - Click on Material library and select Iron (generic)
  - Click OK
  - Tick box Apply to bodies to set the material to mesh.
  - Click OK



E Equation	Helmholtz Eq	uation Linear	elasticity
Properties			
Youngs modu	lus	193.053e9	
Poisson ratio		0.29	
Damping coe	fficient	<b>_</b>	
Rayleigh Dan	nping	E .	
Rayleigh alph	าล		
Rayleigh beta			
Stress 6-vecto	or		
Strain 6-vecto	ər		
Material prie	cipal directio	ns	
Rotate Elastic	ity Tensor	Г	
Unit Vector 1			
ime: Iron (ge		erial library	
ame: Iron (ge <u>9 N</u> ew		orial library	Canc
Material	eneric) Add library nperature)		Cance ? C
Material	Add Add <b>library</b> nperature) generic) ainless steel (A	<u> </u>	
<u>New</u> Material Air (room ten Aluminium (g Austenitic sta Copper (gene Ethanol (room	eneric) Add IIbrary nperature) generic) ainless steel (A ric) n temperature	<u>و م</u> لا پر Steel 201)	
<u>New</u> Material Air (room ten Aluminium (g Austenitic sta Copper (gene Ethanol (room Rused Silica ( Glass (borosi	Add hperature) peneric) ainless steel (A wric) n temperature 25 C) licate)	<u>م</u> لا <u>م</u> لا سر Steel 201)	
<u>New</u> Material Air (room ten Aluminium (g Austenitic sta Copper (gene Ethanol (room Rused Silica ( Glass (borosi	Add Add library perature) peneric) ainless steel (A wric) n temperature 25 C) licate) m temperature	<u>م</u> لا <u>م</u> لا سر Steel 201)	
• New Material Air (room ten Aluminium (g Austenitic sta Copper (gene Ethanol (roon Rused Silica ( Glass (borosi Glycerol (roo Gold (generic) Ton (generic)	Add Add Ibrary perature) peneric) ainless steel (A rric) n temperature 25 C) licate) m temperature 25 C)	<u>م</u> لا <u>م</u> لا سر Steel 201)	
<u>New</u> Material Air (room ten Aluminium (c) Austenitic sta Copper (gene Ethanol (roon Rused Silica ( Glass (borosi Glycerol (roo Gold (generic) Lead (generic) Lead (generic) Lead (generic) Lead (generic)	Add Add Ibrary perature) peneric) sinless steel (A tric) n temperature 25 C) licate) m temperature 25 C) C)	<u>م</u> لا <u>م</u> لا سر Steel 201)	
<u>New</u> Material Air (room ten Aluminium (g Austenitic sta Copper (gene Ethanol (room Rused Silica ( Glass (borosi Gold (generic Iron (generic) Lead (generic Oil, olive (25 Platinum (ge	Add http://www.add/ http://wwww.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://www.add/ http://wwwwwwwwww.add/ http://www.add/ http://www.add/ http://www	<u>م</u> لا <u>م</u> لا سر Steel 201)	
<u>New</u> Material Air (room ten Aluminium (g Austenitic sta Copper (gene Ethanol (room Rused Silica ( Glass (borosi Ing Generic Dil, olive (25 Platinum (gen Polyzarbonat Polyziny) chil	Add Add Iibrary perature) peneric) ainless steel (A ric) n temperature 25 C) licate) m temperature 25 C) c) c) c) c) c) c) c) c) c) c	<u>و</u> لا (K Steel 201) (۲)	
Air (room ten Aluminium (g Austenitic sta Copper (gene Ethanol (roon Fused Silica ( Giass (borosi Giycerol (roo Gold (generic) Lead (generic) Lead (generic) Lead (generic) Joli (ve (25 Platinum (ge Polycarbonat	Add Ilibrary nperature) eneric) n temperature peneric) n temperature 25 c) licate) m temperature 25 c) licate) e (generic) oride (generic) i)	<u>و</u> لا (K Steel 201) (۲)	
Air (room ten Aluminium (g Austenitic sta Copper (gene Ethanol (room Rused Silica ( Glass (borosi Glycerol (roo Gold (generic Ticol (generic) Lead (generic) Lead (generic) Jianum (generic) Platinum (generic) Polycarbonat Polycarbonat Polycarbonat	Add Add Ibrary perature) peneric) ainless steel (A ric) n temperature 25 C) licate) m temperature 25 C) n temperature 25 C) n temperature 25 C) neric) e (generic) oride (generic) oride (generic) )	<u>و</u> لا (K Steel 201) (۲)	



- 4. Definition of boundary conditions
  - To define a boundary condition, click on Model->Boundary condition->Add...
  - Navigate to Linear elasticity
  - Here, we can define displacements and loads on the mesh boundaries

ElmerGUI	BoundaryCondition	ElmerGUI _ 🗆 X
ElmerGUI         File       Mesh       Model       Yiew       Sif       Run       Help         Object       Equation       •         Object       Equation       •         Imput       Body force       •         Initial condition       •         Material       •         Model       Initial condition         Material       •         Boundary condition       •         Add       •         Boundary properties       •         •       •         •       •         •       •         •       •         •       •         •       •         •       •         •       •         •       •         •       •     <	BoundaryCondition       Image: Condition         General       Electrostatics       Heat Equation       Helmholtz Equation       Linear elasticity       Mesh Update       Nix         Normal-Tangential Coordinate System       Image: Conditions       Image: Condition	ElmerGUI       Image: Construction of the sector of the sect
Summary Clear all	Displacement 3 Condition       Traction boundary conditions       Normal Force       Apply to boundaries:       Boundary 1       Boundary 2       Boundary 3       Boundary 4       Name:       Boundary Condition 2       P New       Add       Y OK	



- Definition of wall boundary condition
  - Set Displacement 1 and Displacement 2 to 0. Identification number 1 refers to x-coordinate and 2 refers to y-coordinate
  - Select appropriate Edge on the Mesh
  - Set name of boundary condition to Wall
  - Click OK
- Definition of mechanical load q(x)
  - Create a new boundary condition (Model->Boundary condition->Add...)
  - Navigate to Linear elasticity
  - Now we have to define linear load that points in <u>y-direction</u> and is linearly increasing along <u>x-direction</u>. To do that:
    - Go to Force 2 (remember, 2 refers to y-direction)
    - The variable to specify: Variable Coordinate 1; Real; 0 0; 1 -1.0e7; End
  - Set boundary condition name to Load

• Explanation of Force 2 definition

- The semicolon in Elmer FEM specifies the definition of new keyword
- The first keyword (Variable Coordinate 1) specifies that Force 2 is changing its value along coordinate 1 (remember, 1 refers to x-direction)
- Then the user can define a table of x (coordinate), F (force) values. So the next keywords look like

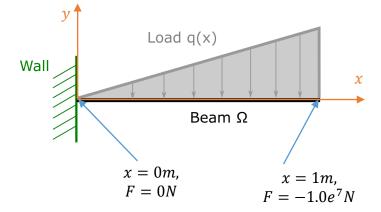
Real;

0 0;

1 -1.0e7;

End

- Keyword Real defines the type of numbers in a table
- Keyword 0 0 specifies that at x = 0m, the force is F = 0N
- Keyword 1 -1.0e7 specifies that at x = 1m, the force is  $F = -1.0e^7N$
- Keyword End specifies end of table



Sctrain KNOWLEDGE

SUPERCOMPUTING



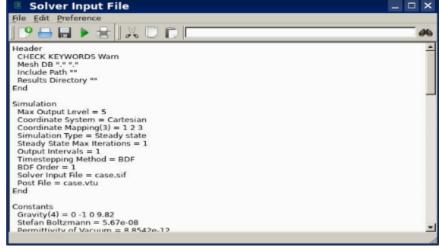
- In case of elasticity solver, the Mesh boundaries are automatically set to 0 load
- Thus it is not necessary to define zero force on remaining two edges
- As an exercise, one can set the forces at these two edges to 0

### Running the case



- 5. Computation
  - Before running the case, one can observe the solver input file (extension .sif)
  - SIF file is a file, generated by Elmer GUI.
  - The file contains the case set-up with all the parameters specified by the user in Elmer GUI
  - This file is then taken as input by the Elmer kernel (ElmerSolver), which then computes the solution based on these parameters
  - Click on Sif->Generate and then Sif->Edit...
  - Here you can manually change properties in the editor

ElmerGUI							
File	Mesh	Model	View	Sif	Run	Help	
15	10	3D	-		Gener	rate	Ctrl+G
Obje	ect			2	Edit		Ctrl+S



#### 19

SUPERCOMPUTING

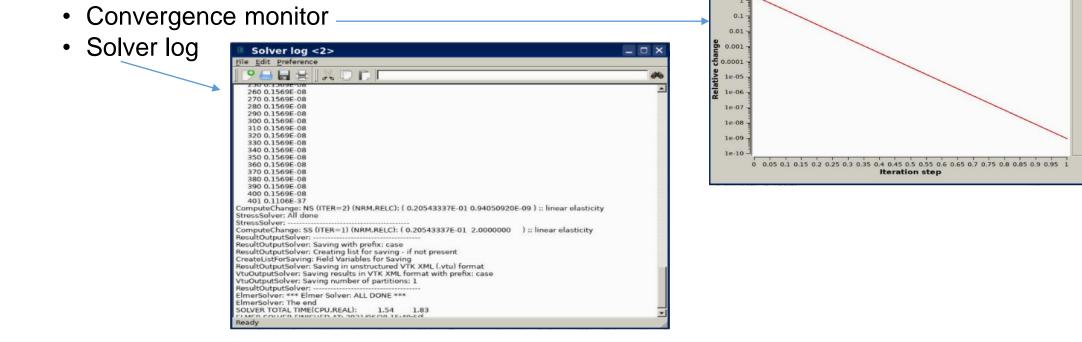
NS/linear elasticit

Sctrain KNOWLEDGE PARTNERSHIP

**Convergence** history

## Running the case

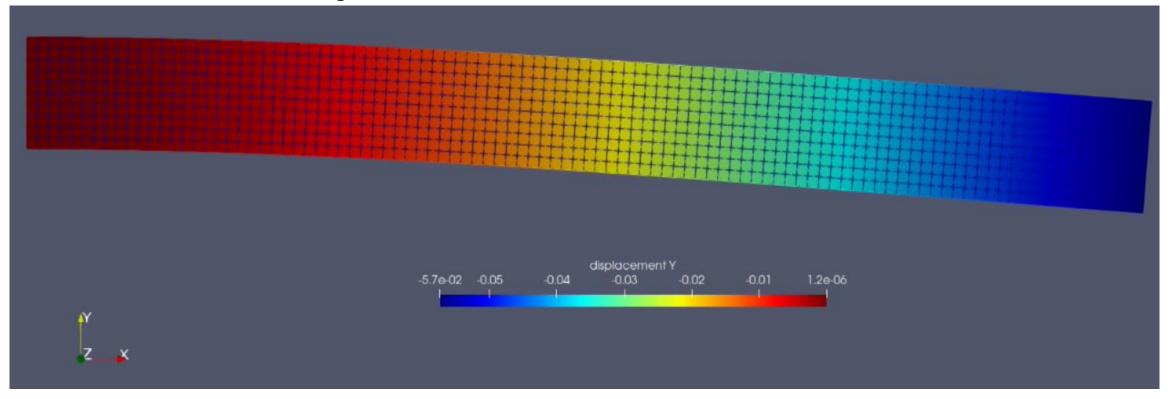
- To run the case, click "Save and run" button (see figure)
- This button will then ask you to select a project folder. Then it will save the changes and run the case. Convergence monitor <2> \_ \_ × le <u>V</u>iew
- Two windows open:



#### Post-processing



Use mouse wheel/right button to rotate and zoom result



#### Post processing



 To change coordinate of displacement, navigate to Properties->coloring->displacement

Properties	Information				
Properties					8
of Apply	Ø Reset	# Delet	e )	?	
Search (us	se Esc to clear te	×t)			1
- Propert	ties (case_tooo	1.vtu)			
✓ Cell/Point	Array Status				$\odot$
personal data					
V 🧩 Geom V 👬 displa					
🗸 🏗 displa		GridReprese		0	
<ul> <li>displa</li> <li>Display</li> </ul>	(Unstructured	GridReprese	ø		-
displa     Display Representation	(Unstructured	Provide the second s	P	0	-
🗸 📫 displa	(Unstructured	Magnitude	P	0	-

- To get the maximum displacement, navigate to Edit->Find data...
- In the dialog, select Point(s) and set displacement(1) to "is min"

	ection					
Find Poi	nt(s) =	from case	_t0001.	vtu		-
displacem	ent (1)		is mi	n	*	+
4						10+
-			Run S	Selection Query		
Current Se	lection	(case_too	01.vtu	: 0)		
Show: • P	oint(s)				* 🗌 Inv	vert selection
Point ID		Points		Points_Magnitude		displacem
0 3220	1	0.0425808	0	1.00479	0.00388738	-0.0574192
						ŀ
۰ Selection I	Display	Properties	5			
Selection I     Selection		Propertie	s Cell Lai		Point Labels	- 00



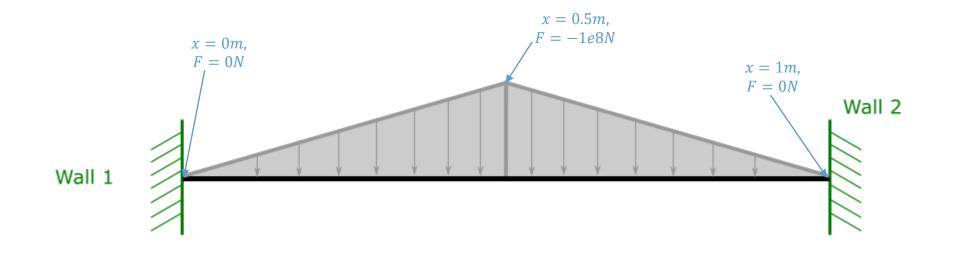
Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

- Change maximum load on the right edge (at x = 1m) to  $F = -1.0e^8N$  and run the case again.
- Change the load again to  $F = -1.0e^9N$ . Run the case and observe the maximum displacement with increase of load



Sctrain RNOWLEDGE PARTNERSHIP

- Add a new wall boundary condition to the right side of the beam (where x = 1m, set displacement in x and y direction to 0)
- Modify the load at the top to have the following dependency in x-direction. What is the maximum displacement in y-direction?



#### **Exercise 3**



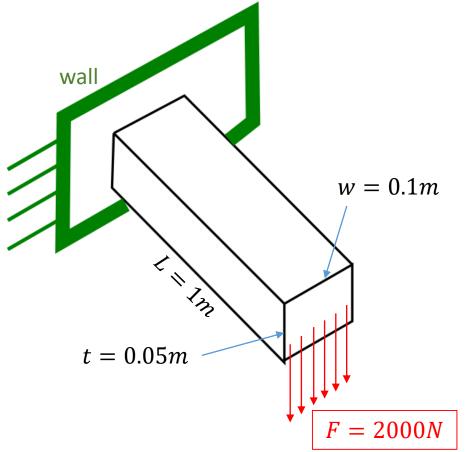
- Transient loading
  - Change Simulation Type to Transient (Setup->simulation type)
  - Set the Time Stepping Method to bdf
  - Set the Time step intervals to 200
  - Set the Time Step Sizes to  $1e^{-4}$

SUPERCOMPUTING

#### Loaded elastic beam - 3D

Case definition

- A homogeneous, elastic beam is rigidly supported on one end. On the other end, the force is 2000N due to an attached object. The weight of the beam itself is also included as additional load.
- The length of the beam is 1m, thickness is 0.05 m and width is 0.1m
- Young's modulus is  $10 \cdot 10^9 N/m^2$  and Poisson's ratio is 0.37
- Density of the material is  $550kg/m^2$
- Goal
  - Calculate displacement field of the beam
  - Obtain the location and value of the maximum displacement
  - Calculate stress field



Sctrain KNOWLEDGE PARTNERSHIP



- 1. Definition of domain of computation
  - Import mesh into Elmer GUI
  - Navigate to your folder and select beam3d.grd
  - Use Mouse wheel/left button to rotate and zoom mesh
  - Verify that mesh consists of 6073 nodes and of 1200 quadratic hexahedral elements



#### 2. Definition of type of physical problem

- **Set the** simulation type **to** Steady state
- Set equation to Linear elasticity
- In Linear elasticity, set checkbox next to Active
- We also want to compute stresses as a post-processing step, so set checkbox of Calculate Stresses
- **Click on** Edit solver settings:
  - **check** Iterative method **and set it to** GCR
  - Set Preconditioning to ILU1

Activate for this equation set   Active   Give Execution priority   Priority   Options   Calculate Stresset   Plane Stress   This and that   Element Codes   Free text input   Control   Max. iterations   S00   Convergence tol.   1.0e-10   Preconditioning   IUJ   Preconditioning   IUJ   Apply to bodies:   Body Property 1   Name: Equation 3   Name: Equation 3   Name: Equation 3   Name: Equation 3   Name: Add _ OK @ Cancel	Equation _ 🗆 ×	Solver control for Linear elasticity	? _ 🗆 ×
	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 Property 1 Edit Solver Settings Name: Equation 3	Method C Direct Banded T C Iterative GCR T C Multigrid Jacobi T Control Max. iterations 500 Convergence tol. 1.0e-10 Preconditioning [10] T ILUT tolerance 1.0e-3 Residual output 10 Prec. recompute 1 BiCGStabl order 2	Linear system
Edit Solver Settings	Edit Solver Settings		🛩 Apply



- 3. Definition of material properties
  - Create a new material and define:
    - Density,
    - Poisson's ratio and
    - Young's modulus
  - Assign the material to 3D beam

#### 29

- 4. Definition of boundary conditions
  - Define zero displacement on left face that lies in x-z plane (at the wall – refer to figure on slide 24)
  - Note that second boundary condition distributes the load of 2000N uniformly on the area of  $t \cdot$  $w = 0.05m \cdot 0.1m = 5.0e^{-3}m^2$
  - The weight of the beam should be defined through Body Force
    - Navigate to Model->Body force->Add...->Linear elasticity
    - Set the force in negative y-direction through mass of beam and gravitational acceleration
    - Apply the body force to the mesh body

🗉 Elme	rGUI	- /wo	rk/	brar	nkm/	elmerg
<u>File M</u> esh	Model	View	Sif	Run	Help	
9901	Set	Setup				
Object	Equ	Equation •				
Geometr		erial			•	
⊕ Body ⊕ Boun	Boo	ly force	e		•	Add

BodyForce	×
holtz Equation Linear elasticity	Mesh Update
Volume forces	-
Force 1	
Force 2	
Force 3	
Pressure	
Stress 6-vector	
Strain 6-vector	
<b>Bodywise Dirichlet Conditions</b>	
Displacement 1	
Displacement 2	
Displacement 3	
Displacement 1 Condition	
Distanta condition	
Apply to bodies:	
F Body Property 1	
Name: BodyForce 2	
New Add 🖌	OK Orancel

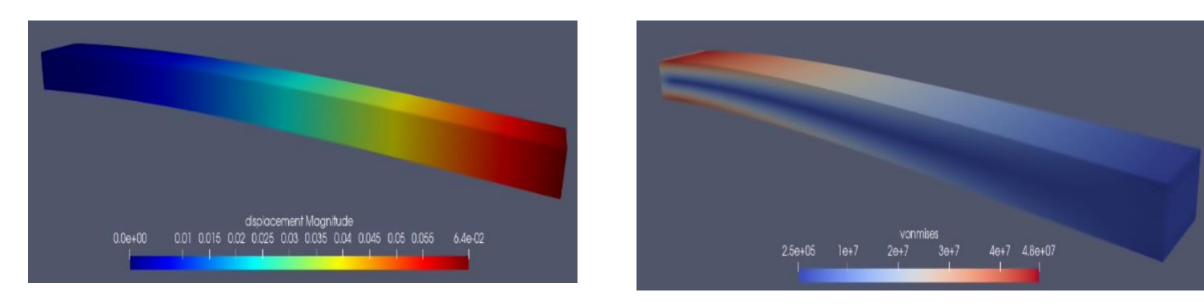




- 5. Computation
  - Generate and check SIF file
  - Save the case and run it



- 5. Post processing
  - What is the maximum displacement and where is its location
  - Plot displacements and Von mises stresses



#### Exercise



#### Gravity in x-direction

- The beam should be more rigid if the beam is orientated differently
- Change the direction of gravity and force of attached object in the negative xdirection

#### Exercise



#### Gravity in x-direction

- The beam should be more rigid if the beam is orientated differently
- Change the direction of gravity and force of attached object in the negative xdirection

SUPERCOMPUTING

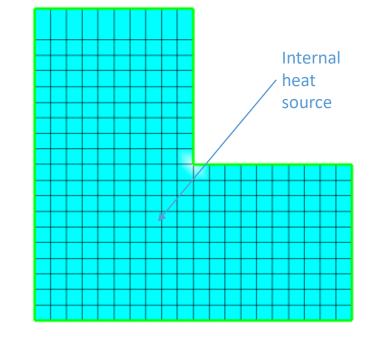
#### Heat equation

Case definition

• A structure in L-shape is heated by internal source with magnitude of  $1W/m^3$ . The density of the structure is  $1kg/m^3$  and the heat conductivity is 1W/mK. The temperature of boundaries of the structure is set to 0.

Goal

 Obtain temperature distribution in the structure



Sctrain KNOWLEDGE

 $T_{boundary} = 0^{\circ} C$ 

#### Heat equation



Mathematically the problem to be solved is

$$\begin{cases} -k\Delta T = \rho f & on \Omega \\ T = 0 & on \Gamma \end{cases}$$

- Parameters
  - *k* ... heat conductivity
  - T ... temperature
  - $f \dots$  heat source

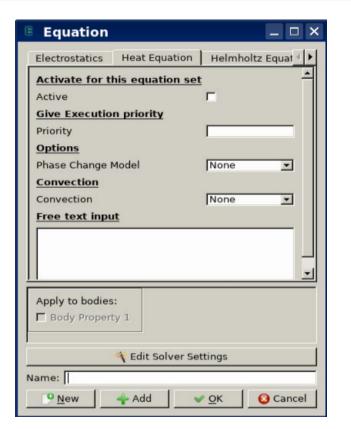
#### Heat equation

Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

- 1. Definition of domain of computation
  - Import mesh into Elmer GUI
  - Navigate to your folder and select angle.grd
  - Use Mouse wheel/left button to rotate and zoom mesh
  - Verify that mesh was successfully imported and that it consists of 341 nodes and of 300 bilinear elements

# 2. Definition of type of physical problem

- Set the simulation type to Steady state
- Set equation to Heat equation
- In Heat equation check checkbox next to Active



- 3. Definition of material properties
  - Create a new material and define:
    - Density,
    - Heat conductivity (in Heat)
  - Assign the material to the body

Material		_ 🗆 ×
General Electrostatics	Heat Equation	Helmholtz 4 🕨
Properties Heat Conductivity Heat Conductivity Model Emissivity Turbulent Prandtl Number Enthalpy Specific Enthalpy Pressure Coefficient <u>Free text input</u>	None 0.85	
Apply to bodies: Body Property 1 Mame: Material 2 Name: Material 2 Add	iterial library	Cancel

- 4. Definition of boundary conditions
  - Define zero temperature on boundaries
  - Temperature is defined under Dirichlet conditions
  - Apply temperature to all boundaries

BoundaryCondition _ D ×
General Electrostatics Heat Equation Helm
Dirichlet Conditions
Temperature
Temperature Condition
Heat Flux conditions
Heat Flux
Heat Transfer Coeff.
External Temperature
Latent heat of phase change
Phase Change
Heat Gap
Heat Gap
Radiation Settings
Radiation None 💌
Emissivity
Radiation Boundary Open
Apply to boundaries:
F Boundary 1 F Boundary 2
F Boundary 3 F Boundary 4
🗖 Boundary 5 🗖 Boundary 6
Name: BoundaryCondition 3
<u> </u>



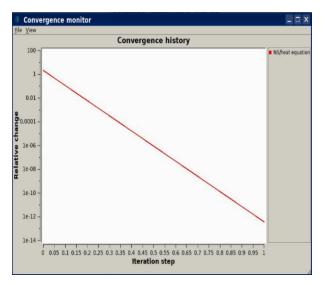
- 4. Definition of boundary conditions
  - Define volumetric heat source in the structure
  - A Body force represents the right-hand side of the equation which in this case represents the heat source
  - Navigate to Body force
  - Set Heat Source to 1 and apply the condition to a body, then click OK

BodyFord	e	_ 0
Electrostatics	Heat Equation	Helmholtz Equat
Volume source	es	4
Heat Source		
Friction Heat		Г
Joule Heat		Г
Bodywise Dir	ichlet Conditions	
Temperature		
Temperature C	ondition	
<b>Perfusion</b>		
Perfusion Rate		
Free text inp	ut	
Apply to bodies	CONSISTER AND A DESCRIPTION OF A DESCRIP	
ame: BodyFor		OK OK Cance

- 5. Computation
  - Generate and check SIF file
  - Save the case and run it

Solver log

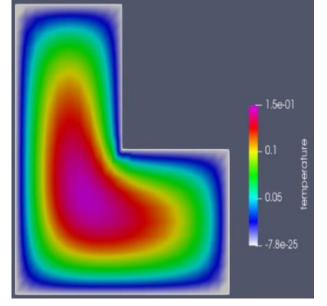
HeatSolve: iter: 1 Assembly: (s) 0.00 0.00	
HeatSolve: iter: 1 Solve: (s) 0.00 0.00	
HeatSolve: Result Norm : 7.7824030917542059E-002	
HeatSolve: Relative Change : 2.000000000000000	
HeatSolve:	
HeatSolve:	
HeatSolve:	
HeatSolve: TEMPERATURE ITERATION 2	
HeatSolve:	
HeatSolve:	
HeatSolve: Starting Assembly	
HeatSolve: Assembly:	
HeatSolve: Assembly done 1.0.4952E-11	
ComputeChange: NS (ITER=2) (NRM,RELC): ( 0.77824031E-01 0.366809)	Inc. 10 hast smither
HeatSolve: iter: 2 Assembly: (s) 0.00 0.01	UE-12 / :: neat equation
HeatSolve: iter: 2 Solve: (s) 0.00 0.00	
HeatSolve: Result Norm : 7.7824030917570605E-002	
HeatSolve: Relative Change : 3.6680970111809652E-013	) ~ heat equation
HeatSolve: Relative Change: 3.6680970111809652E-013 ComputeChange: SS (ITER=1) (NRM,RELC): (0.77824031E-01 2.000000	) :: heat equation
HeatSolve: Relative Change: 3.6680970111809652E-013 ComputeChange: SS (ITER=1) (NRM.RELC): ( 0.77824031E-01 2.000000 ResultOutputSolver:	) :: heat equation
HeatSolve: Relative Change: 3.6680970111809652E-013 ComputeChange: SS (ITER=1) (NRM.RELC): (0.77824031E-01 2.000000 ResultOutputSolver: ResultOutputSolver: Saving with prefix: case	) :: heat equation
HeatSolve: Relative Change: 3.6660970111809652E-013 ComputeChange: SS (ITER=1) (NRM.RELC): (0.77824031E-01 2.000000 ResultOutputSolver: Saving with prefix: case ResultOutputSolver: Creating list for saving - if not present	) :: heat equation
HeatSolve: Relative Change : 3.6680970111809652E-013 ComputeChange: S5 (ITRE-1) (NMA.RELC): (0.77824031E-01 2.00000) ResultObutSolver: Suring with prefix: case ResultObutSolver: Saving with prefix: case ResultObutSolver: Saving ist for saving - if not present CreateListForSaving: Field Variables for Saving	) :: heat equation
HeatSolve: Relative Change : 3.660097011809652F-013 ComputeChange: S0 (TER=1) (NRM.RELC): (0.77824031E-01 2.000000 ResultOutputSolver: Saving with prefix: case ResultOutputSolver: Creating list for a saving - if not present CreateListPorSaving: Field Vanables for Saving ResultOutputSolver: Saving in unstructured VTK KWL (J.vtu) format	) :: heat equation
HeatSolve: Relative Change: 3.660071011800652F-013 ComputeChange: SS (ITER-1) (NRM.RELC): (0.77824031E-01 2.000000 ResultOdputSolver: Saving with prefix: case ResultOdputSolver: Creating list for saving - if not present CreatListForStaving: Field VariaBets for Saving ResultOdputSolver: Saving in unstructured VTX XML (shuf) format Vb0.dputSolver: Saving results in VTX XML (shuf) with prefix: case	) :: heat equation
HeatSolve: Relative Change: 3.660071011800652F-013 ComputeChange: S0 (TRE-1) (NRR.REL): (0.77824031E-01 2.00000) ResultOupuSolver: Saving with prefix: case ResultOupuSolver: Saving with prefix: case ResultOupuSolver: Saving in unstructured VTX XML (vhu) format VooLopuSolver: Saving results in VTX XML format with prefix: case VtooLopuSolver: Saving results in VTX XML format with prefix: case VtooLopuSolver: Saving results in VTX XML format with prefix: case VtooLopuSolver: Saving results in VTX XML format with prefix: case	) :: heat equation
HeatSolve: Relative Change : 3.6680970111809652-013 ComputeChange SS (ITER=1) (NRM.RELC): (0.77824031E-01 2.000000 ResultDutputSolver:	) ) :: heat equation
HeatSolve: Relative Change: 3.660071011800652F-013 ComputeChange: S0 (TRE-1) (NRR.REL): (0.77824031E-01 2.00000) ResultOupuSolver: Saving with prefix: case ResultOupuSolver: Saving with prefix: case ResultOupuSolver: Saving in unstructured VTX XML (vhu) format VooLopuSolver: Saving results in VTX XML format with prefix: case VtooLopuSolver: Saving results in VTX XML format with prefix: case VtooLopuSolver: Saving results in VTX XML format with prefix: case VtooLopuSolver: Saving results in VTX XML format with prefix: case	) ) :: heat equation
HeatSolve: Relative Change : 3.6680370118096526-013 ComputeChange SS (ITER=1) (INRN.RELC): ( 0.778240316-01 2.000000 ResultDutputSolver: Solving with prefix case ResultDutputSolver: Saving neutronic for Saving ResultDutputSolver: Saving neutronic for Saving ResultDutputSolver: Saving neutronic for Saving ResultDutputSolver: Saving neutronic for Saving ResultDutputSolver: Saving neutronic for partitions: 1 ResultDutputSolver: Saving neutronic for partitions: 1 ResultDutputSolver: Resign number of partitions: 1 ResultDutputSolver: Resign number of partitions: 1 ResultDutputSolver: Resolver: ALL DONE *** ElimerSolver: The end SOVER TOTAL INDECOMPACIL: 0.13 0.47	0 ) :: heat equation
HeatSolve: Relative Change: 3.666037011806652F-013 ComputeChange: S0 (TRE-1) (NRM.RELC): (0.77824031E-01 2.00000) ResultDQUpUSOlver: Saving with prefix: case ResultDQUpUSOlver: Saving in unstructured VTX XML (styl) format VpuOpUpUSOlver: Saving routis in VTX XML format with prefix: case ResultDQUpUSOlver: Saving routis in VTX XML format with prefix: case VpuOpUpUSOlver: Saving routis in VTX XML format: 1 ResultDQUpUSOlver: Saving routis in VTX XML format: TelensOlver: *** Elimer Solver: ALL DOVE *** ElimerSolver: *** elimer Solver: ALL DOVE *** ElimerSolver: ***	) ) : heat equation

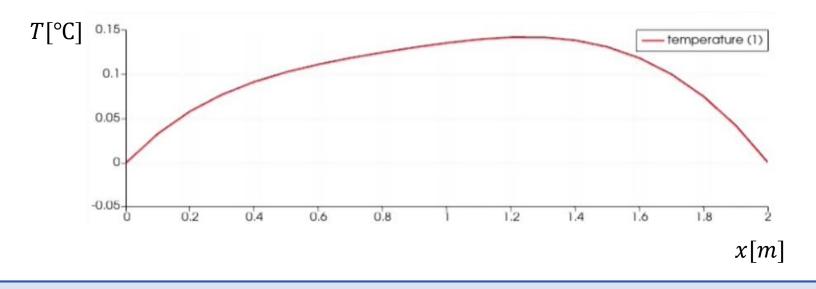






- 6. Post-processing
  - Open the result in ParaView
  - What is the value of maximum temperature
  - Plot temperature as a function of y-coordinate at x = 0.5m
    - Go to Filters->Alphabetical->Plot On Intersection Curves
    - In  ${\tt Properties},$  specify the intersection plane and then click  ${\tt Apply}$



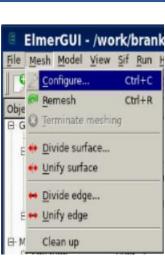


#### Exercise

Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

- 6. Post-processing
  - Examination of computational time vs. number of mesh nodes
  - Check output of Solver Log and read second line from bottom to get the CPU time (see below)

- Let's increase the mesh density (i.e. increase the number of nodes and elements) and rerun the case
- Navigate to Mesh->Configure...
- In elmergrid->String:, change -autoclean -relh 1.0 to -autoclean -relh 0.6
- Click Apply
- Click Mesh->Remesh to recompute mesh



<b>c</b> tetlib	(.stl .smesh .poly .off .ply .mesh)	
nglib	(.in2d .stl .brep .step .stp .iges .igs)	
C elme	rgrid (.grd .FDNET .msh .mphtxt .unv	1)
tetlib		
String:	nnjApVq1.414	
nglib		
Max H:	1000000	_
Fineness	: 0.5	
Bgmesh	: [	
elmergr	id	
String:	-autoclean -relh 0.2	
Element	t codes (for solver):	
Codes:		
		_

#### Exercise

- Run the case with denser mesh and observe the time difference
- Try to increase the density even more, for example change the value of 0.6 to 0.1
- Compare CPU times vs. number of mesh nodes



- elmerf ... Command to compile user defined Fortran routines
- ElmerGrid ... Command to generate and convert mesh data
- ElmerGUI ... Command to run the user interface
- ElmerSolver ... Command that reads SIF file and mesh data and performs FEM calculations
- ElmerSolver\_mpi ... Command that performs parallel FEM calculations



#### elmerf

- Users can define their own functions in Fortran that are then passed to ElmerSolver
- User defined functions are defined in files with  $\mbox{.}\pm90$  extension
- In SIF file, the reference to this file can be defined in boundary condition block, material block,...
- ElmerSolver then calls this function by reading reference in SIF file



- Let's examine the zero temperature boundary condition in Heat transfer case
- In SIF file, the boundary condition block is defined as:

```
Boundary Condition 1
Target Boundaries(1) = 1
Name = "zero_temp"
Temperature = 0
End
```

- Here, the temperature of the whole target boundary is set to 0.
- To call an external function, delete the Temperature = 0 line and add these two lines
   Temperature = Variable Coordinate 2
   Real Procedure "define temp" "defineTemp"
- The first line means that temperature will be a variable of y-coordinate
- The second line means that we are calling a function "defineTemp" that is located in define\_temp.f90 file.



• File define temp.f90 should look like this:

```
FUNCTION defineTemp(Model, elmer_node, t, y) RESULT(elmer_temperature)
USE DefUtils
TYPE(Model_t) :: Model
TYPE(Nodes_t) :: Nodes, EdgeNodes
INTEGER :: elmer_node, i
REAL(KIND=dp) :: t, y
.
```

. \$ Do your own calculations

```
END FUNCTION defineTemp
```



The name of the function is defineTemp

- Input parameters are:
  - Model -> structure of ElmerSolver, it contains different parameters that we can access during function execution
  - elmer\_node -> ID of the node that is currently being processed
  - t-> current timestep
  - y-> variable specified in SIF file, in our case y-coordinate
- Output parameter is temperature, as defined in SIF file
- Inside this function we can now write different routines and specify our own boundary condition



#### Thank you for your attention!

http://sctrain.eu/





This project has been funded with support from the European Commission.

This publication [communication] reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.