

Elmer FEM – Parallelisation, nonlinear and time dependent problems

dr. Borut Černe

University of Ljubljana, Faculty of Mechanical Engineering





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.



Step 1 – Open ElmerFEM - New project – Choose directory and analysis type

module load elmer/foss-2018b

➢ ElmerGUI

E New project				?	\times
Project directory Select project dir Geometry input C Elmer mesh C Geometry file	Select mesh dir Select geometry file				
Equation definition fi <u>Default EDFs</u> electrostatics.xml heatequation.xml linearelasticity.xml meshdeform.xml navier-stokes.xml resultoutput.xml	es (EDFs) - you can anytime add extra EDFs from F <u>Extra EFDs to be added</u> nonlinearelasticity.xml	<< Add Remove >>	advection-diffusion.xml divergencesolver.xml elasticplate.xml fluxsolver.xml freesurface.xml k-epsilon.xml magnetodynamics.xml magnetodynamics2d.xml model-pde.xml poissonboltzmann.xml reynolds.xml saveline.xml savescalars.xml savescalars.xml		
	ОК	Cance	2		





Step 2 – Load Geometry

Open > select *u_turn.grd* file

ElmerGUI - E:/Projekti/20	021_Erasmus_plus_S	Ctrain/4_C	01/1_Cases/1	1_ElmerFE	M_Nonlin	ar_Cern	e/1_Nor	nlinear	_elasticity/	/test			
le <u>M</u> esh <u>M</u> odel <u>V</u> iew	<u>S</u> if <u>R</u> un <u>H</u> elp												
9980	🗎 🔚 📕] 🖉] .	م	+ +	•	**	5	┗	S P	۱		
bject	Value												
Geometry													
The Body	u_turn.grd												
Model													
Equation	[Add]												
Body force	[Add]			1777 7 7	\$ \$\$1								
Initial condition	[Add]		W.	T+++	+++1								
Boundary condition	[Add]		1H	11H	+++								
			l I	HHT									
			ų į	1++++	++++					6			
			1	HITH	+++1								
			١		HTH	植						TH	
				11111	++++	1				K		THI	
				HITH								++ +10	
				11H	TTH	13				1	\mathbb{R}	+++1	
				HTT!		HA.					$\mathbb{H}^{\pm\pm}$	HTH!	
				HT1	HH	PHA.				, N		TH	
				MH+	HH	FH						+++11	
				TT -	HH	777	2			l III	HTT++	H	
				A.	Htt	XX	B					TH	
					HTH	XXX				- A			
						\times	\times	×			BATH	HU	
						\propto	XX	Æ	2999	aa axxx	$\lambda + \Xi \pm$	TH	
						$\sim \sim$	XH	LF A	444	HXXX	AH	H	
						A.	H	£FF-	HHH	∞	XH		
						A.		£#		54XX	H	F	
							A.C.	Ħ	###	200	H		
									att ?	22			



Step 3 – Model setup

- Scale geometry to correct unit system
- Define time-dependent type analysis

Setup				?
Header				
Check keywords	warn			
MeshDB .				
Include path				
Results directory				
Free text				
Simulation				
Max. output level	5	Steady state max. iter	1	
Coordinate system	Cartesian 💌	Timestepping method	BDF	•
Coordinate mapping	123	BDF order	1	•
Simulation type	Transient 💌	Timestep intervals	20	
Output intervals	1	Timestep sizes	0.05	
Coordinate Scaling	0.01	Angular Frequency		
Solver input file	case.sif	Post file	case.vtu	
Free text				_
Constants	,			
Gravity	0 -1 0 9.82	Boltzmann 1.38	07e-23	
Stefan Boltzmann	5.67e-08	Unit charge 1.60	2e-19	
Vacuum permittivity	8.8542e-12			
Free text				
				Apply



Step 4 – Select model

- *Equation* > Nonlinear elasticity
- Edit Solver Settings

E Equation X	E Solver control for Nonlinear elasticity	? X
Navier-Stokes Result Output Nonlinear elasticity	Solver specific options General Steady state Nonlinear system Linear system Parallel	Adaptive 🔳
Activate for this equation set	Procedure "ElasticSolver"	
Active 🔽	Variable Displacement(cdim)	
Give Execution priority	Additional Variables	
Priority	Exported Variable 1	
Options	Miscellaneous options	
Plane Stress	Calculate Stresses 🔽	
This and that	Calculate Strains	
	Calculate Principal 🔽	
	Calculate Loads	
	Elasticity Solver Linear	
	Displace mesh 🔽	
	Free text input	
<u> </u>		
Apply to bodies:		
Body Property 1		
🜱 Edit Solver Settings		
Name: Equation 1		
	,	
<u></u> Vpdate <u></u> <u></u> Remove		V Apply

Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

Step 5 – material

 Material > Material library > Austenitic stainless steel

Material library	r	>
Air (room temperature)		•
Aluminium (generic)		
Austenitic stainless steel (AK Steel 201)		
Copper (generic)		
Ethanol (room temperature)		
Fused Silica (25 C)		
Glass (borosilicate)		
Glycerol (room temperature)		
Gold (generic)		
Iron (generic)		
Lead (generic)		
Oil, olive (25 C)		
Platinum (generic)		
Polycarbonate (generic)		
Polyvinyl chloride (generic)		
Silicon (liquid)		
Silicon (solid)		
Silver (generic)		-
- 17 ⁻		
Amount A class A		ol.
	lose	UK



Step 6 – Boundary conditions

• Boundary condition > Left

E BoundaryCondition	- 🗆 X
oltz Equation Linear elasticity Mesh Up	date Navier-Stokes Nonlinear elasticity
Normal-Tangential Coordinate System	<u>.</u>
Use normal-tangential coordinate system	
Change of variables	
Dirichlet Conditions	
Displacement 1	Variable "time"; Real MATC "0.006*tx"
Displacement 2	0
Displacement 3	0
Displacement 1 Condition	
Displacement 2 Condition	
Displacement 3 Condition	
Traction boundary conditions	
Normal Force	
<u></u>	
Apply to boundaries:	<u> </u>
Boundary 1 Boundary 2	
I ■ Boundary 3	
Name: Left	
P New ✔ Update	V OK Remove

• Boundary condition > Right

BoundaryCondition	>	<
uation Linear elasticity Mesh Update Navier-Sto	okes Nonlinear elasticity	Þ
Normal-Tangential Coordinate System	<u>-</u>	
Use normal-tangential coordinate system		
Change of variables		
Dirichlet Conditions		
Displacement 1 Variable "tim	ne"; Real MATC "-0.006*tx"	
Displacement 2		
Displacement 3		
Displacement 1 Condition		
Displacement 2 Condition		
Displacement 3 Condition		
Traction boundary conditions		
Normal Force		
		4
Apply to boundaries:	-	
Roundary 1 Roundary 2		
F Boundary 3		•
ame: Right		
🔍 New 🗸 Update	OK - Remove	1



Step 7 – Run analysis

• Generate .sif file

E Solver Input File	_	×
File Edit Preference		
🖻 🖶 🖶 🚔 🗶 🖸	D	<i>8</i>
Header CHECK KEYWORDS Warn Mesh DB "." "." Include Path "" Results Directory "" End		-
Simulation Max Output Level = 5 Coordinate Mapping(3) = 1 2 3 Simulation Type = Transient Steady State Max Iterations = 1 Output Intervals = 1 Timestepping Method = BDF BDF Order = 1 Timestep Nites = 20 Timestep Sizes = 0.05 Coordinate Scaling = 0.01 Solver Input File = case.sif Post File = case.vtu End		
Constants Gravity(4) = 0 -1 0 9.82 Stefan Boltzmann = 5.67e-08 Permittivity of Vacuum = 8.8542e-12 Boltzmann Constant = 1.3807e-23 Unit Charge = 1.602e-19 End		
Body 1 Target Bodies(1) = 1 Name = "Body 1" Equation = 1 Material = 1 End		

• Use batch file to run the cases

CaseRu	n_Elmer.txt 🗵
1	<mark>¤</mark> #!/bin/bash
2	#SBATCHexport=ALL,LD_PRELOAD
3	#SBATCHpartition=westmere
4	#SBATCH -n 2
5	#SBATCH -o output.log
6	
7	module purge
8	module load elmer/foo-2018b
9	
10	N=\$SLURM_NTASKS
11	
12	dir=\$ (pwd)
13	
14	<pre>echo case.sif > ELMERSOLVER STARTINFO</pre>
15	cat ELMERSOLVER STARTINFO
16	ElmerGrid 2 2 <mark>\$dir</mark> -metis \$N 3
17	srun -n <mark>\$N</mark> -o output <mark>\$N</mark> .log ElmerSolver mpi
18	

Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

Step 8 – Postprocessing -Paraview

module load ParaView/5.8.1-foss-2020b-mpi

➤ paraview



Case 2: Beam twisting

Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

- Use similar procedure as in the previous example
- Analysis is again time-dependent: use 10 x 0.1 s time steps
- Mesh file: beam3d.grd
- Use nonlinear elasticity model
- The material in this case is Polycarbonate
- One side is fixed





Case 2: Beam twisting

 $u_x = (\cos(\phi) - 1)x - \sin(\phi)y)$

 $u_y = (\cos(\phi) - 1)y + \sin(\phi)x)$

Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

- On the other side apply rotational (twist) boundary condition
- Use set of equations below
- Use variables tx(0) for time and tx(1), tx(2) for coordinate displacements

euy	
out file	
dy	
undary	
uation	[Add
Equation 1	
terial	[Add
Polycarbonate (generic)	
dy force	[Add
tial condition	[Add
undary condition	[Add
Fixed	
Twist	



Normal-Tangential Coordinate System	n	-
Use normal-tangential coordinate system		
Change of variables		
Dirichlet Conditions		
Displacement 1	Variable "time, Coordinate"; Real MATC "	
Displacement 2	Variable "time, Coordinate"; Real MATC " Variable "time,	
Displacement 3	0	
Displacement 1 Condition		
Displacement 2 Condition		
Displacement 3 Condition		
Traction boundary conditions		
Normal Force		- -
Apply to be indexion		
Boundary 1 Boundary 2		
Boundary 3 Boundary 4		•

Case 2: Beam twisting

Sctrain SUPERCOMPUTING KNOWLEDGE PARTNERSHIP

- Perform analysis using .sif input file and CRun_Elmer.txt
- Visualize time-dependent results in ParaView





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.