

Guided examples using ELMER FEM

Presenter: Matic, Brank, Faculty of Mechanical Engineering, UL

Date: 12-03-2021



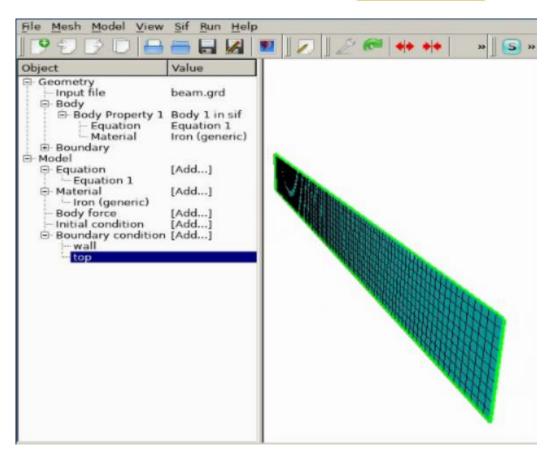


SLING

• 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>



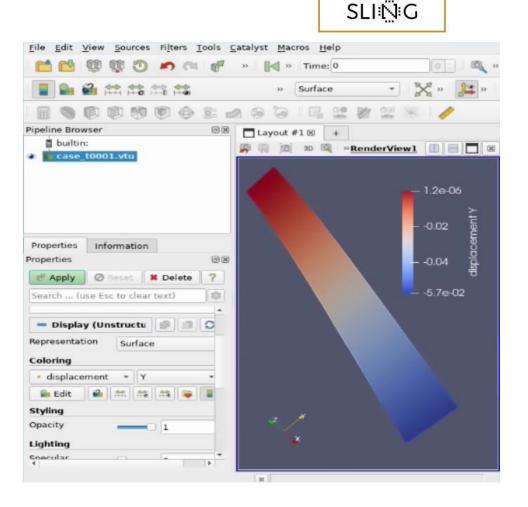
Quick software overview



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

• On Viz

- Connect to Viz using NoMachine
- · Click bottom left icon on desktop to open the start menu (See figure)
- Open terminal

_		SLI⊠G
	Najbolj uporabljani programi	
	Terminal	
	Web Browser (Firefox)	
1	Osebne datoteke (Domov)	
	🕈 Text Editor (Emacs)	
Terminal	Terminal Program Quake-Style (Yakuake)	
	Vsi programi	
u 🗾	Press '/' to search	
	🖁 Grafika 🔸	
	g Igre •	
	Internet +	
	Nastavitve ·	
1	Pisarna +	
4	Potrebščine	
	💈 Razvoj 🔹 🔸	
9	🕈 Sistem 🔸	
é	Večpredstavnost	
	Wolfram Mathematica	
	Zabava in učenje	
	Znanost in matematika	
	Izgubljeno in najdeno	
	🛛 Najdi datoteke in mape	
tent	Osebne datoteke (Domov)	
	🦻 Pomoč	
Wird	Dejanja	
E C	Nadzorno središče	
kto	🛛 Sistem 🔸	F ' C ' '
Start menu	ን Poženi ukaz	Fig: Start me
jĘi /	🛾 Zakleni sejo	tripity dealer
) Odjava	trinity deskt
8	🗋 🥥 🚺 2 3 4 E	

Start menu of ity desktop.

C

EURO

Software loading and running

C

EURO

SLI₩G

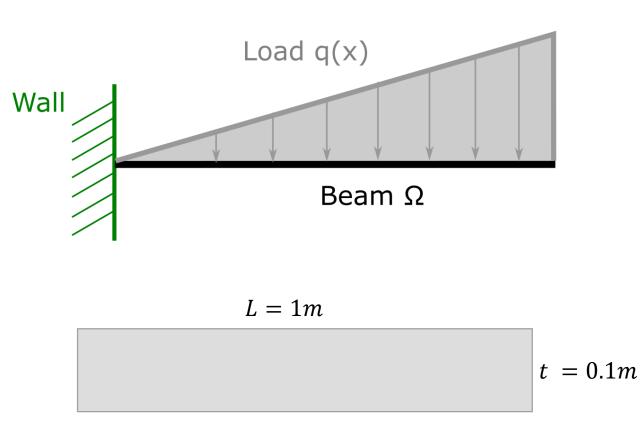
• To open Elmer GUI:

- 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/5.6.2-foss-2020b-mpi
- To run ParaView
 - paraview

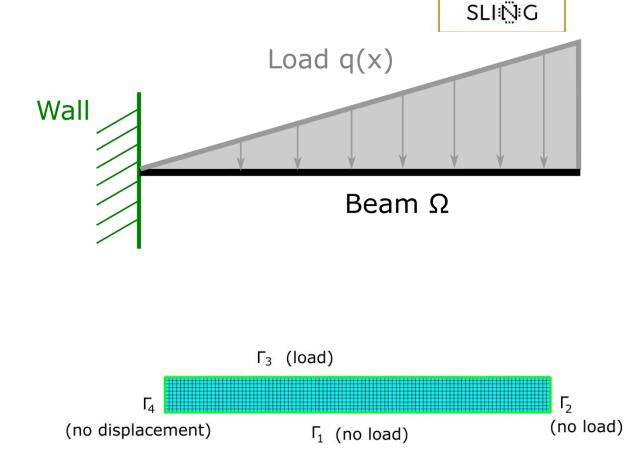
- Case definition
 - 2D case
 - Homogeneous, elastic beam, defined on x - y plane
 - (length 1m and thickness 0.1m)
 - Rigid support at the wall
 - Space dependent mechanical load, which grows linearly from 0 to $q_0 = 1e7N$
- Goal
 - Obtain the displacement of the beam





- Mathematical definition of the problem
 - Domain of computation:
 - Beam area Ω
 - Boundary conditions
 - Mechanical load Γ_3
 - No displacement at Γ_4
 - Zero load on $\Gamma_{\! 1}$ and $\Gamma_{\! 2}$
- Problem solution

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





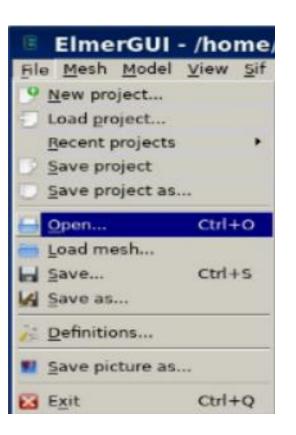
Case preparation - overview

- 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)



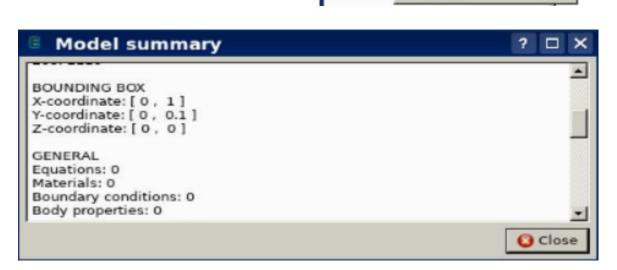
- 1. Definition of domain of computation (Elmer GUI)
 - Import mesh into Elmer GUI
 - File->Open
 - Navigate to your folder and select beam.grd
 - Use Mouse wheel/left button to rotate and zoom mesh

ElmerGUI		_ 🗆 ×
File Mesh Model View Sif Run	<u>+</u> elp 6] ■] / _/ @ @ ++ ++ ++ ▼, □,] ⊆ @ }	
Object Value Geometry Input file Body Boundary Model Equation Material Add] Body force Add] Body force Add] Boundary condition Add] Boundary condition Add] Boundary condition		



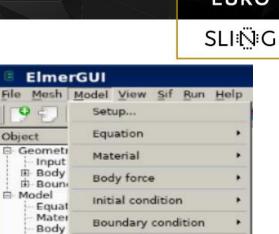


- Go to Model->summary to observe mesh information
- Check that mesh contains 3221 nodes
- How many surface elements are in the mesh?
- What is the type of the surface elements?
 - Triangles, quadrangles?
- Check Elmer documentation to see ID numbers for different elements



Initial

Bound



Set body properties

Summary... Clear all

Set boundary properties





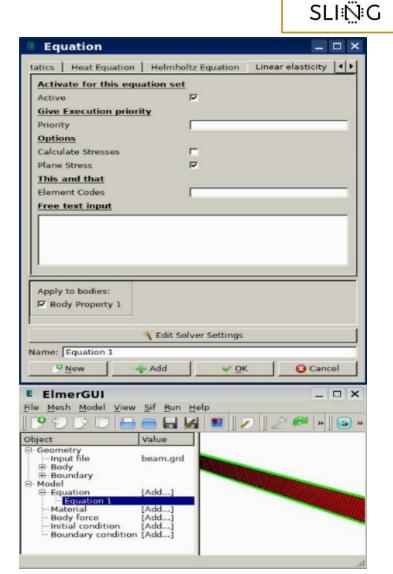
- 2. Definition of type of physical problem
 - In Model->Setup... one can define properties related to the simulation, such as results directory, input mesh directory, name of output file, constants, etc...
 - Make sure the simulation type is set to Steady state

🗉 Elme	rGUI		
File Mesh	Model View Sif Run He	lp	
1991	Setup		
Object	Equation		
Geometri Input Body Bound Model Equat Body Initial Bound	Material		
	Body force		
	Initial condition		
	Set boundary properties		
	Summary		
	Clear all		

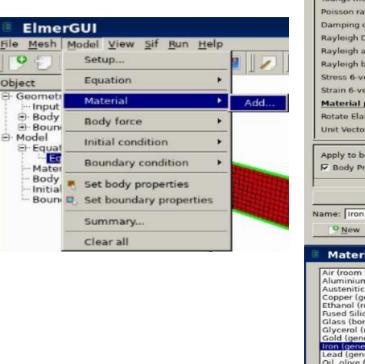
			? 🗆 :
Header Check keywords w MeshDB Include path Results directory	varn		
Coordinate system Coordinate mapping Simulation type Output intervals Coordinate Scaling	5 Cartesian 1 2 3 Steady state 1 1 case.sif	Steady state max. iter Timestepping method BDF order Timestep intervals Timestep sizes Angular Frequency Post file	
	0 -1 0 9.82 5.67e-08 8.8542e-12	Boltzmann 1.380 Unit charge 1.602	

EURO

- 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



Material	_ 🗆 ×
at Equation Helmholtz Equa	tion Linear elasticity
Properties	-
Youngs modulus	193.053e9
Poisson ratio	0.29
Damping coefficient	
Rayleigh Damping	Г
Rayleigh alpha	
Rayleigh beta	
Stress 6-vector	
Strain 6-vector	
Material pricipal directions	
Rotate Elasticity Tensor	-
Unit Vector 1	
Apply to bodies:	al library
1 Materia	al library
ame: Iron (generic)	
ame: Iron (generic)	
ame: Iron (generic)	✓ <u>O</u> K Orancel
	♥ QK Cancel
<u>♀</u> New → Add	? □ ×



SLING

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
Ele Mesh Model View Sif Run Help Object Setup Equation Image: Setup Image: Setup Object Equation Image: Setup Image: Setup Image: Setup Image: Setup Body force Image: Setup Image: Setup Image: Setup Image: Setup Body force Image: Setup Image: Setup Image: Setup Image: Setup Boundary condition Add Add Image: Set body properties Set boundary properties Set boundary properties Image: Set boundary properties Summary Summary	General Electrostatics Heat Equation Helmholtz Equation Linear elasticity Mesh Update Nz Image: Nicholar System Normal-Tangential Coordinate System	File Mesh Model View Sif Run Help Object Valu Object
Clear all	Traction boundary conditions Normal Force Apply to boundaries: Ø Boundary 1; Boundary 3 Boundary 4 Name: BoundaryCondition 2 Ø New Ø New	



- 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

EURO

- 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 ${\tt Load}$

- 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

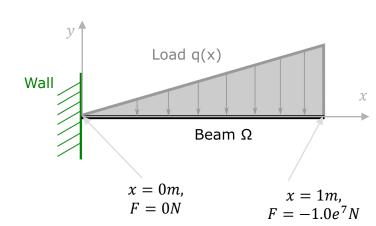
Real;

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





• In case of elasticity solver, the Mesh boundaries are automatically set to 0 load

C

EURO

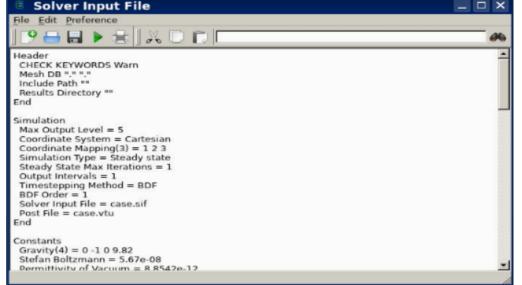
SLIℕG

- 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







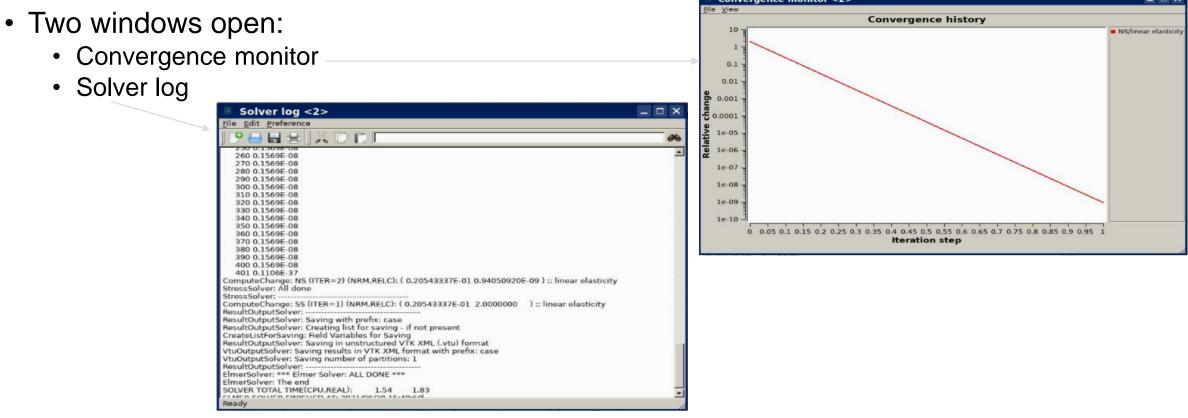


Running the case

- 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.

EURO

SLI:NⁱG

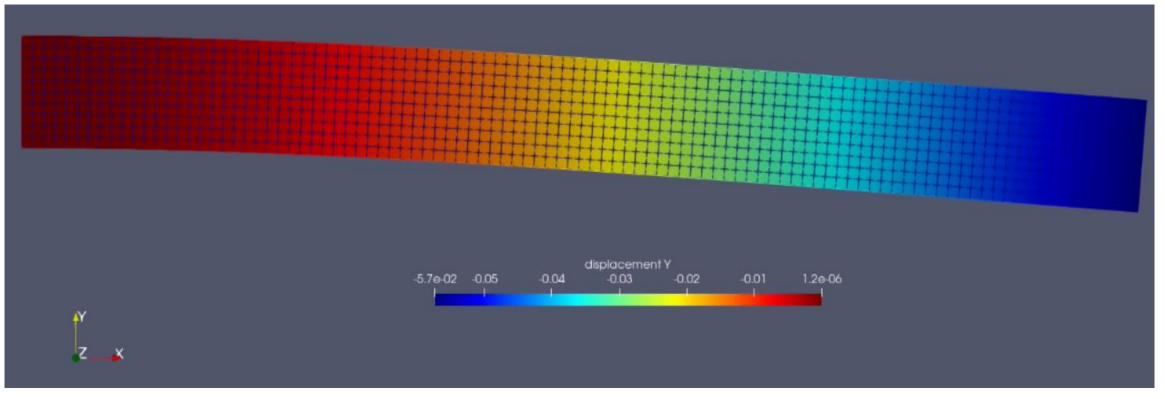


Post-processing



SLIŅG

• Use mouse wheel/right button to rotate and zoom result



Post-processing



SLI₩G

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

Properties Info	rmation		
Properties			R
d Apply	Ø Reset	🞽 Delete	?
Search (use Es	: to clear te	×t)	1
- Properties (case_t000	1.vtu)	
✓ Cell/Point Arra	y Status		0
✓ ∯ Geometryl ✓ 📬 displaceme			
— Display (Un	structured	GridReprese	III O 🖬
Representation	Surface	Magnitude	
Coloring		x	
 displacement 	-	Y	
🔒 Edit		z	

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

U.	Find I	Data					? □ >
Cr	eate Sel	ection					
Fir	nd 🧧 Poi	nt(s) =	from case	_t0001	.vtu		*
0	lisplacem	ent (1)	-	is m	in	*	+
4							
				Run	Selection Query		
cı	irrent Se	lection	n (case_t00	01.vt	u : 0)		
sh	ow: P	oint(s)				* 🗆 In	vert selection
	Point ID		Points		Points_Magnitude		displacem
0	3220	1	0.0425808	0	1.00479	0.00388738	-0.0574192
Ea	dection I		/ Properties				
30	nection i	Jispiaj	Properties	2			
0	Selection	n Color	#	Cell La	ibels *	Point Labels	• @ @
-	Freeze S	alactia	e Eutr	act Sel	action Dist Color	tion Over Time	Close
	menze 3	electio	exu	0.07 261	errion blot selec	uon over time	~ C1050



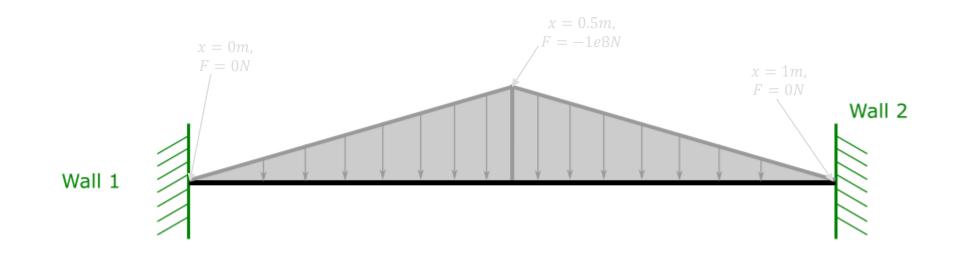
- Change maximum load on the right edge (at x = 1m) to $F = -1.0e^8N$ and run the case again.
- Change the load again to $F = -1.0e^9N$. Run the case and observe the maximum displacement with increase of load







- 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

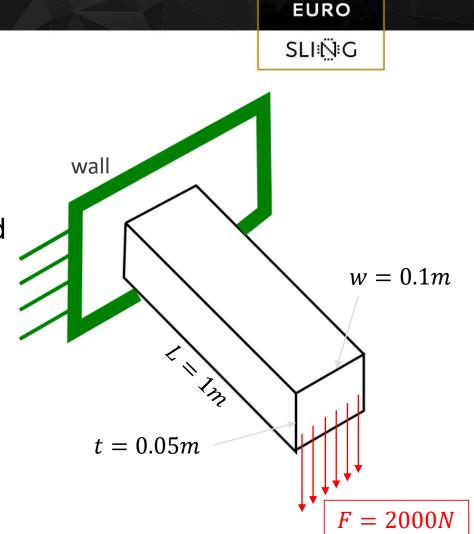
- Change Simulation Type to Transient (Setup->simulation type)
- Set the Time Stepping Method to bdf
- Set the Time step intervals to $200\,$
- Set the Time Step Sizes to $1e^{-4}$

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 $550 kg/m^2$

Goal

- Calculate displacement field of the beam
- Obtain the location and value of the maximum displacement
- Calculate stress field



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

R

EURO



SLI⊧Ņ⊧G

al Steady state Nonlinear system Linear system 📢

🖌 Apply

near elasticity

not converge

- 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

Equation	_ _ ×	Solver control for
Equation Helmholtz Equation Activate for this equa Active Give Execution priorit Priority Options Calculate Stresses	Linear elasticity	Olver control for Olver specific options Gen Method Olirect Banded Olirect Banded Olirect GCR Oli
Plane Stress <u>This and that</u> Element Codes <u>Free text input</u>		Max. iterations 500 Convergence tol. 1.0e-10 Preconditioning ILU1 ILUT tolerance 1.0e-3 Residual output 10 Prec. recompute 1 BiCGStabl order 2
Apply to bodies: F Body Property 1 Edit Solve Name: Equation 3 Name: Add	er Settings	C Abort if the solution die

Edit Solver Settings

R

EURO

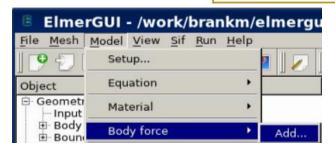
SLIЮ́G

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

- 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





BodyForce	_ 🗆 ×
holtz Equation Linear elasticity	Mesh Update
Volume forces	<u> </u>
Force 1	
Force 2	
Force 3	
Pressure	
Stress 6-vector	
Strain 6-vector	
Bodywise Dirichlet Conditions	
Displacement 1	
Displacement 2	
Displacement 3	
Displacement 1 Condition	
Disslament 2 Condition	<u> </u>
Apply to bodies:	
F Body Property 1	
Name: BodyForce 2	
🤒 <u>N</u> ew 🚽 Add 🗸	OK Orancel

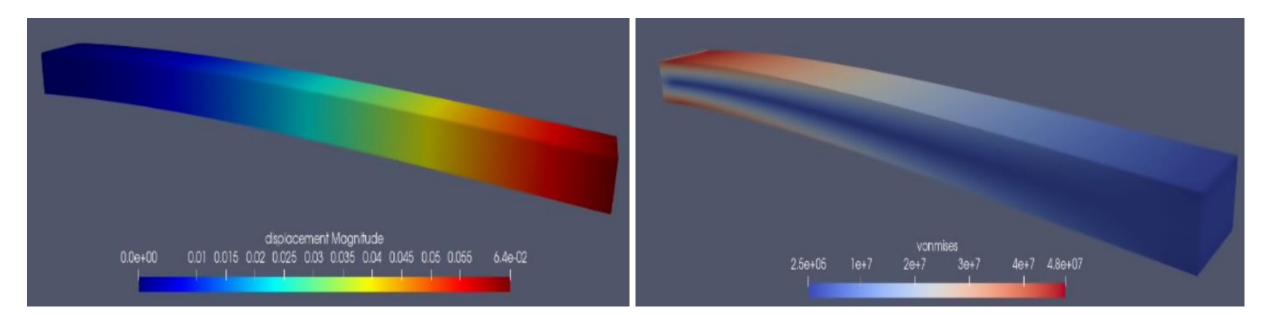
- 5. Computation
 - Generate and check SIF file
 - Save the case and run it



SLING

5. Post processing

- What is the maximum displacement and where is its location
- Plot displacements and Von mises stresses





SLIЮ́G



Gravity in x-direction

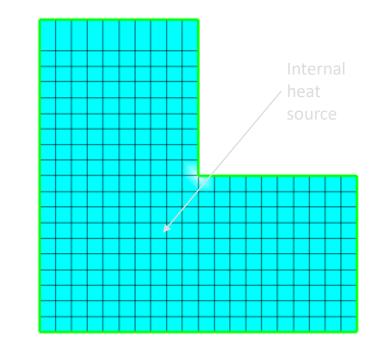
- The beam should be more rigid if the beam is orientated differently
- Change the direction of gravity and force of attached object in the negative xdirection

C

EURO

SLIЮ́G

- Case definition
 - A structure in L-shape is heated by internal source with magnitude of 1W/m³. The density of the structure is 1kg/m³ 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



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



- 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 ... heat source



SLI⊠G

SLI:N:G

- 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

Equation - × Helmholtz Equat Electrostatics Heat Equation Activate for this equation set Active **Give Execution priority** Priority Options Phase Change Model None . Convection ٠ Convection None Free text input Apply to bodies: ■ Body Property 1 K Edit Solver Settings Name: - Add V OK 🙆 Cancel New



SLIЮ́G



SLIЮ́G

3. Definition of material properties

- Create a new material and define:
 - Density,
 - Heat conductivity (in Heat)
- Assign the material to the body

Е	Mater	ial		_ 🗆 ×				
G	ieneral	Electrostatics	Heat Equation	Helmholtz 4				
н	Propertie leat Cond		None					
E	missivity		, 					
E	nthalpy pecific E							
Р		Coefficient	, ,					
Ī								
				<u> </u>				
	Body P	oodies: roperty 1						
	🌂 Material library							
Nar	me: Mat	erial 2						
	<u>P</u> ew	Add	<u> </u>	🙆 Cancel				



SLIЮ́G

4. Definition of boundary conditions

- Define zero temperature on boundaries
- Temperature is defined under Dirichlet conditions
- Apply temperature to all boundaries

BoundaryConditio	on _ 🗆 >
General Electrostatics	Heat Equation Helm
Dirichlet Conditions	<u> </u>
Temperature	
Temperature Condition	
Heat Flux conditions	
Heat Flux	
Heat Transfer Coeff.	
External Temperature	
Latent heat of phase ch	ange
Phase Change	
<u>Heat Gap</u>	
Heat Gap	
Radiation Settings	
Radiation	None 💌
Emissivity	
Radiation Boundary Open	
Apply to boundaries:	
Г Boundary 1 Г Bo	undary 2
Г Boundary 3 Г Bo	undary 4
🗖 Boundary 5 👘 Bo	undary 6
ame: BoundaryCondition 3	3
New Add	V OK Cancel

- 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

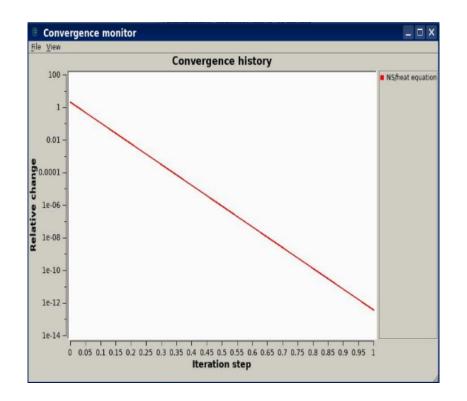
BodyForc	e	>
Electrostatics	Heat Equation	Helmholtz Equat
Volume source	es	-
Heat Source		
Friction Heat		Г
Joule Heat		Г
Bodywise Diri	ichlet Condition	<u>s</u>
Temperature		
Temperature Co	ondition	
Perfusion		
Perfusion Rate		
Free text inpu	<u>it</u>	
		<u> </u>
Apply to bodies	s:	
🗖 Body Proper	ty 1	
lame: BodyFord	ce 4	



SLI⊠G

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

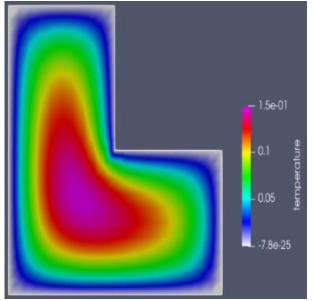
🗄 Solver log	_ 0
Ele Edit Preference	
	_
THeatSolve: Iter: 1 Solve: (s) 0.00 0.00 HeatSolve: Result Norm: 7.7824030917542059E-002 HeatSolve: Relative Change: 2.00000000000000 HeatSolve: Relative Change: 2.00000000000000 HeatSolve: HeatSolve: HeatSolve: TEMPERATURE ITERATION 2 HeatSolve: Starting Assembly HeatSolve: Starting Assembly HeatSolve: Starting Assembly HeatSolve: Starting Assembly HeatSolve: Starting Assembly HeatSolve: Assembly: HeatSolve: Starting Assembly HeatSolve: Temperature (s) 0.00 0.01 HeatSolve: Temperature (s) 0.00 0.01 HeatSolve: Temperature (s) 0.00 0.01 HeatSolve: Temperature (s) 0.00 0.01 HeatSolve: Result Norm: 7.78240315750605E-002 HeatSolve: Relative Change: 3.6680970111809652E-013 ComputeChange: SS (ITER=1) (NRM.RELC): (0.77824031E-01 0.3000000)):: heat equation ResultOutputSolver: Temperature (s) 0.00 0.00 HeatSolve: Relative Change: 3.6680970111809652E-013 ComputeChange: SS (ITER=1) (NRM.RELC): (0.77824031E-01 2.0000000)):: heat equation ResultOutputSolver: Saving with prefix: case ResultOutputSolver: Saving results in VTK XML (xtu) format VtuOutputSolver: Saving results in VTK XML (xtu) format VtuOutputSolver: Saving nunstructured VTK XML (xtu) format VtuOutputSolver: The end SolVER TOTAL TIME(CPU.REAL): 0.13 0.47 ELMER SOLVER FINISHED AT: 2021/06/29 12:37:01	

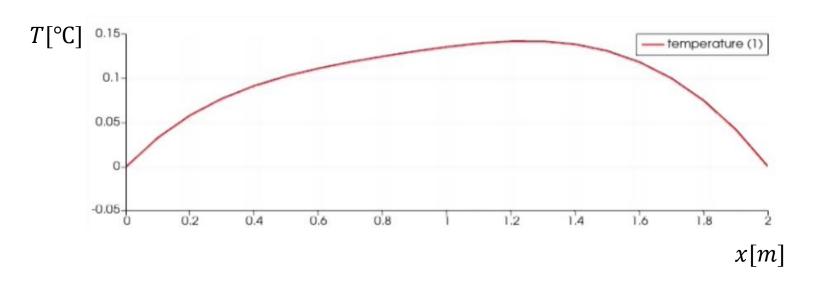






- 6. Post-processing
 - Open the result in ParaView
 - What is the value of maximum temperature
 - Plot temperature as a function of y-coordinate at x = 0.5m
 - Go to Filters->Alphabetical->Plot On Intersection Curves
 - In Properties, specify the intersection plane and then click Apply





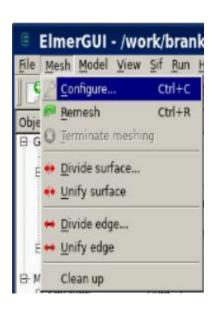




Exercise

- 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



Preferre	d generator	
r tetlib	(.stl .smesh .poly .off	.ply .mesh)
🕫 nglib	(.in2d .stl .brep .step	.stp .iges .igs)
C elme	rgrid (.grd .FDNET .ms	h .mphtxt .unv)
tetlib		
String:	nnjApVq1.414	
nglib —		
Max H:	1000000	
Fineness	: 0.5	
Bgmesh	:	
elmergri	id	
String:	-autoclean -relh 0.2	
Element	codes (for solver);—	
Codes:		



SLIŅG



- 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

C

EURO

SLIЮ́G

• Compare CPU times vs. number of mesh nodes

Overview of commands





- 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

Overview of commands



elmerf

- Users can define their own functions in Fortran that are then passed to ElmerSolver
- User defined functions are defined in files with $. \pm 90$ 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"

EURO

SLIℕG

- 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



SLI₩G



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 you attention!



This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 951732. The JU receives support from the European Union's Horizon 2020 research and innovation programme and Germany, Bulgaria, Austria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, United Kingdom, France, Netherlands, Belgium, Luxembourg, Slovakia, Norway, Switzerland, Turkey, Republic of North Macedonia, Iceland, Montenegro

