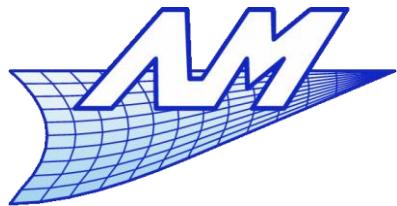
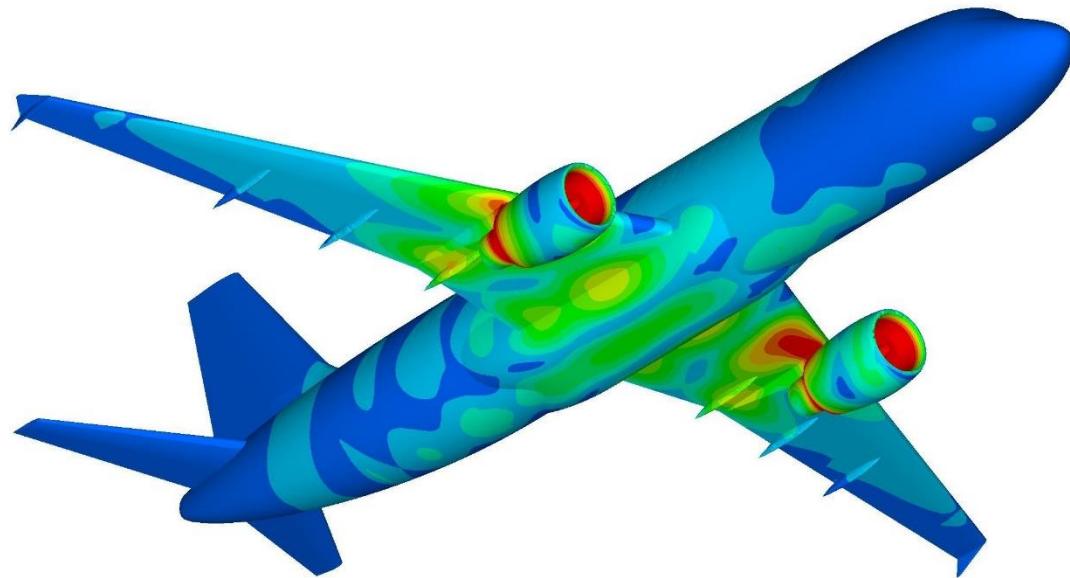


MECA-0036 Finite Element Method



Project 2022



LARUELLE Cédric: cedric.laruelle@uliege.be

VOLVERT Martin: m.volvert@uliege.be

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

Presentation Plan

- 1. Installation of the softwares**
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

1. Installation of the software

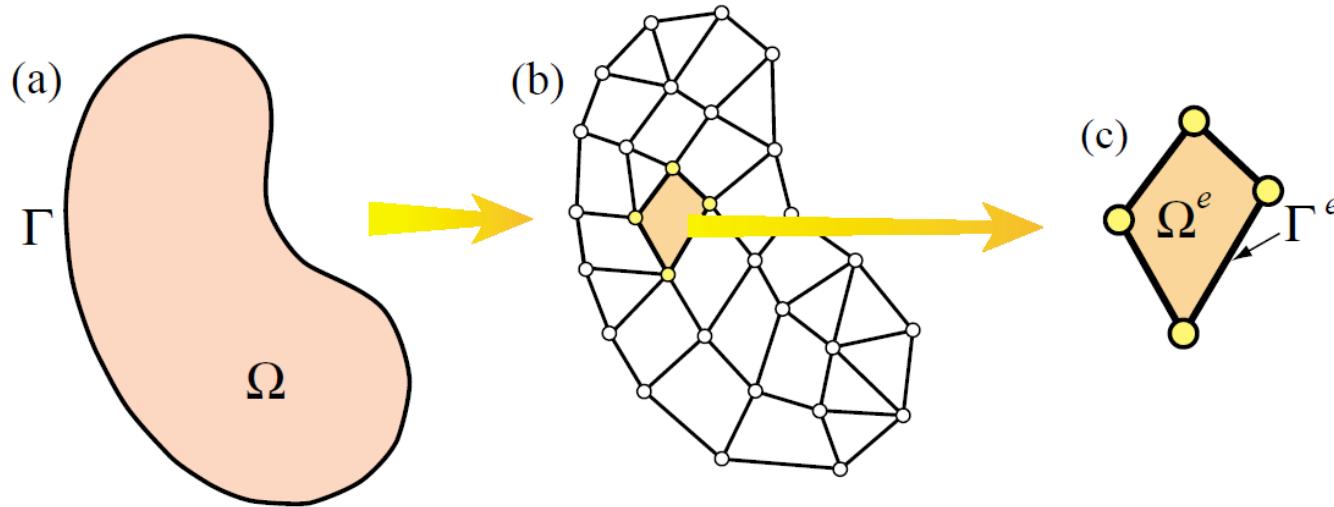
- Install Siemens NX 1859, follow the instructions provided online:
http://pegase.ltas.ulg.ac.be/doku.php?id=siemensnx_en
- Do not install RLM if asked during the installation!!
- If you are not using the ULiège Wifi Secure, use the ULiège Network with VPN:
https://my.segi.uliege.be/cms/c_11650735/en/mysegi-vpn-f5-big-ip

Presentation Plan

1. Installation of the softwares
2. **Quick introduction to FEM**
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

2. Quick introduction to FEM

- Decompose the structure/solid into small pieces => finite elements of very simple and known geometry:



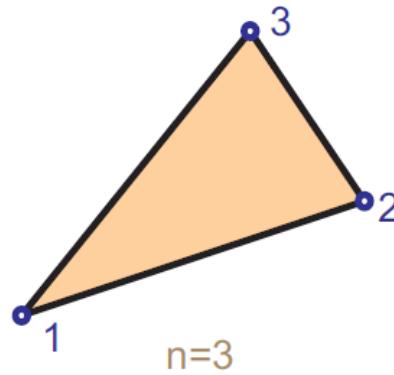
- Volume/Surface approximated.
- Mass/Total Potential Energy at equilibrium can change with discretization changes even with the Boundary Conditions unchanged!



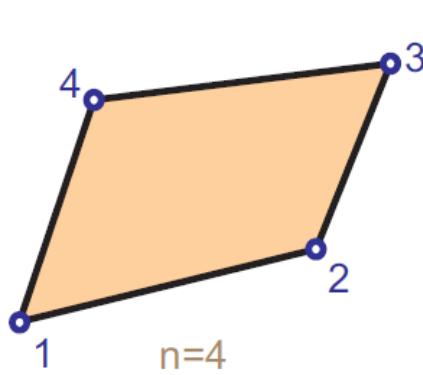
Using isoparametric elements renders possible the accurate description of curved surfaces!

2. Quick introduction to FEM

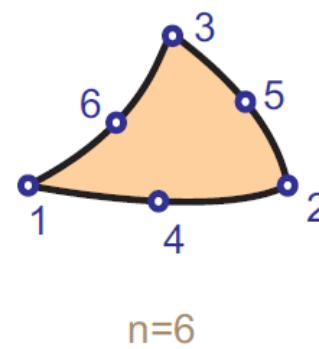
Element geometry is defined by node locations:



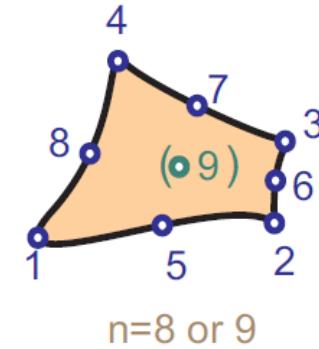
T3



Q4



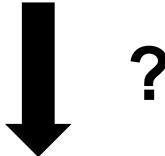
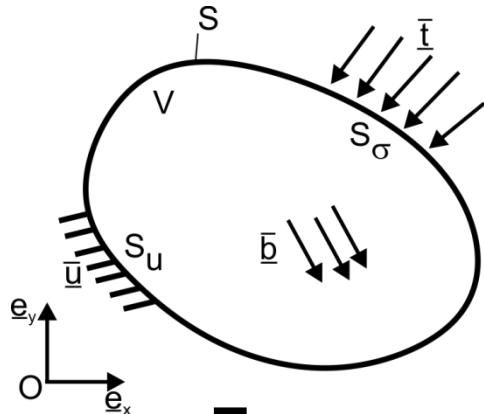
T6



Q8 & Q9

2. Quick introduction to FEM

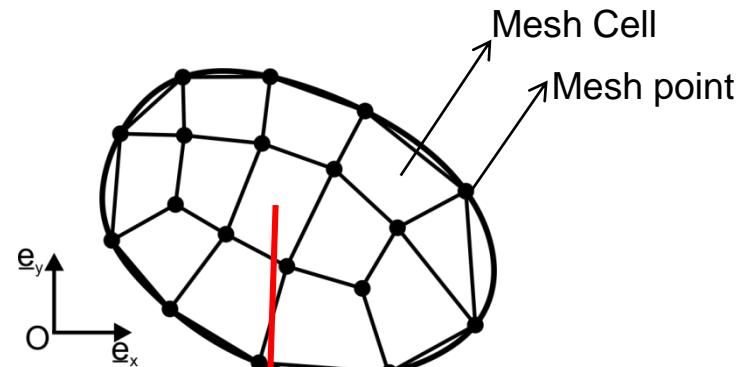
Mechanical problem:



Spatial discretization



Mesh:



Finite Element = Cell + Material + Hypothesis:



Linear system of equations:

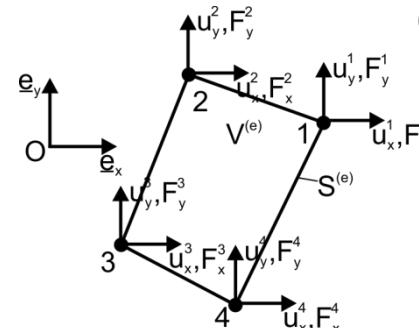
$$\underline{\underline{K}}^{(e)} \underline{u}^{(e)} = \underline{F}^{(e)}$$

Variational principle:
Rayleigh Ritz's method,
Galerkin,...



Finite number of unknowns:

Approximate solution.



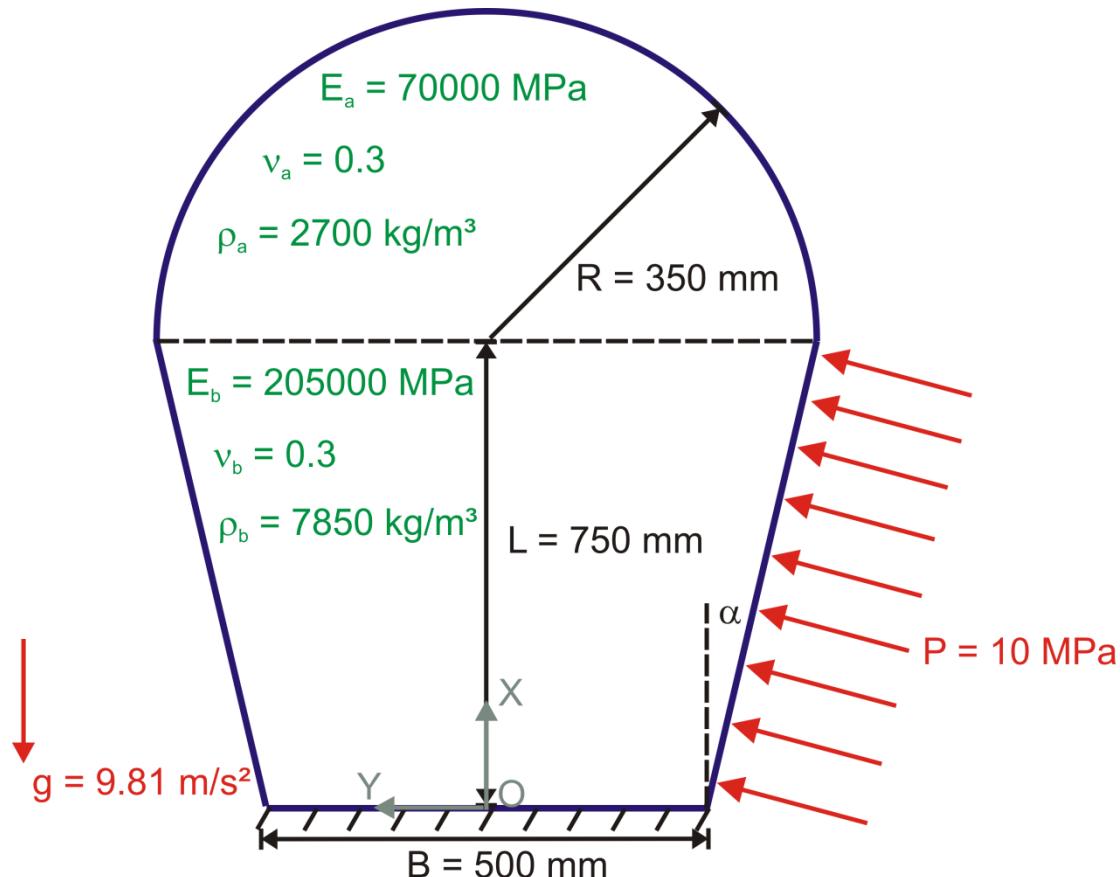
Governing equations:
PDEs + BCs

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
- 3. Mechanical problem description**
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

3. Mechanical problem description

Consider the following structure in plane strain state:



3. Mechanical problem description

➤ Governing equations:

The exact solution satisfies the following equations:

1. Equilibrium equations.
2. Static-displacements equations.
3. Constitutive equations.
4. Kinematic (= Essential) boundary conditions.
5. Static (= Natural) boundary conditions.

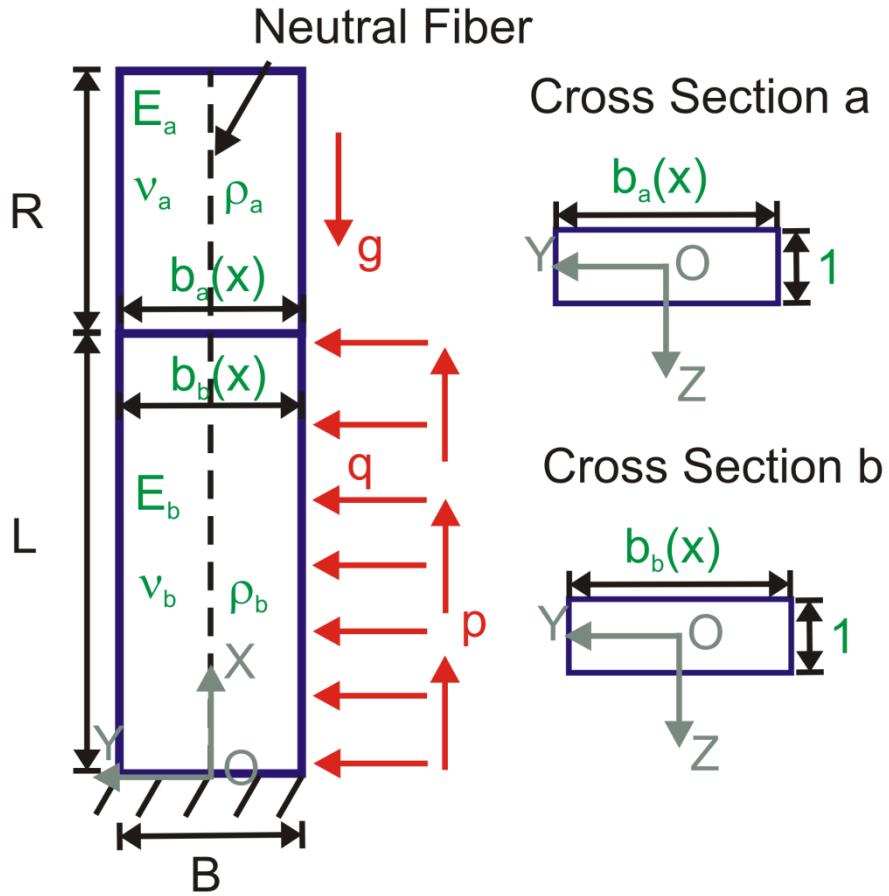
Remark: see the detailed equations in the annexes.

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
- 4. Analysis with strength of materials**
 - 1. Idealization of the problem**
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

4.1 Idealization of the problem

➤ Beam Idealization:



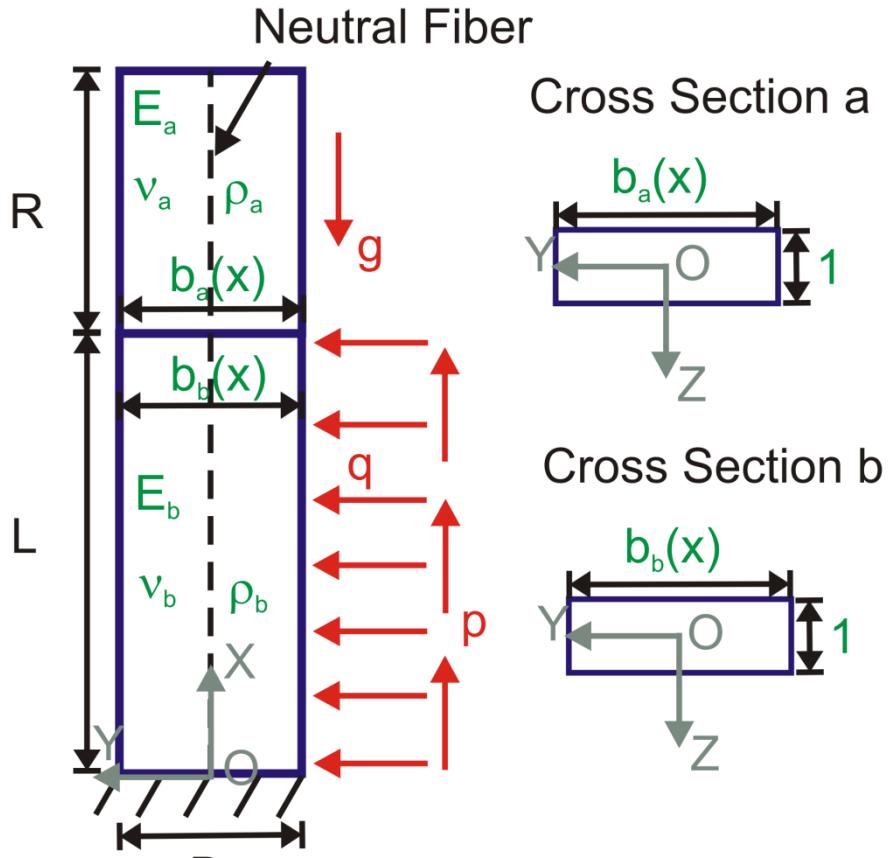
Beam a: $x \in [L, L + R]$

- Width distribution $b_a(x) = 2 \sqrt{R^2 - (x - L)^2}$
- Height distribution $h_a(x) = 1$
- Area distribution:
$$A_a(x) = b_a(x)h_a(x) = 2 \sqrt{R^2 - (x - L)^2}$$
- Volume $V_a = \frac{\pi R^2}{2}$
- Second principal moment of inertia around Z axis:

$$I_{z_a}(x) = \frac{b_a^3(x)h_a(x)}{12} = \frac{2}{3} (R^2 - (x - L))^{\frac{3}{2}}$$

4.1 Idealization of the problem

➤ Beam Idealization:



Beam b: $x \in [0, L]$

➤ Width distribution : $b_b(x) = B + \frac{2R - B}{L}x$

➤ Height distribution $h_b(x) = 1$

➤ Area distribution:

$$A_b(x) = b_b(x)h_b(x) = B + \frac{2R - B}{L}x$$

➤ Volume : $V_b = (B + 2R) \frac{L}{2}$

➤ Second principal moment of inertia around Z axis:

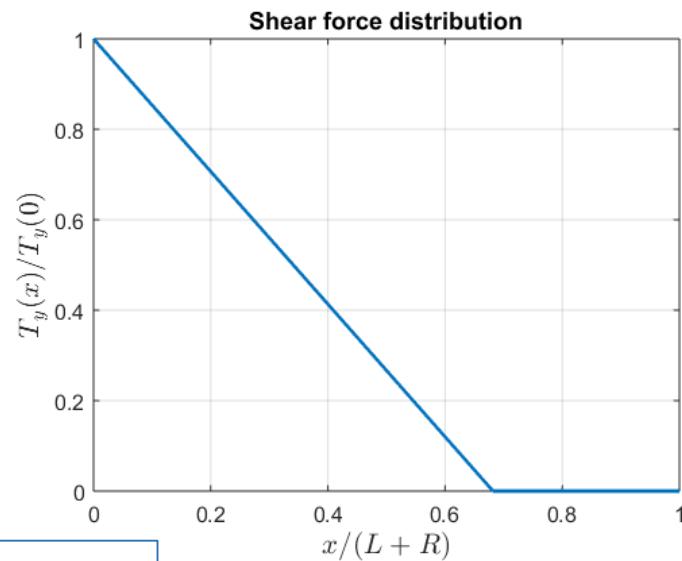
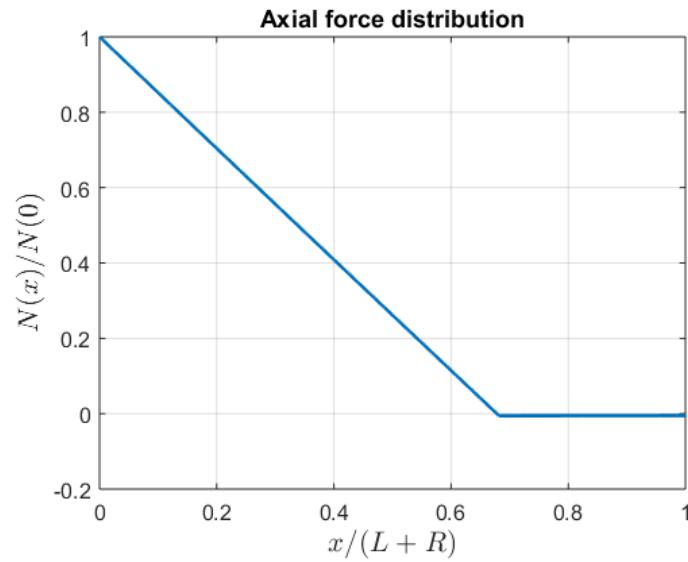
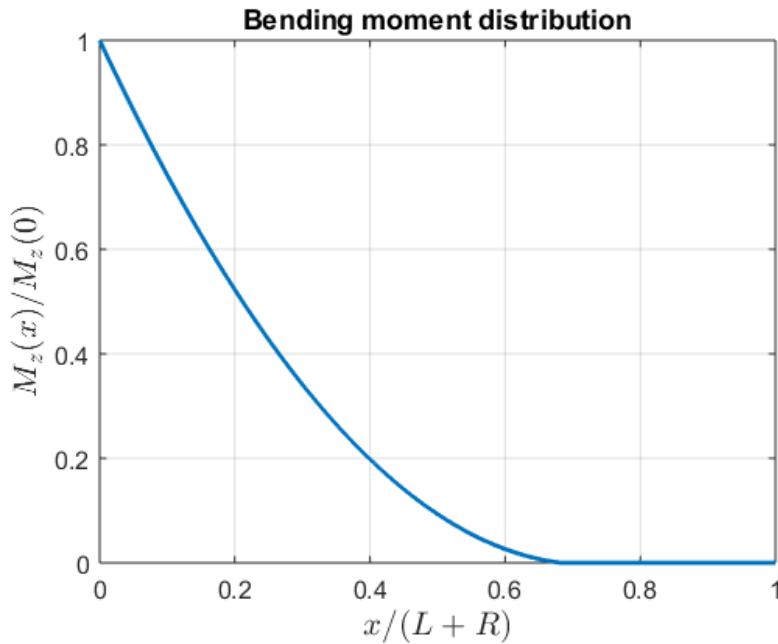
$$I_{z_b}(x) = \frac{b_b^3(x)h_b(x)}{12} = \frac{1}{12} \left(B + \frac{2R - B}{L}x \right)^3$$

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
- 4. Analysis with strength of materials**
 1. Idealization of the problem
 - 2. Relevant results**
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

4.2 Relevant results

➤ Stress Resultants: Sum up



In this case:
The maximum stresses
occurs at the restraint!

Remark: see the detailed equations in the annexes.

4.2 Relevant results

➤ Effective von Mises stress at the restraint? ($x = 0$)

- Find the stress resulting from axial force N : $\sigma_{xx}^{Tension}$
 - Find the stress resulting from shear force T_y : $\tau_{xy}^{Shear}(y)$
 - Find the stress resulting from bending moment M_z : $\sigma_{xx}^{Bending}(y)$
 - Find the effective von Mises stress:
 - i. Beam: $\sigma_{yy} = 0$
 - ii. Plane strain: $\sigma_{zz} = \nu_b \sigma_{xx}$ & $\tau_{zx} = \tau_{yz} = 0$
- $\bar{\sigma}^{VM} = \sqrt{(1 - \nu_b + \nu_b^2) \sigma_{xx}^2 + 3\tau_{xy}^2}$

➤ Upper layer ($y = \frac{b_b(0)}{2}$) : $\bar{\sigma}^{VM} \left(\frac{b_b(0)}{2} \right) = 64.121 \text{ MPa}$

➤ Neutral fiber ($y = 0$) : $\bar{\sigma}^{VM}(0) = 38.666 \text{ MPa}$

➤ Lower layer ($y = -\frac{b_b(0)}{2}$) : $\bar{\sigma}^{VM} \left(-\frac{b_b(0)}{2} \right) = 67.504 \text{ MPa}$

Remark: see the detailed equations in the annexes.

4.2 Relevant results

➤ Total Potential Energy:

- By definition, the total potential energy is : $TPE = \mathcal{U} - \mathcal{P}$
 - \mathcal{P} is the potential energy of external applied loads.
 - \mathcal{U} is the strain energy.
- If the **essential boundary conditions** consist to impose the displacement field to **zero** at some boundary parts, it may be proven **at equilibrium** that :

$$TPE = \mathcal{U} - \mathcal{P} = -\frac{1}{2}\mathcal{P} = -\mathcal{U}$$

Since, $\mathcal{U} = \frac{1}{2}\mathcal{P}$ **at equilibrium**.

- In our example: $TPE = -459.916J$
- In FEM, the absolute value of the TPE converges to the exact one when the number of DOFs increases (Mesh refinement).
- In practice, the asymptotic value is reached for simple mechanical problem only.
- In the SAMCEF solver:
 - The given value of “Total Potential Energy” is actually $W^{int} = \mathbf{q}_s^T \mathbf{K}_s \mathbf{q}_s = 2 \times |TPE|$.
 - Which is also the work done by the applied external forces: $W^{ext} = \mathbf{q}_s^T \mathbf{g}_s$.

4.2 Relevant results

➤ Displacement Field: Results

$$\Rightarrow v_{max} = v_{max}^{(a)} + v_{max}^{(b)} = 0.322 \text{ mm}$$

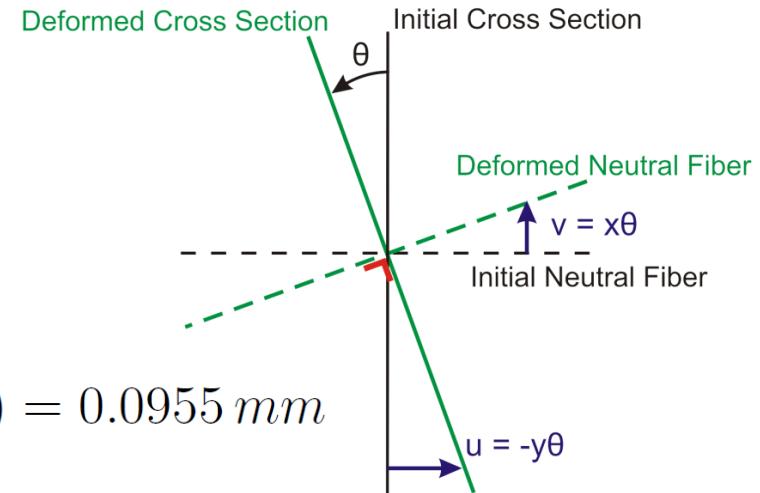
$$\Rightarrow u(L) = u^{(a)} + u^{(b)} = 0.00278 \text{ mm}$$

$$\Rightarrow \theta(L) = \theta^{(a)} + \theta^{(b)} = 0.000265 \text{ radians}$$

➤ Euler Bernoulli beam assumption:

$$u(x, y) = u(x) - y\theta(x)$$

$$\Rightarrow u_{max} = u(x = L, y = -R) = u(L) + R\theta(L) = 0.0955 \text{ mm}$$



Remark: see the detailed equations and assumptions in the annexes.

4.2 Relevant results

➤ Range of validity:

- Euler Bernoulli assumptions.
- Tapered Beam:
 - The cross section variation has been neglected for stress computation.
- Stress Gradient/Concentration:
 - Stress gradient in the y direction neglected.
 - Poisson effect neglected.
- St. Venant's Principle.

Beam theory yields a 1D model along the beam longitudinal axis !

Remark: see the detailed equations
and assumptions in the annexes.

4.2 Relevant results

➤ Reference Books (<http://fr.bookzz.org/>; <http://lib.ulg.ac.be/>):

- Elasticity, J.R. Barber, Springer, 2009
- Peterson's Stress Concentration Factors, Walter D.P. & Deborah F. P., John Wiley & Sons, 2008
- Theory of Elasticity, Timoshenko S., McGraw-Hill, 1934
- Applied mechanics of solids, A. F. Bower, <http://solidmechanics.org/>, 2009
- Elasticity in engineering mechanics, A. P. Boresi, K. P. Chong, and J. D. Lee, Wiley-Interscience New York, 2010.
- Rules of Thumb for Mechanical Engineers, Pope Edward J., Elsevier, 1996
- Roark's Formulas for Stress and Strain, Warren C. Young and Richard G. Budynas, McGraw-Hill, 2002.
- Shigley's Mechanical Engineering Design, Budynas Richard and Nisbett Keith, McGraw-Hill, 2008.
- Contact Mechanics, K. L. Johnson, Cambridge University Press, Cambridge, 1985.

The last book is helpful for checking whether the loading yields the presence of singularities or a finite jump of stress components at the points of application.

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

5. Analysis with NX 12

- 1. Introduction**
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
 7. General remarks
 8. General project instructions

5.1 Introduction

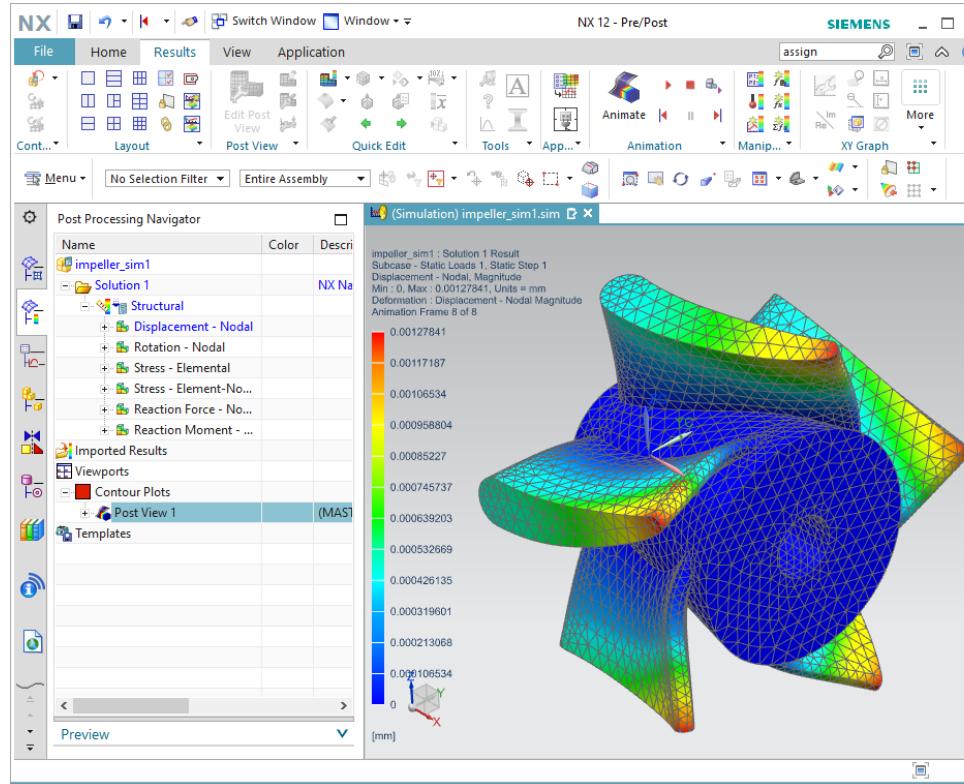
Please note that the project was done on NX 12 the previous years.

From this year, NX 1859 is now used. The interface is similar to the one of NX 12, though the design of the icons have changed.

Therefore, the tutorial shown here, which was made on NX 12 is still valid on NX 1859.

5.1 Introduction

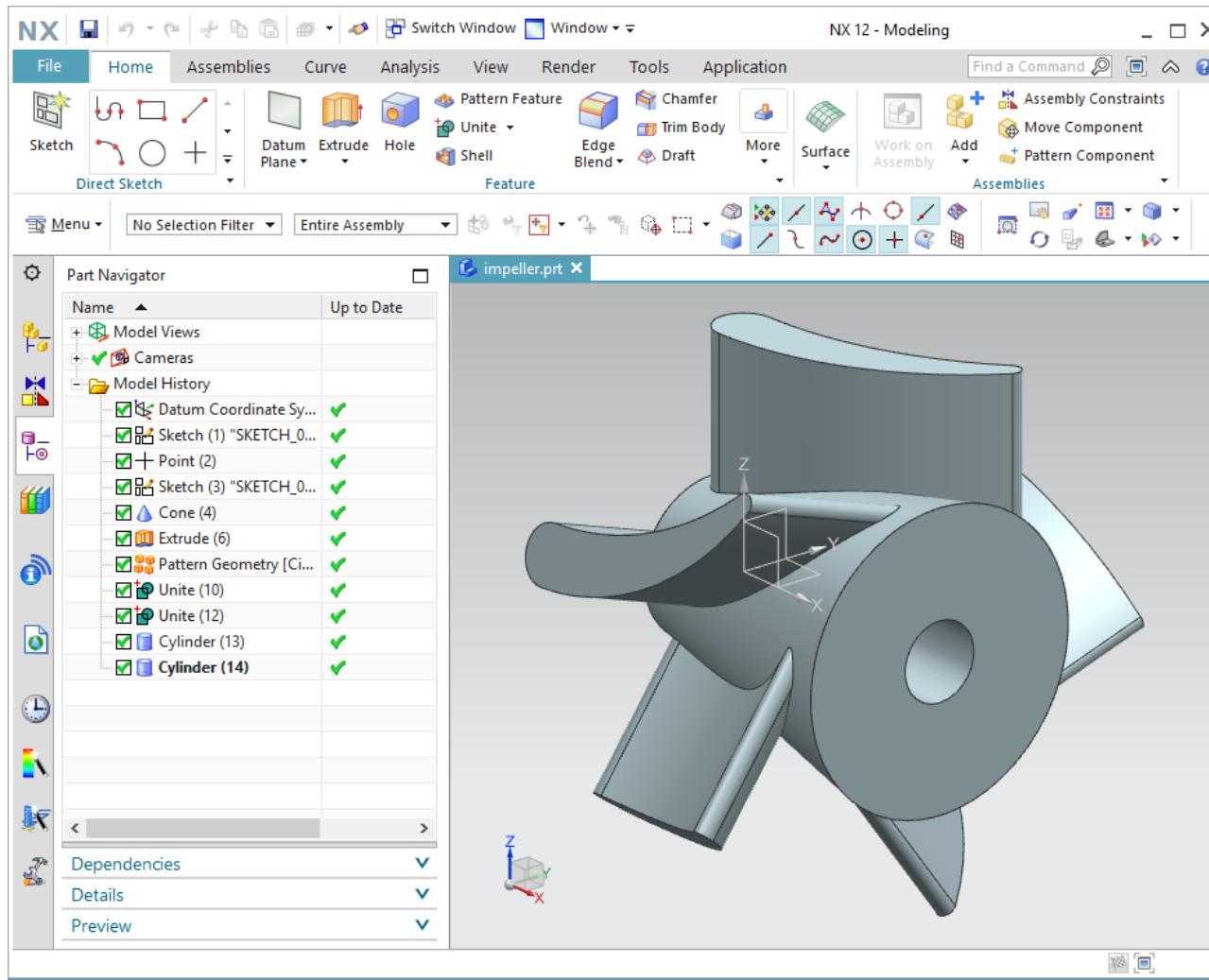
Finite element analysis with NX 12:



1. .prt: Geometry of the model.
2. .fem: Mesh, physical and material properties.
3. .sim: Boundary conditions, loads and solutions.

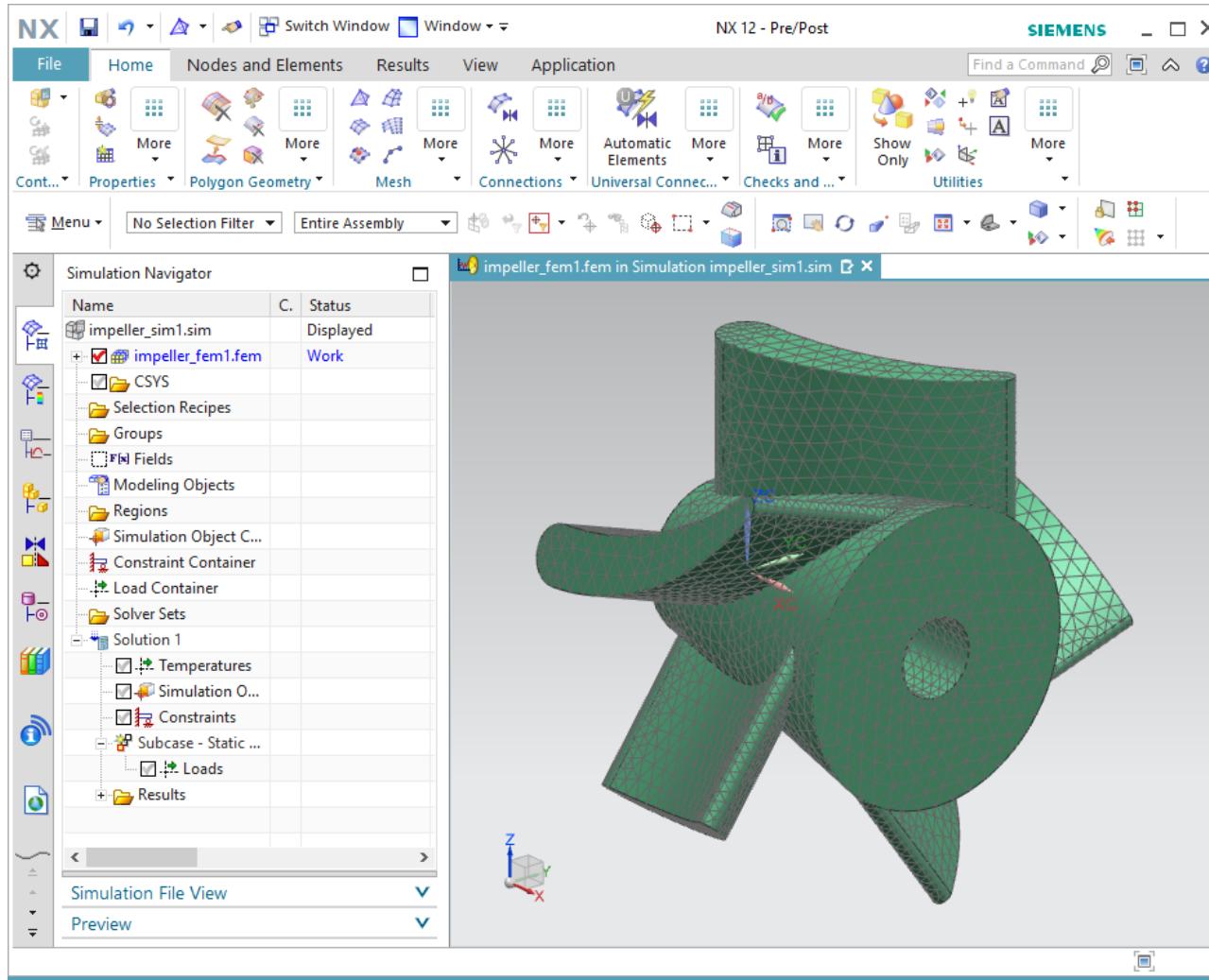
5.1 Introduction

1. .prt: Geometry of the model.
2. .fem: Mesh, physical and material properties.
3. .sim: Boundary conditions, loads and solutions.



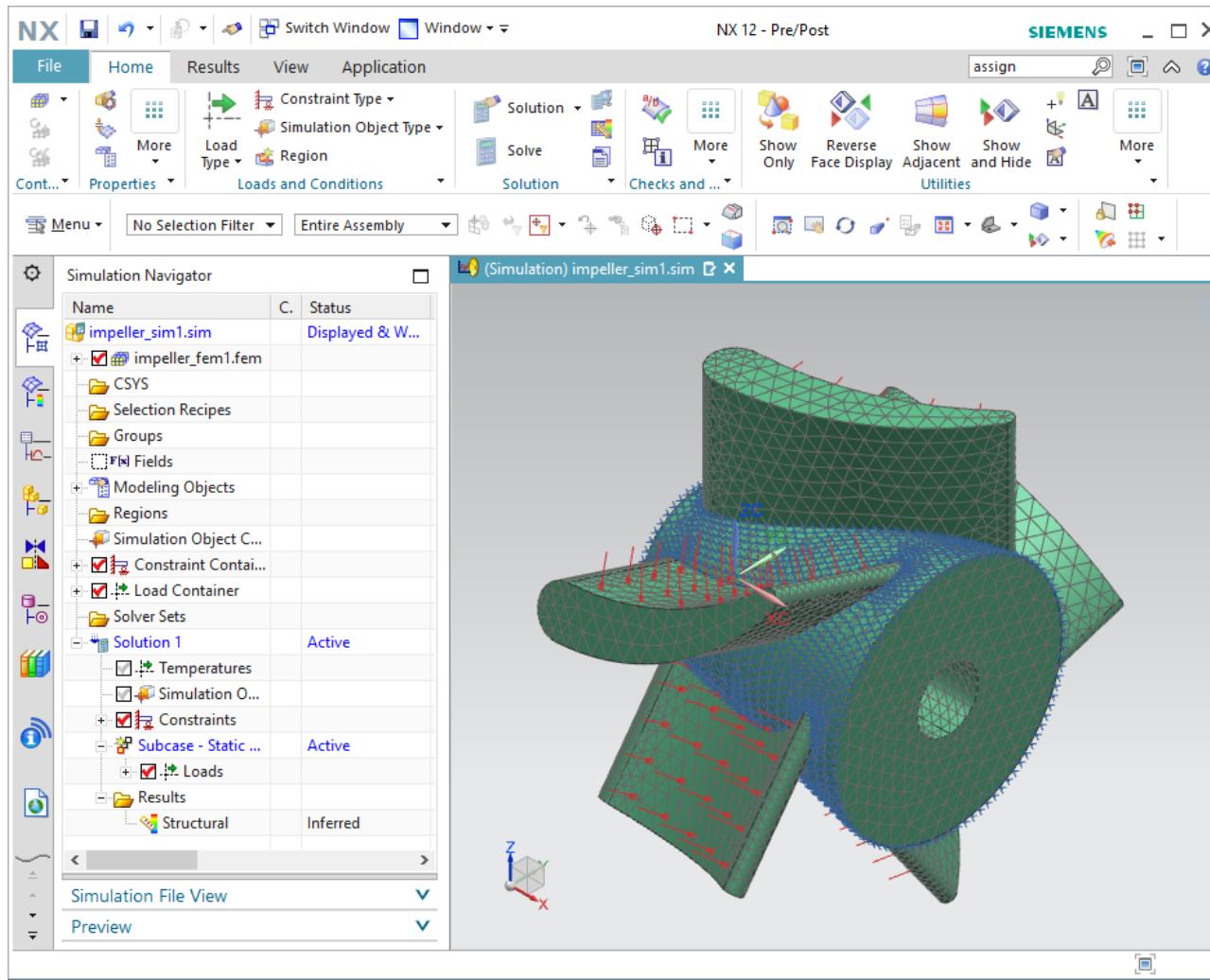
5.1 Introduction

1. .prt: Geometry of the model.
2. .fem: Mesh, physical and material properties.
3. .sim: Boundary conditions, loads and solutions.



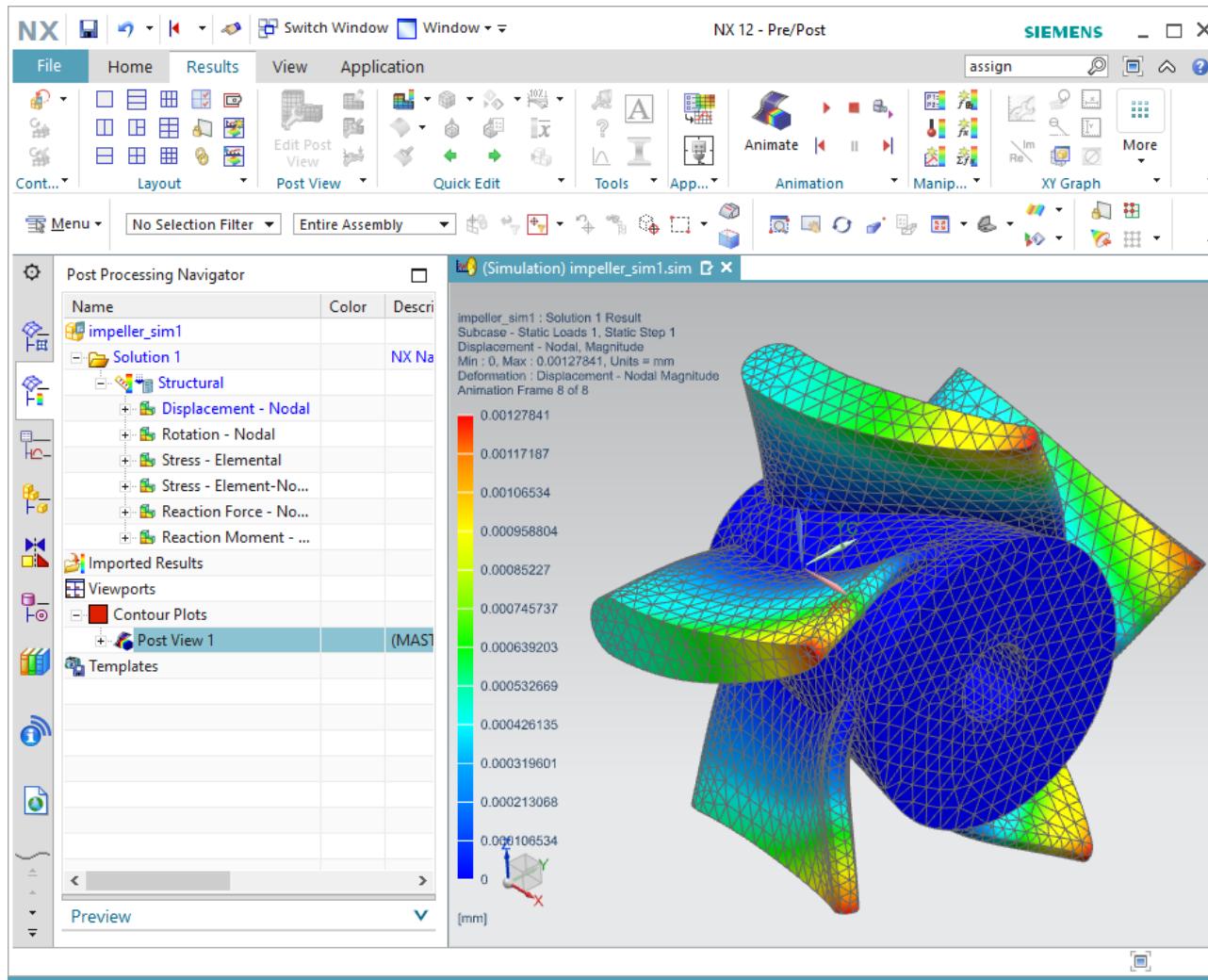
5.1 Introduction

1. .prt: Geometry of the model.
2. .fem: Mesh, physical and material properties.
3. .sim: Boundary conditions, loads and solutions.



5.1 Introduction

If there are no errors, you can solve your model!



Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

5. Analysis with NX 12

1. Introduction
 - 2. Moving around in NX**
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
 7. General remarks
 8. General project instructions

5.2 Moving around in NX

Visualisation Windows Command:

- [Left Click]: Select or Drag Objects.
- [Shift] + [Left Click]: Deselect Objects.

- [Middle Click]: Rotate the view.
- [Right Click]: Scroll in lists, menus and the information window.
- [Wheel] or [Ctrl] + [Middle Click]: Zoom.
- [Middle Click] + [Right Click] or [Shift] + [Middle Click]: Pan.

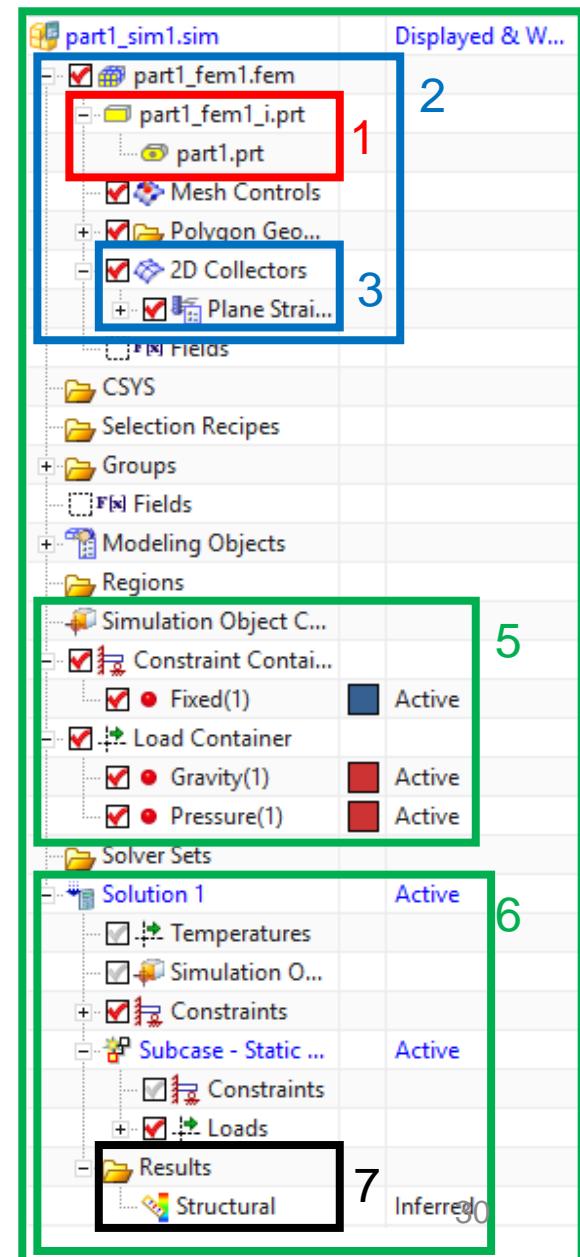
- [F8]: Go to nearest view.
- [Ctrl]+ [F]: Fit the work part to the screen.
- In sketch: [Shift]+[F8]: Go back to sketch plane.

5.2 Moving around in NX

Simulation Navigator:

- 1: CAD Geometry.
- 2: Associated FEM components.
- 3: Mesh collectors and meshes:
- 4: Associated SIM components.
- 5: Boundary Conditions: Loads and Constraints.
- 6: Solutions
- 7. Results.

- .prt
- .fem
- .sim

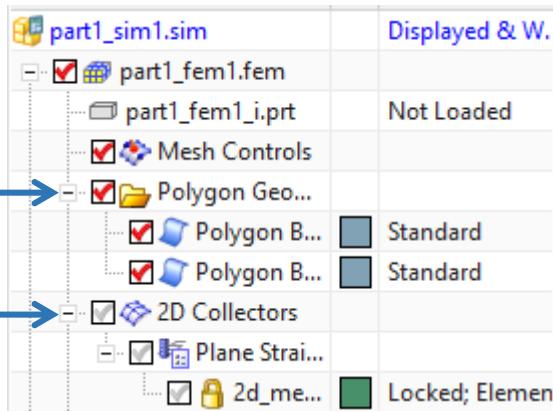


Easy switch
between files.

Simulation File View	
Name	Status
Session	
- model1_sim1	Displayed
- model1_fem1	Work
- model1_fe...	
- model1	

5.2 Moving around in NX

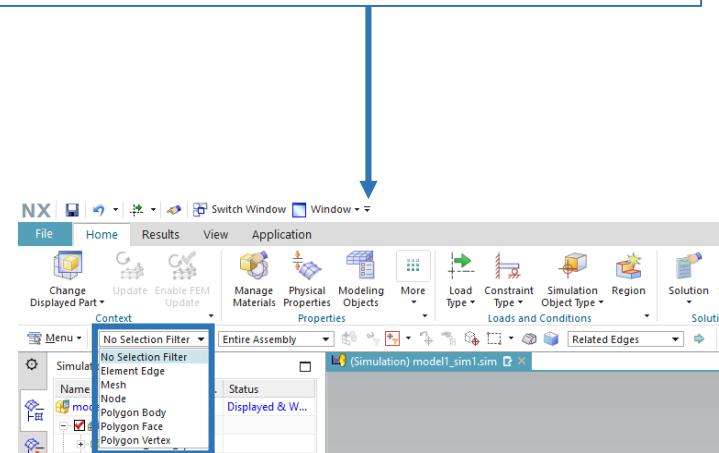
Show/hide:



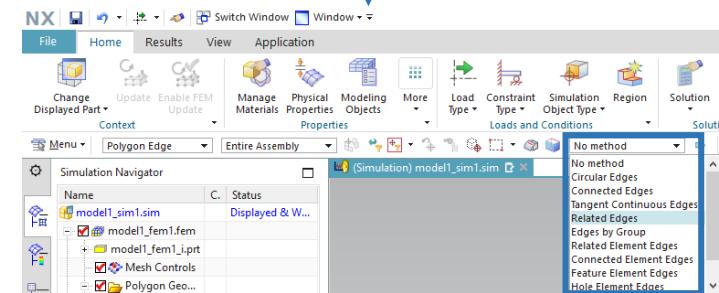
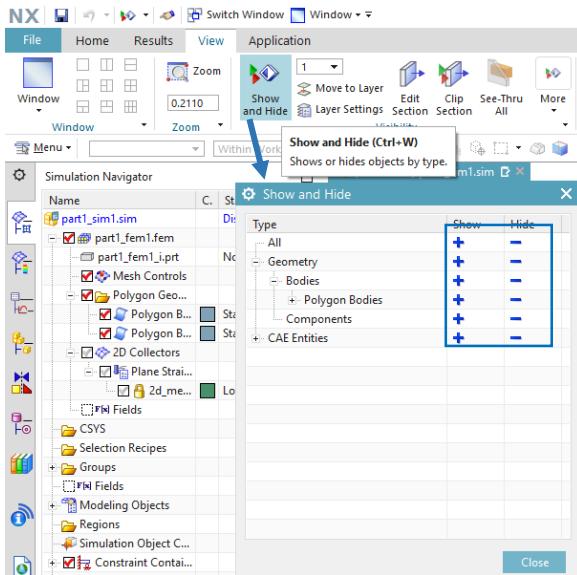
Shown

Hidden

Smart selection toolbar:



Detailed show/hide options:



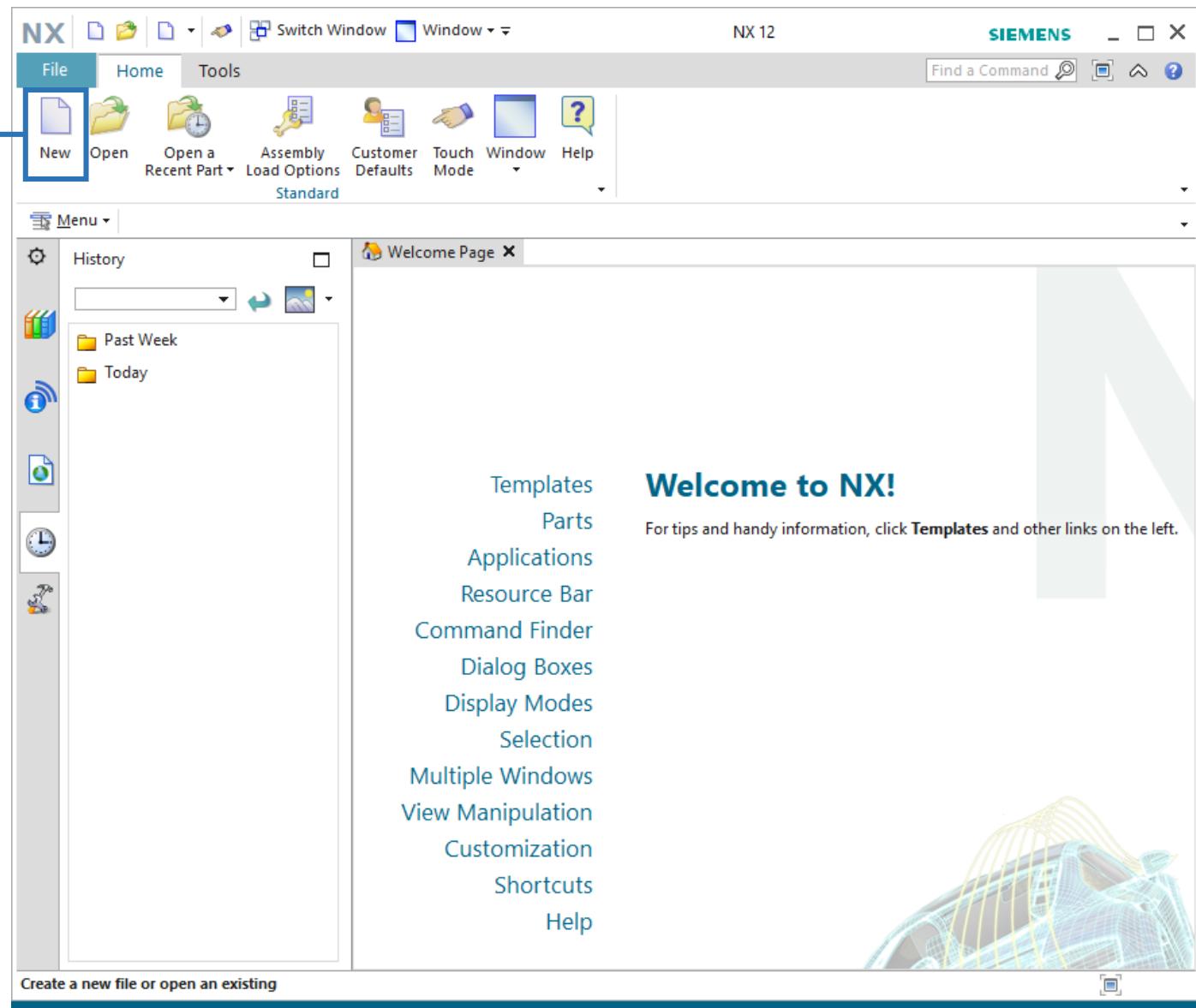
Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

5. Analysis with NX 12

1. Introduction
2. Moving around in NX
- 3. Geometry drawing (.prt)**
4. Generation of .fem and .sim files
5. Material properties (.fem)
6. Mesh generation(.fem)
7. Boundary conditions and loads (.sim)
8. Launch a linear static analysis (.sim)
9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

5.3 Geometry drawing (.prt)



5.3 Geometry drawing (.prt)

Click on “ Model”.

Choose a name and a folder for your new model.

OK

Cancel

Click on “OK”

New

Machining Line Planner Manufacturing Inspection Mechatronics Concept Designer Press Line Line Designer Ship Structures

Ship General Arrangement Model DMU Drawing Layout Simulation Additive Manufacturing

Templates

Filters

Units Millimeters

Name	Type	Units	Relationship	Owner
Model	Modeling	Millimeters	Stand-alone	NT AUTH...
Assembly	Assemblies	Millimeters	Stand-alone	NT AUTH...
Shape Studio	Shape Studio	Millimeters	Stand-alone	NT AUTH...
Sheet Metal	Sheet Metal	Millimeters	Stand-alone	NT AUTH...
Routing Logical	Routing Logical	Millimeters	Stand-alone	NT AUTH...
Routing Mechanical	Routing Mecha...	Millimeters	Stand-alone	NT AUTH...
Routing Electrical	Routing Electrical	Millimeters	Stand-alone	NT AUTH...
Blank	Gateway	Millimeters	Stand-alone	none

Preview

Properties

Name: Model
Type: Modeling
Units: Millimeters
Last Modified: 09/11/2017 06:07 AM
Description: NX Example with datum CSYS

New File Name

Name: model1.prt

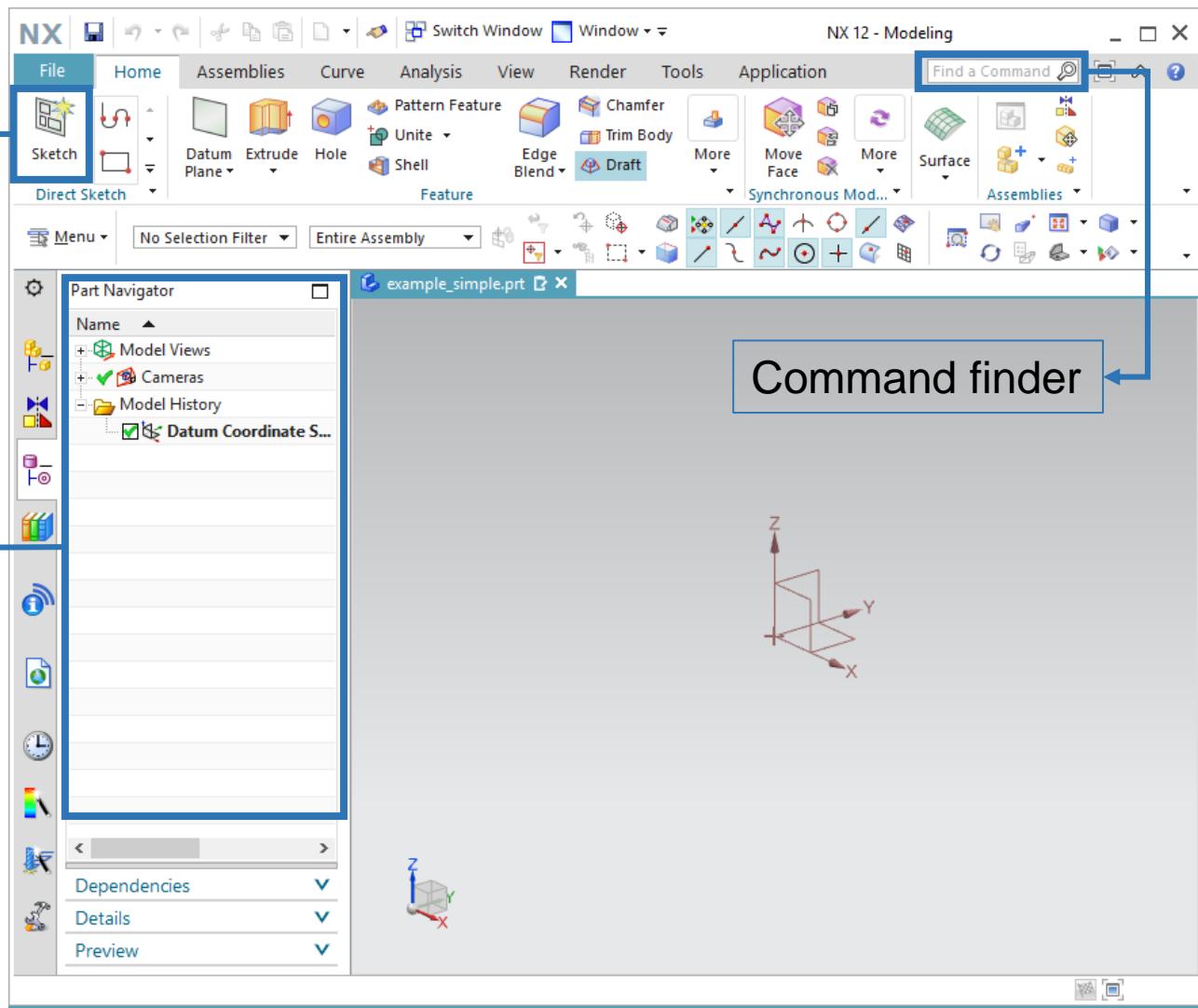
Folder: D:\Doctorat\Cours\Finite Element Method\Project\

Part to reference

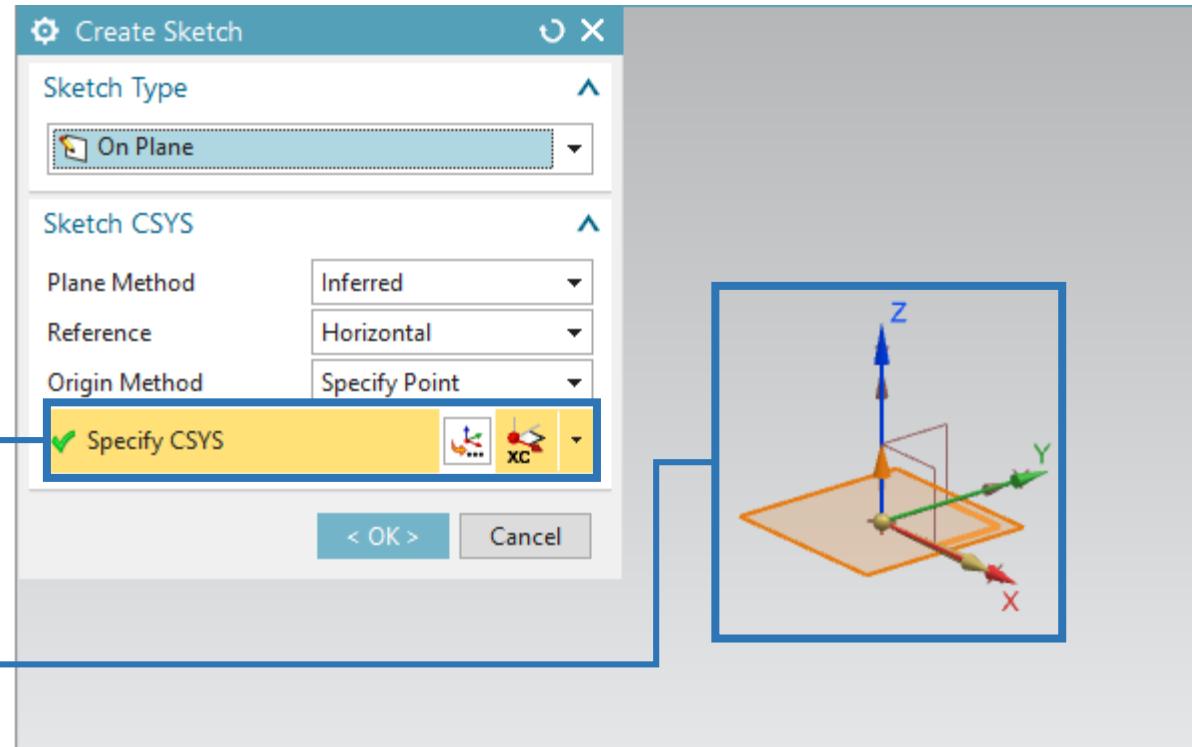
Name:

5.3 Geometry drawing (.prt)

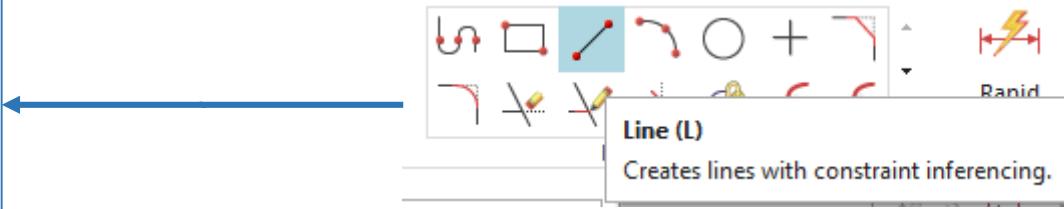
1. Create a new “Sketch”.



5.3 Geometry drawing (.prt)

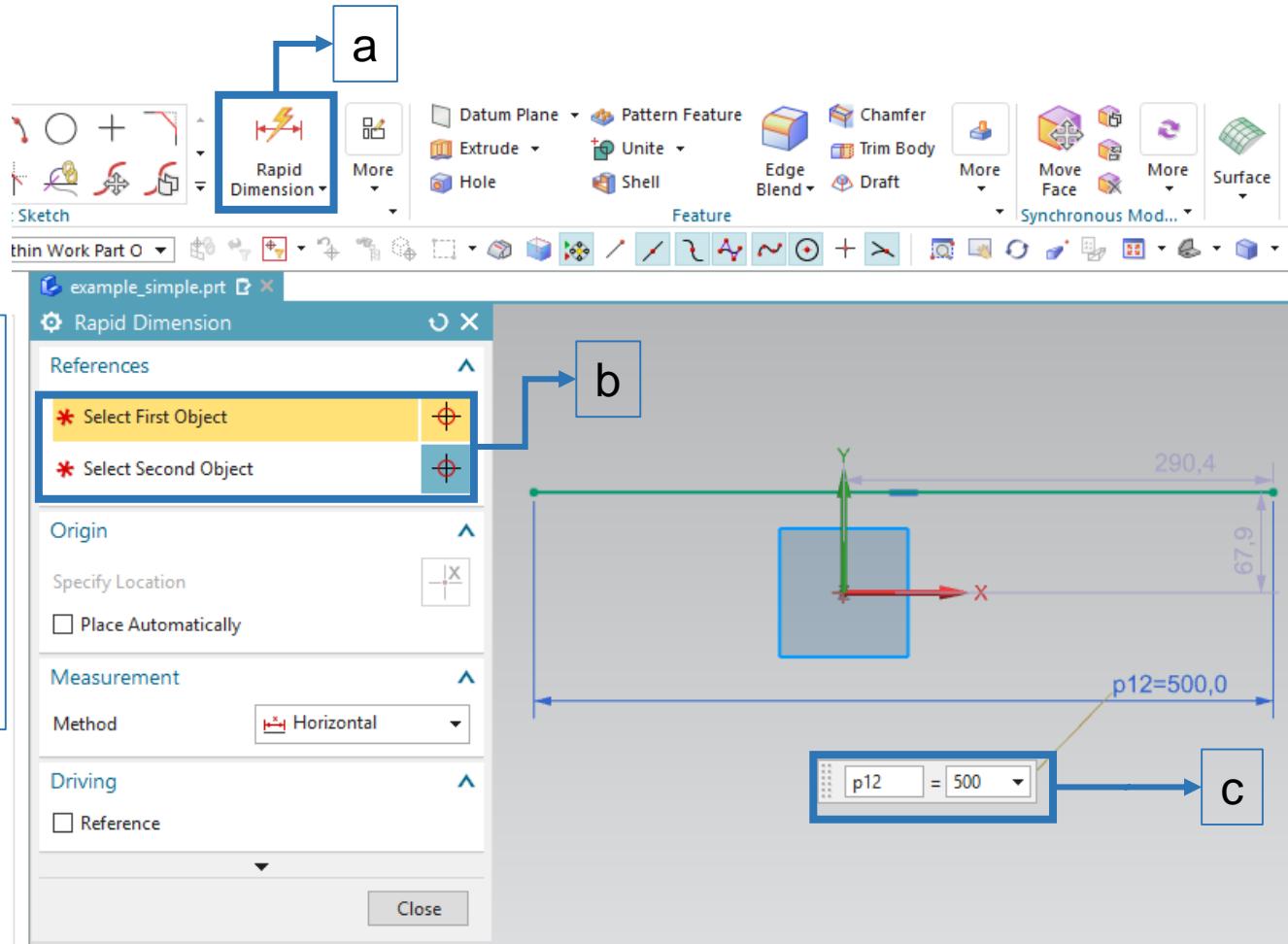


2. In “Specify CSYS”, select the XY plane in the coordinate frame (in orange).
3. Click on “Line” and draw a horizontal line of random length.

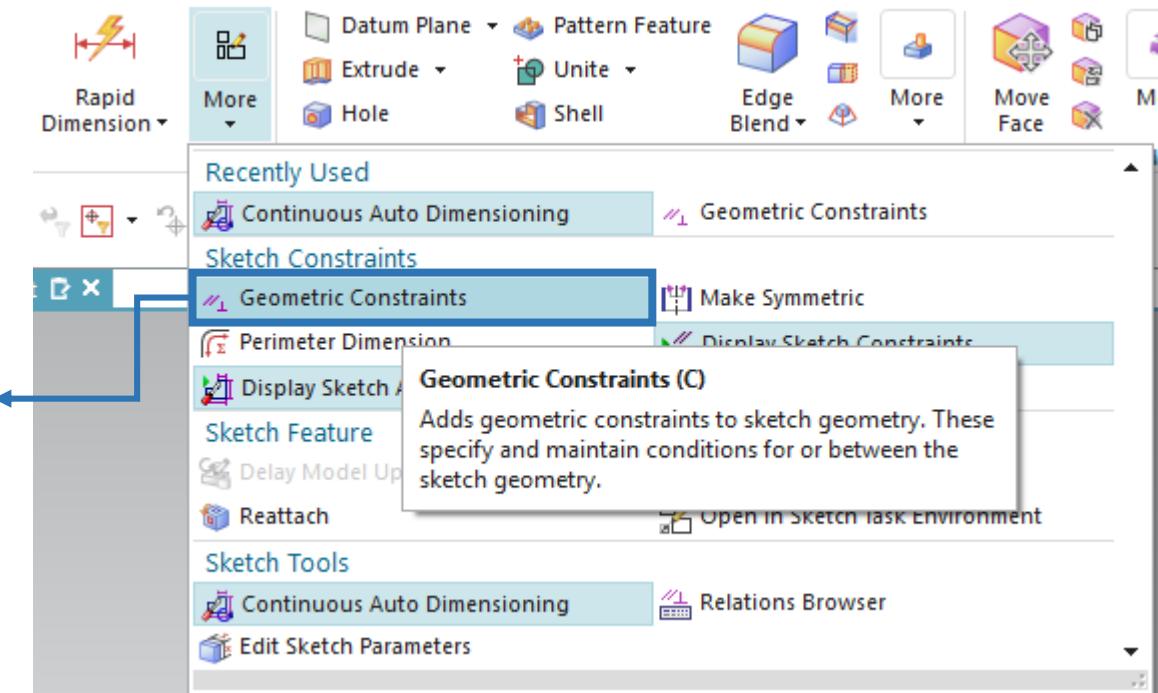


5.3 Geometry drawing (.prt)

4. Specify the length of the line:
 - a. Click on “Rapid Dimension”.
 - b. Select the line
 - c. Enter “500mm”.



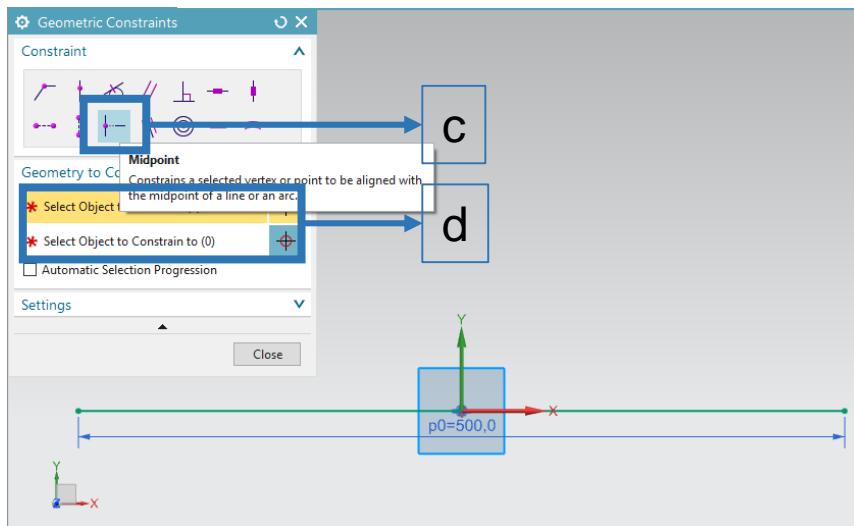
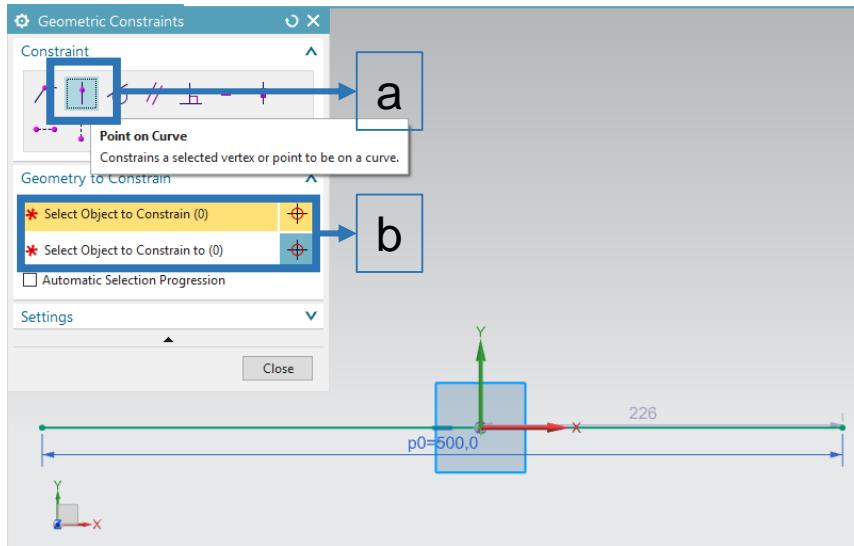
5.3 Geometry drawing (.prt)



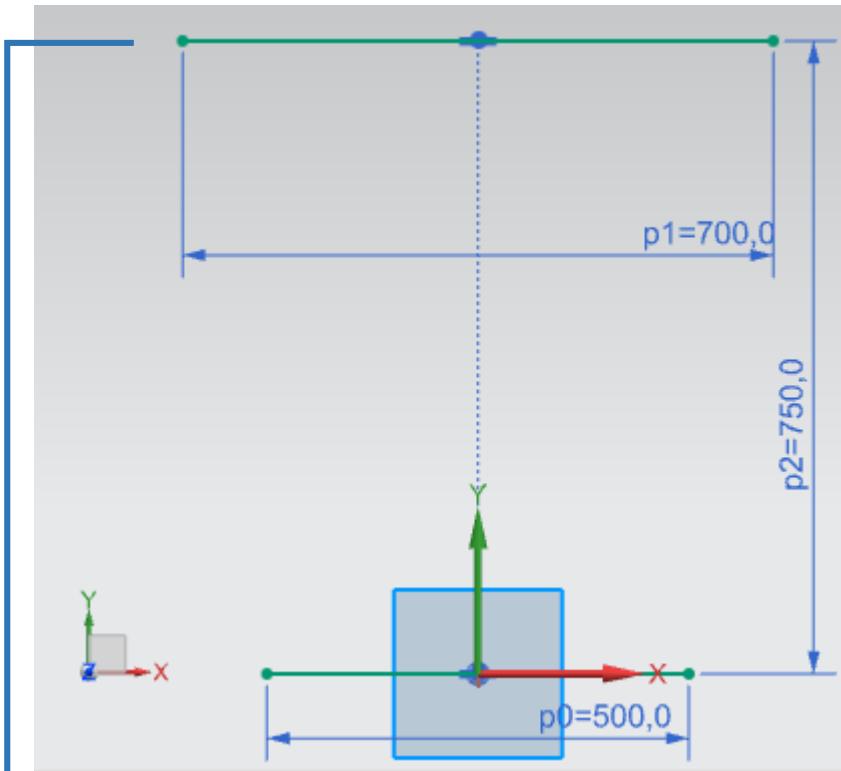
5. To place the line at the right position, click on "More" → "Geometric Constraints".

5.3 Geometry drawing (.prt)

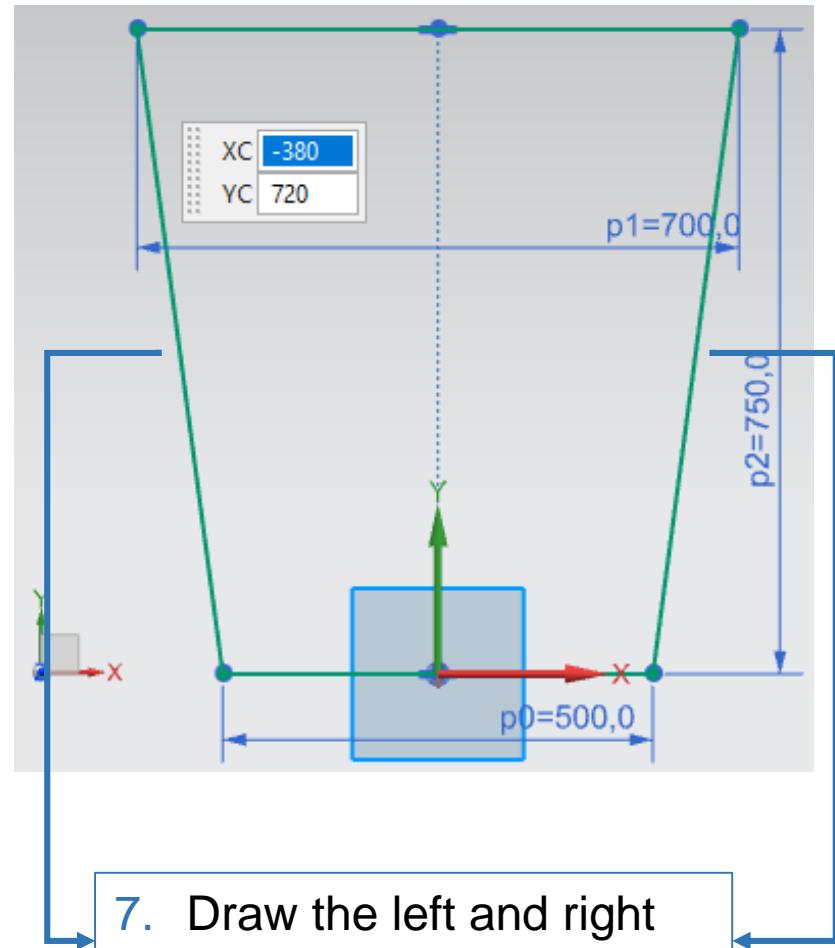
5. To place the line at the right position, click on “More”
→ “Geometric Constraints”:
 - a. Click on “Point on Curve”.
 - b. Select the centre of the XYZ frame & your line.
➤ “Apply”.
 - c. Click on “Midpoint”.
 - d. Select the centre of the XYZ frame & your line.
➤ “Apply”.



5.3 Geometry drawing (.prt)

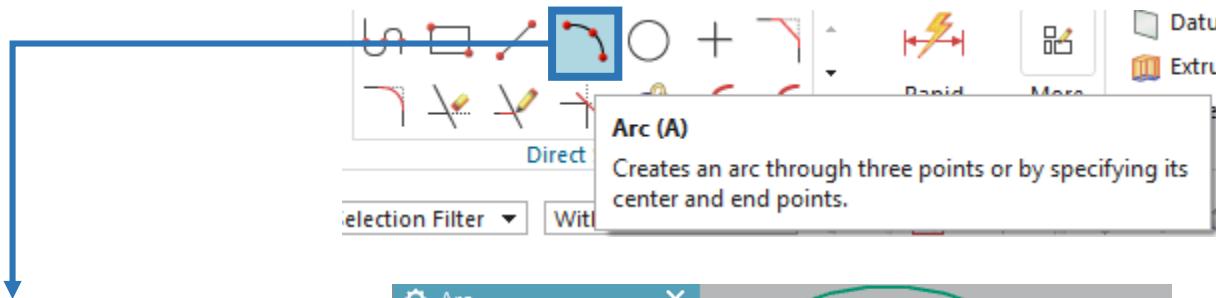


6. Using the same methodology, draw the upper line of the model.

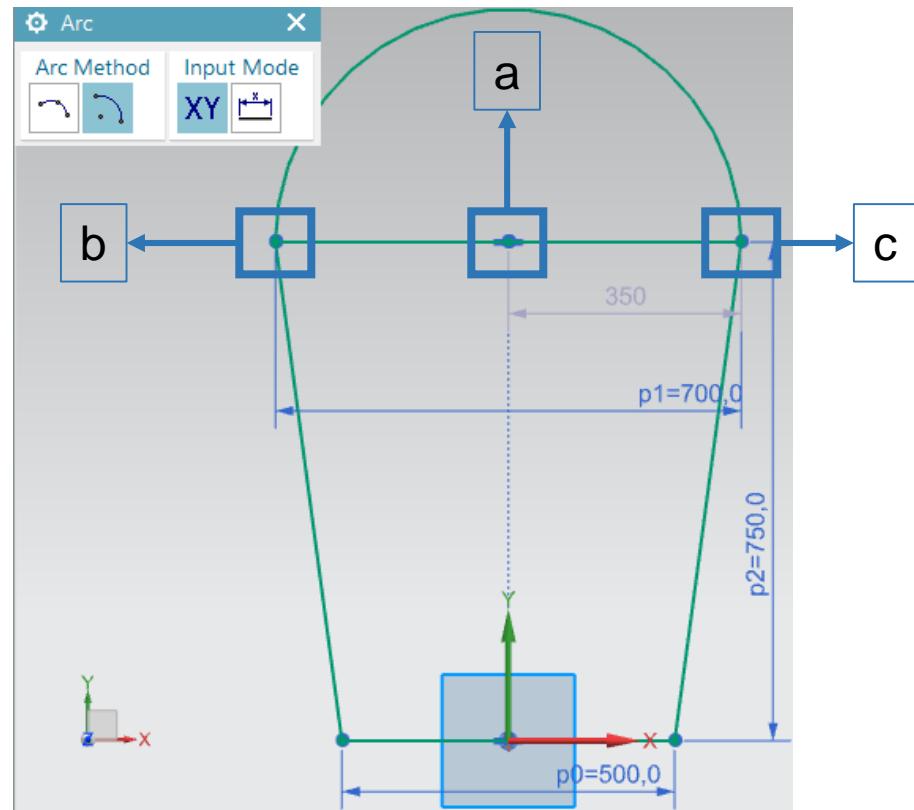


7. Draw the left and right edges of the model.

5.3 Geometry drawing (.prt)



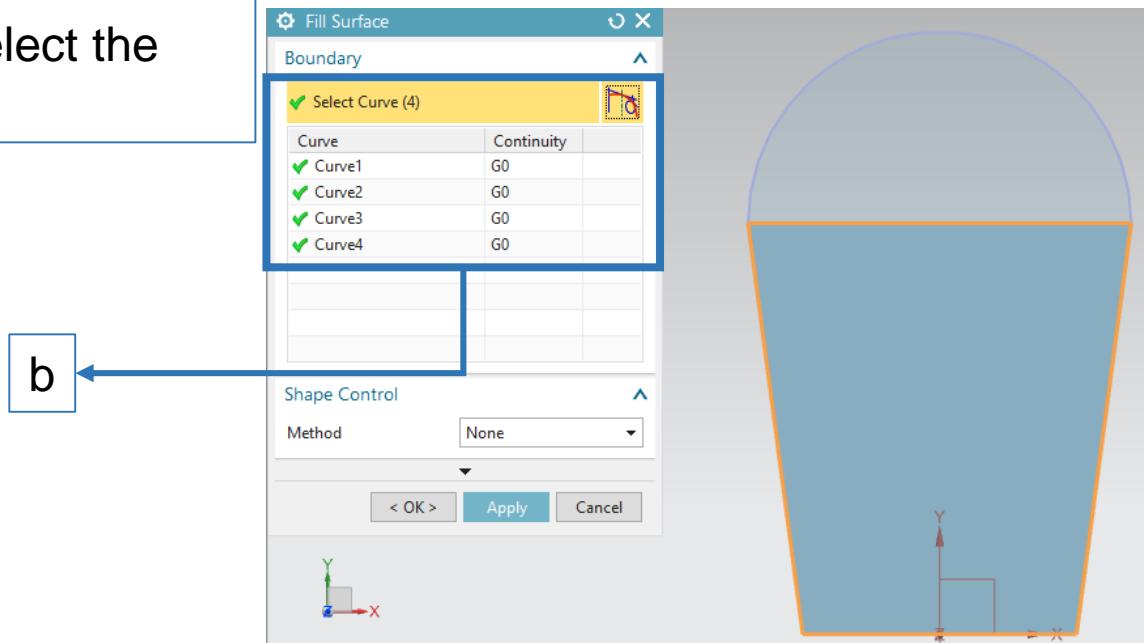
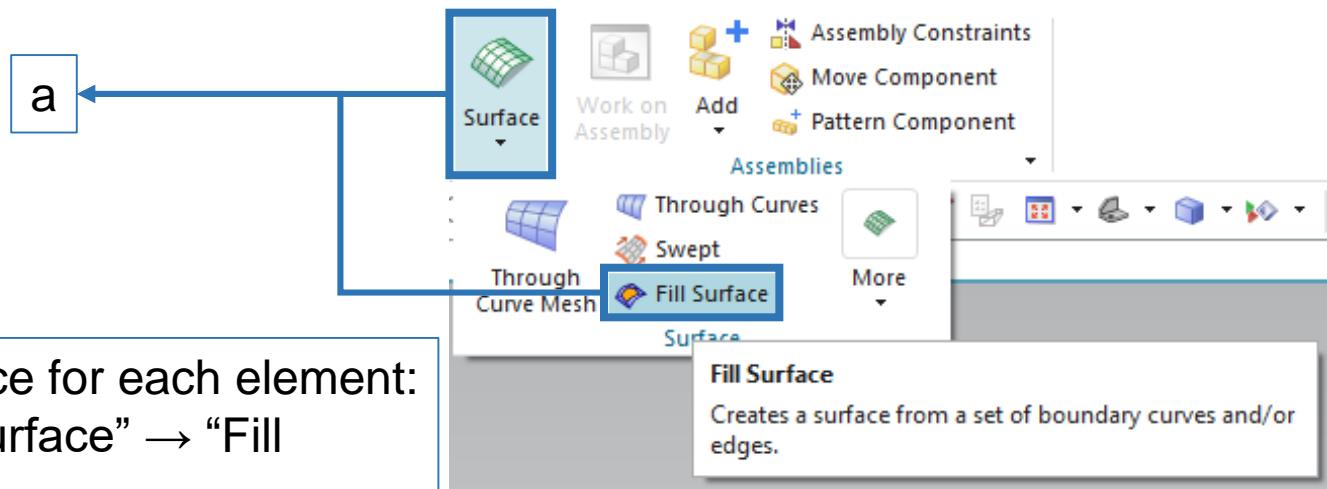
8. Click on “Arc”:
 - a. Select midside point.
 - b. Select start point.
 - c. Select end point.



9. Click on “Finish Sketch”.

5.3 Geometry drawing (.prt)

10. Create a surface for each element:
- Click on “Surface” → “Fill Surface”.
 - For each element select the boundary curves.



Presentation Plan

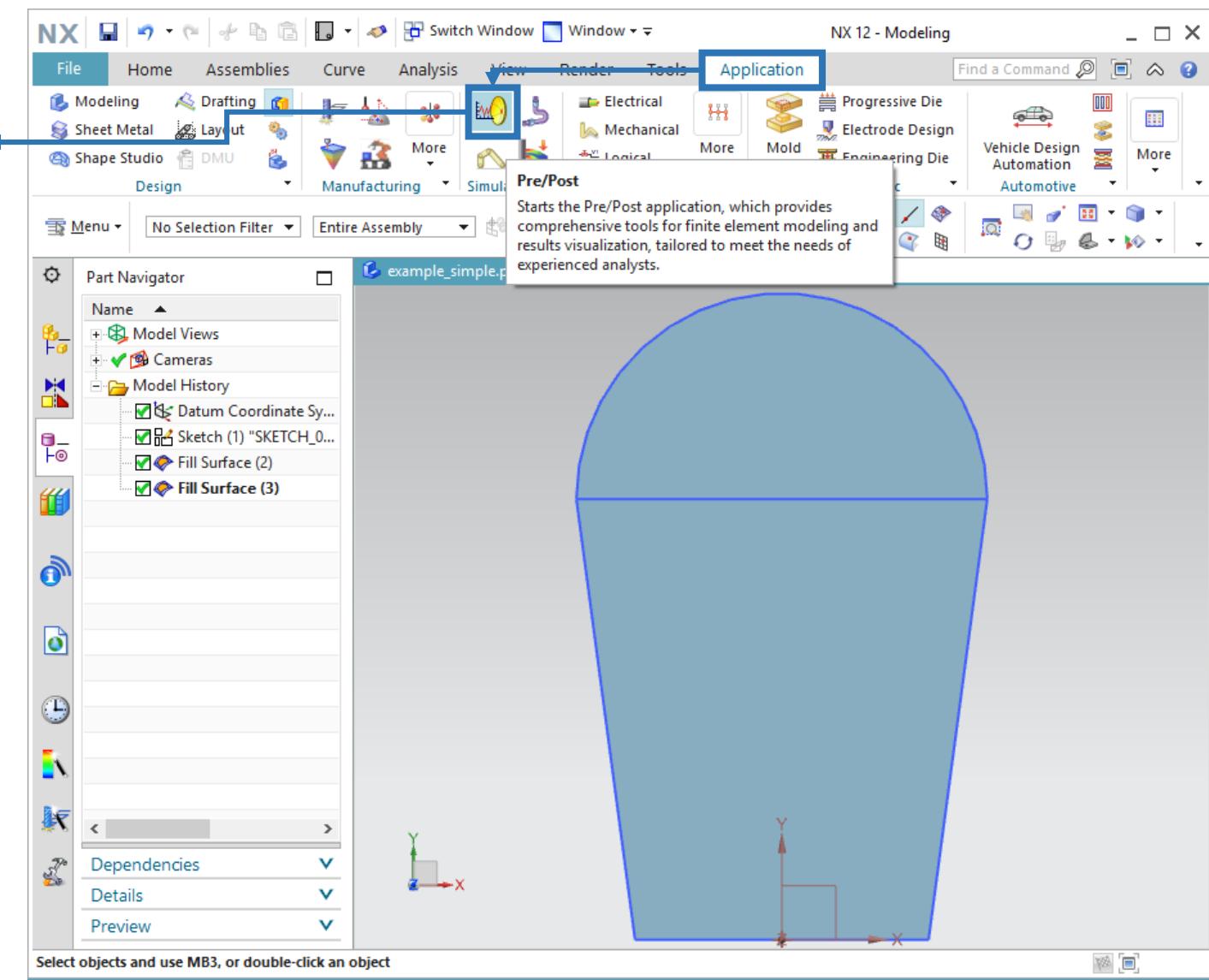
1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

5. Analysis with NX 12

1. Introduction
2. Moving around in NX
3. Geometry drawing (.prt)
- 4. Generation of .fem and .sim files**
5. Material properties (.fem)
6. Mesh generation(.fem)
7. Boundary conditions and loads (.sim)
8. Launch a linear static analysis (.sim)
9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

5.4 Generation of *.fem* and *.sim* files

1. Go to the “Application” window and click on “Pre/Post”.



2. Click on “New FEM and Simulation” to generate the *.fem* and *.sim* files.



New FEM and
Simulation ▾

5.4 Generation of *.fem* and *.sim* files

3. Choice of the solver and properties:
 - a. Solver: “Samcef”.
 - b. Analysis Type: “Axisymmetric and Other 2D Structural”.
 - c. 2D Solid Option: “XY Plane, Y Axis”.
 - d. Uncheck “Create Idealized Part”.

➤ “OK”.

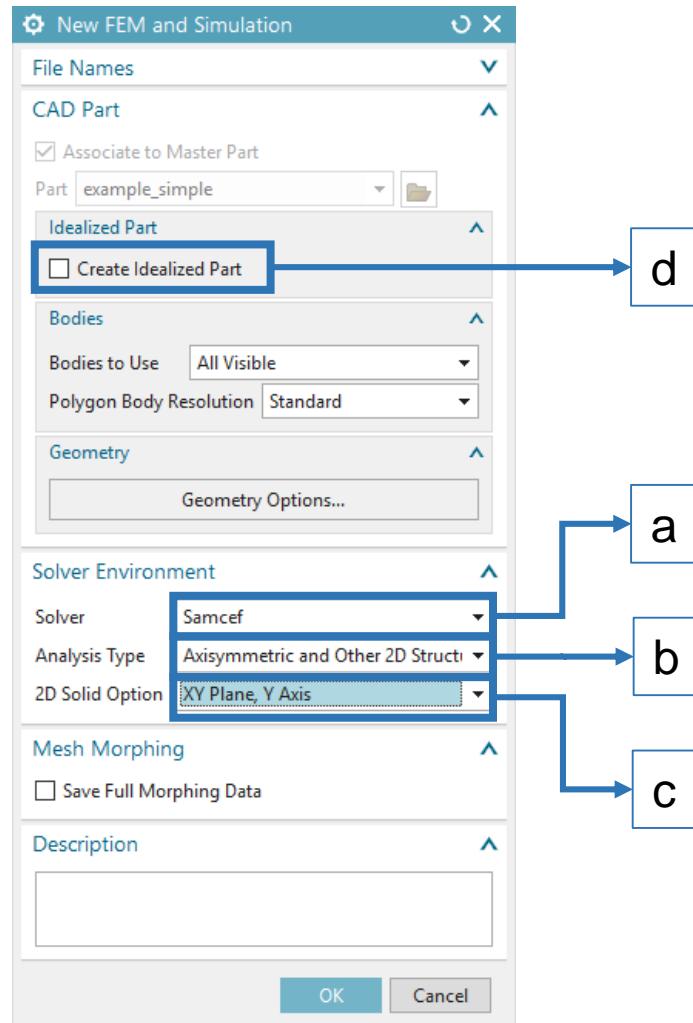


Axisymmetric and 2D structural problems
are defined in the same environment!
Therefore:

\vec{X} = “Radial direction”

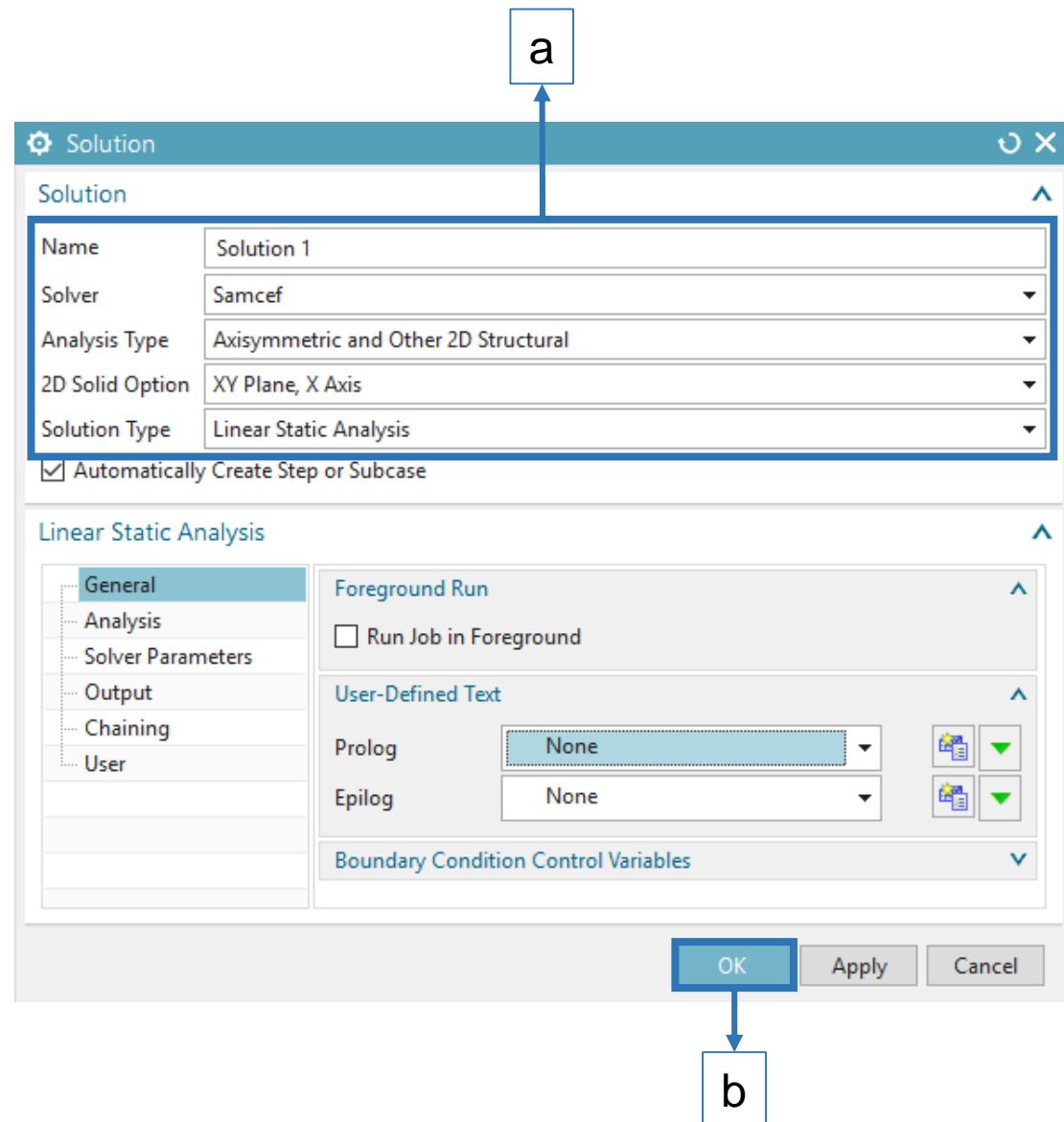
\vec{Y} = “Axial direction”

If XY Plane, Y Axis” has been selected.



5.4 Generation of *.fem* and *.sim* files

4. Created solution:
 - a. Check if everything is set correctly.
 - b. Click on “OK”.



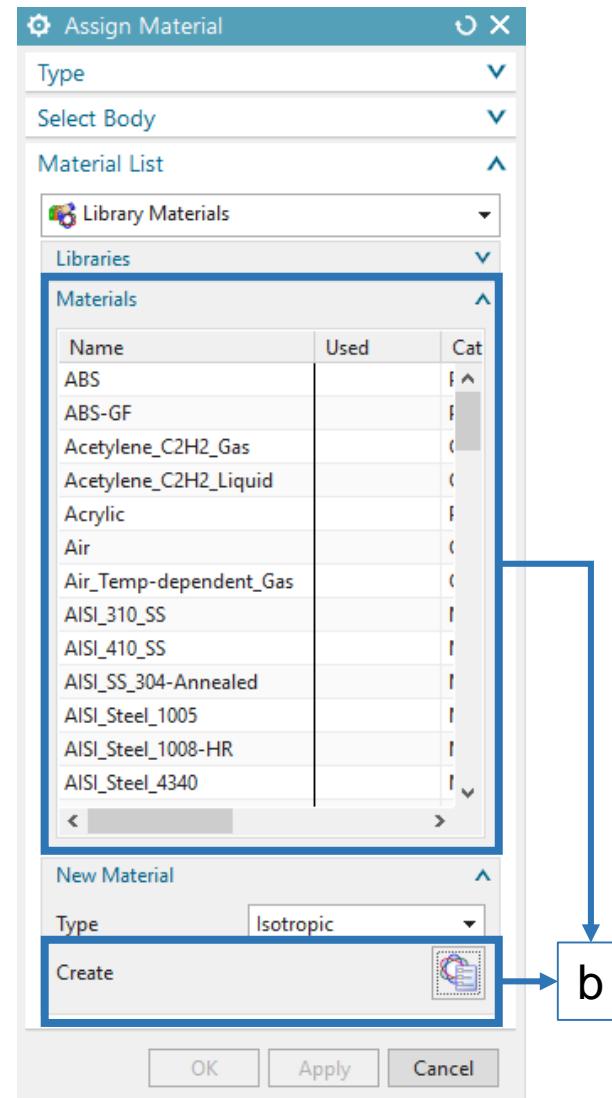
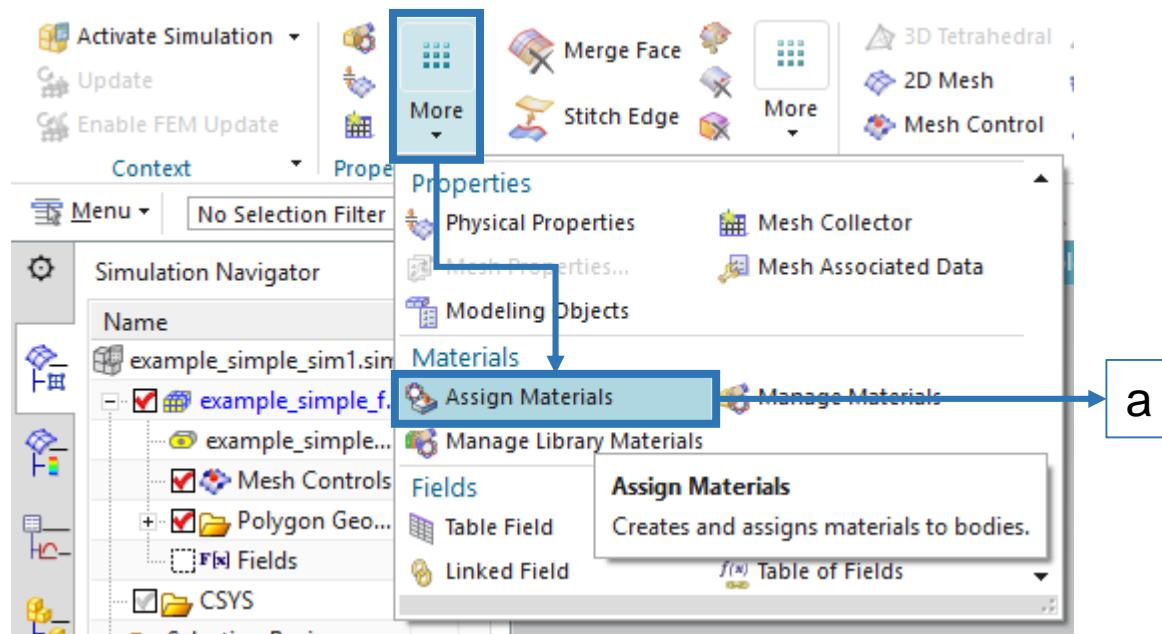
Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

5. Analysis with NX 12

1. Introduction
2. Moving around in NX
3. Geometry drawing (.prt)
4. Generation of .fem and .sim files
- 5. Material properties (.fem)**
6. Mesh generation(.fem)
7. Boundary conditions and loads (.sim)
8. Launch a linear static analysis (.sim)
9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

5.5 Material properties (.fem)



1. Assign mechanical properties to the different materials:
 - a. Click on “More” → “Assign Materials”.
 - b. Select a material from the list or create a new one by clicking on “Create” if needed:
 - Create new materials for this example.

5.5 Material properties (.fem)

- Fill in with the correct properties.



Pay attention
to the units !

Isotropic Material

Property View

All Properties

Name - Description

Element 1

Label 3

Description

Categorization

Properties

Mass Density (RHO) 2700 kg/m³

Mechanical

Strength

Durability

Formability

Thermal/Electrical

Creep

Viscoelasticity

Viscoplasticity

Damage

Miscellaneous

Elastic Constants

Young's Modulus (E) 70000 MPa

Major Poisson's Ratio 0.3

Poisson's Ratio (NU)

Shear Modulus (G)

Structural Damping Coefficient (GE)

Stress-Strain Related Properties

Stress-Strain Input Data Type Engineering Stress-Strain

Stress-Strain (H)

Type of Nonlinearity (TYPE) PLASTIC

Yield Function Criterion (YF) von Mises

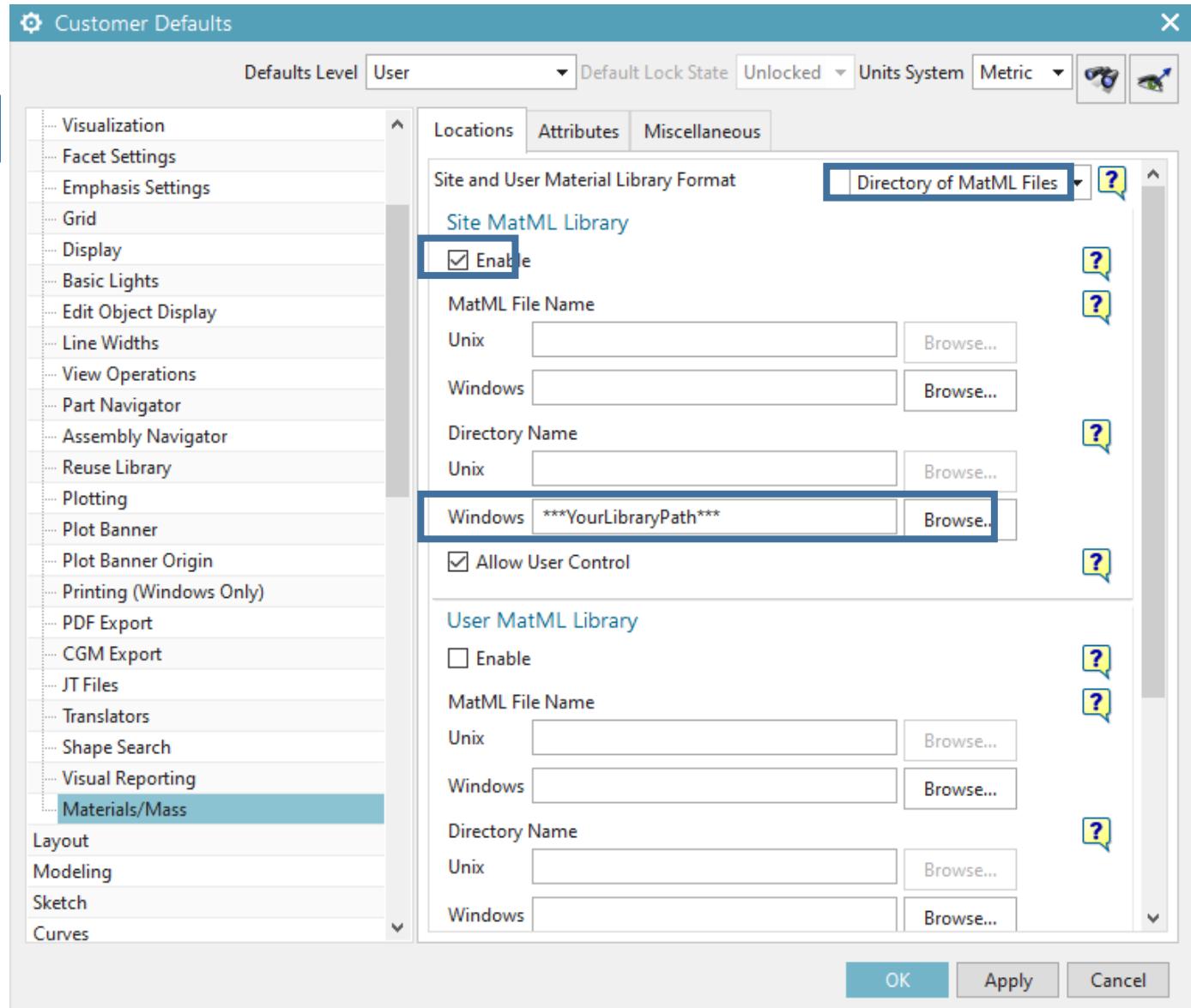
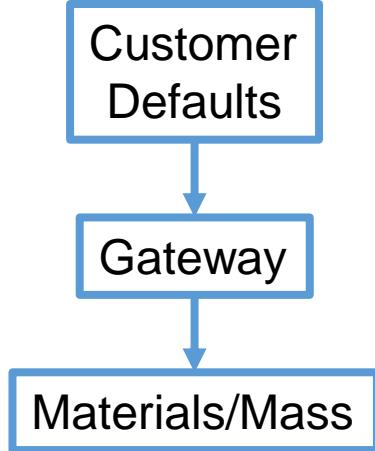
Hardening Rule (HR) Isotropic

Card Name .MAT

OK Cancel

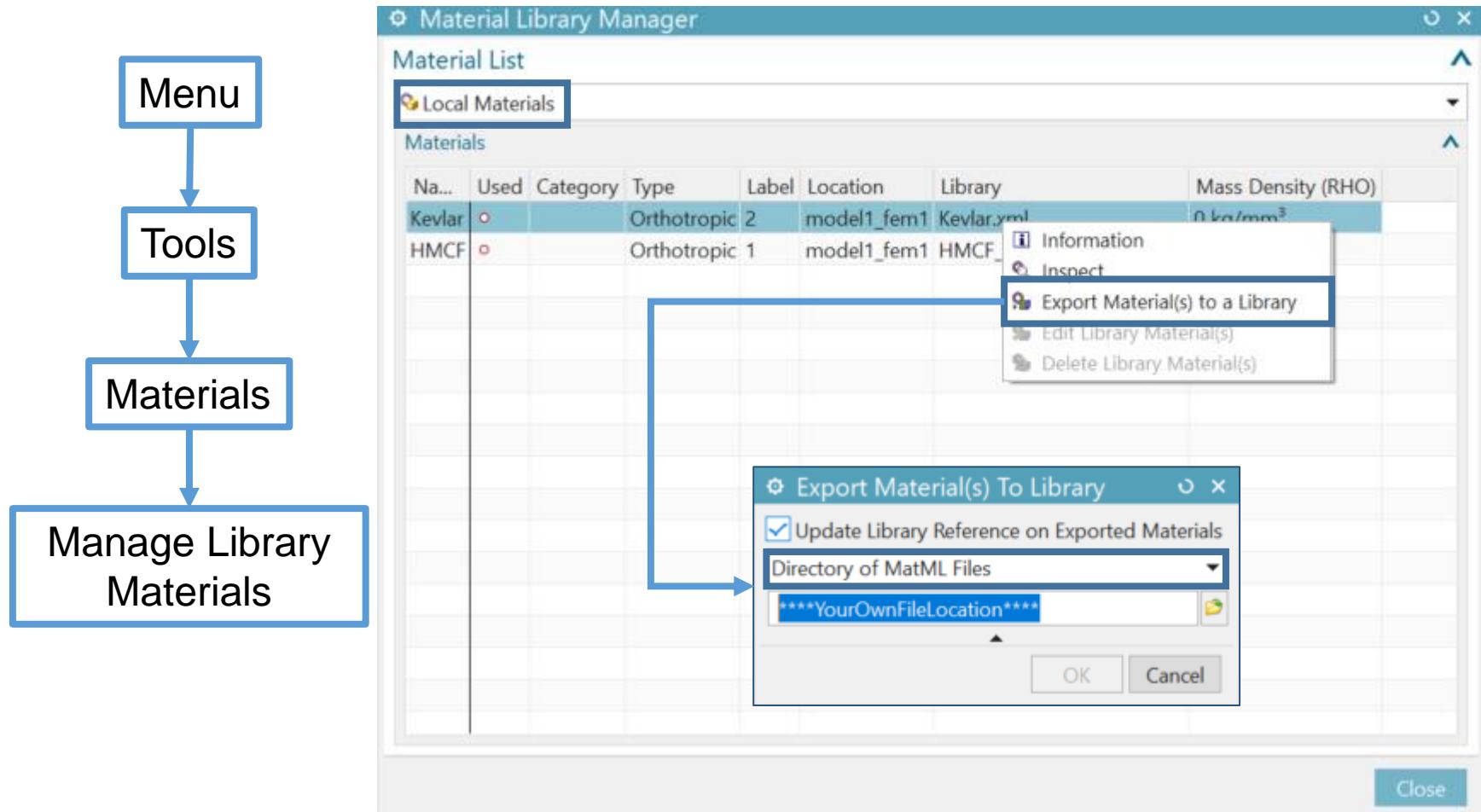
5.5 Material properties (.fem)

Save your materials!



5.5 Material properties (.fem)

Save your own materials as .xml files



5.5 Material properties (.fem)

Manage Materials

Material List

Library Materials

Libraries

Default Material Library

Site MatML Library
YourLibraryPath 

User MatML Library 

Materials

Name	Used	Category	Type	Label	Library
Steel		METAL	Isotropic		physicalmateriallibrary.xml
Sulfur_Dioxide_Liquid		OTHER	Fluid		physicalmateriallibrary.xml
Titanium-Annealed		METAL	Isotropic		physicalmateriallibrary.xml
Titanium_Alloy		METAL	Isotropic		physicalmateriallibrary.xml
Titanium_Ti-6Al-4V		METAL	Isotropic		physicalmateriallibrary.xml
Tungsten		METAL	Isotropic		physicalmateriallibrary.xml
Waspaloy		METAL	Isotropic		physicalmateriallibrary.xml
Water		OTHER	Fluid		physicalmateriallibrary.xml
Water_saturated_Liquid		OTHER	Fluid		physicalmateriallibrary.xml
Water_vapour_Gas		OTHER	Fluid		physicalmateriallibrary.xml
Manten		METAL	Isotropic		physicalmateriallibrary.xml
Carbon-Epoxy			Orthotropic		Carbon epoxy.xml
HMCF			Orthotropic		HMCF.xml
Kevlar			Orthotropic		Kevlar.xml

 52

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

5. Analysis with NX 12

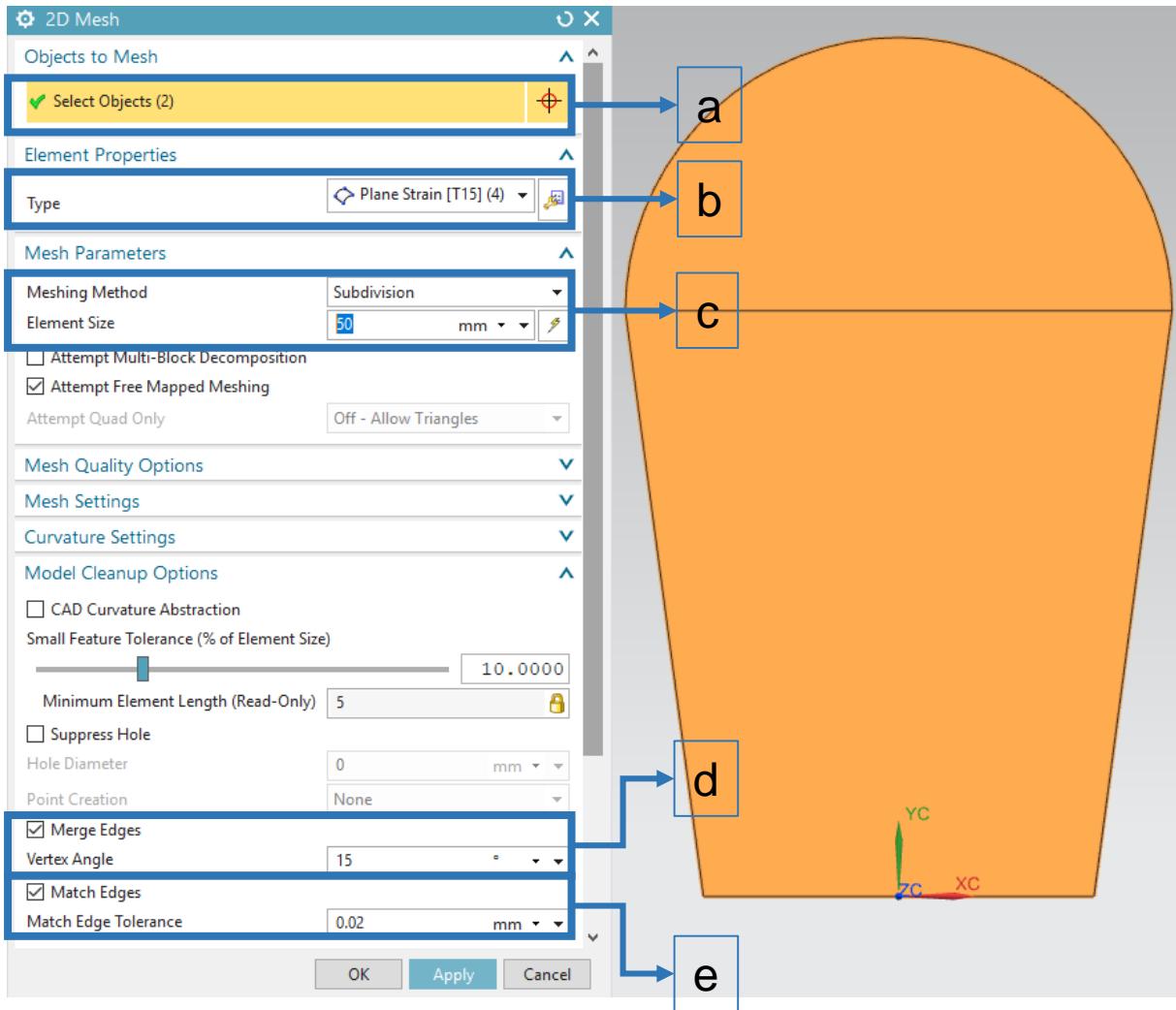
1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 - 6. Mesh generation(.fem)**
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
 7. General remarks
 8. General project instructions

5.6 Mesh generation (.fem)



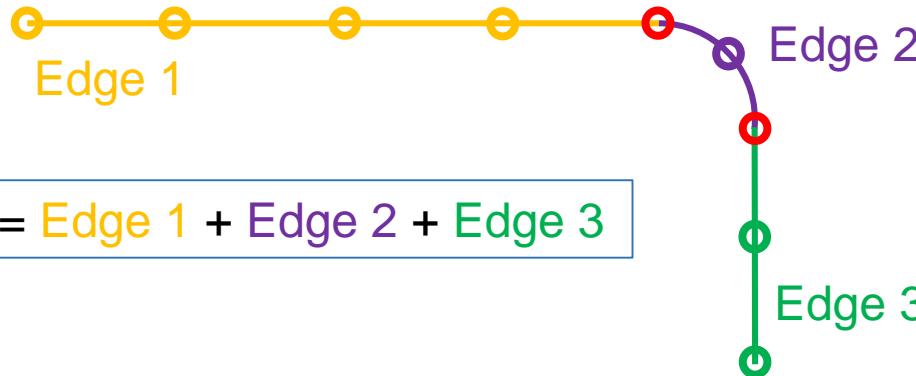
1. Click on “2D Mesh”:
 - a. Select the two faces.
 - b. Select “Plane Strain [T15] (4).
 - c. Element Size of 50 mm.
 - d. Check “Merge Edges”.
 - e. Check “Match Edges”.

➤ “OK”.

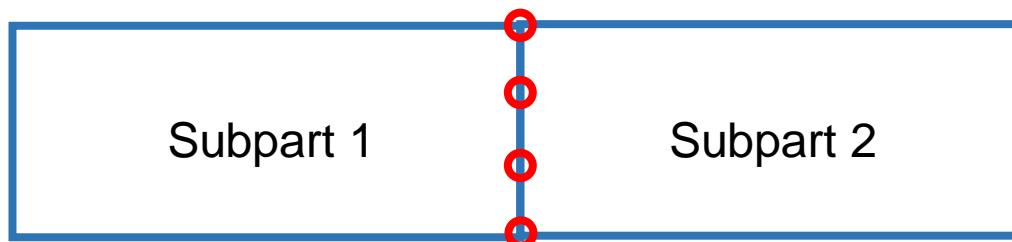


5.6 Mesh generation (.fem)

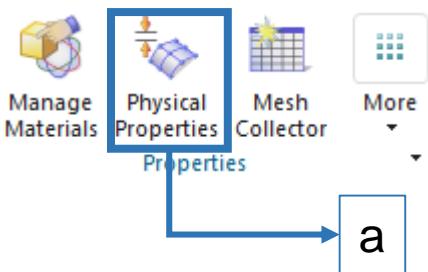
Merge edges: connect every aligned edge together through their **common node** (you can no longer select each edge individually)



Match edges: match nodes of common edges of different subparts of the models (attach different subparts together!)

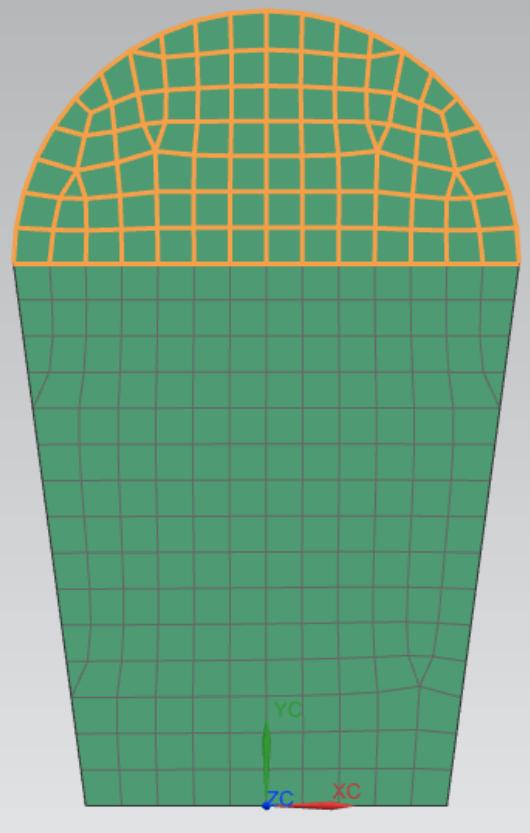
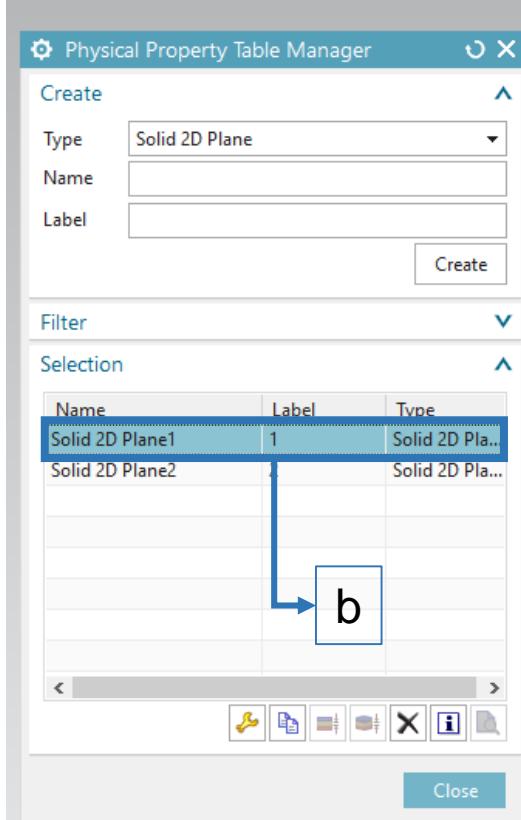


5.6 Mesh generation (.fem)

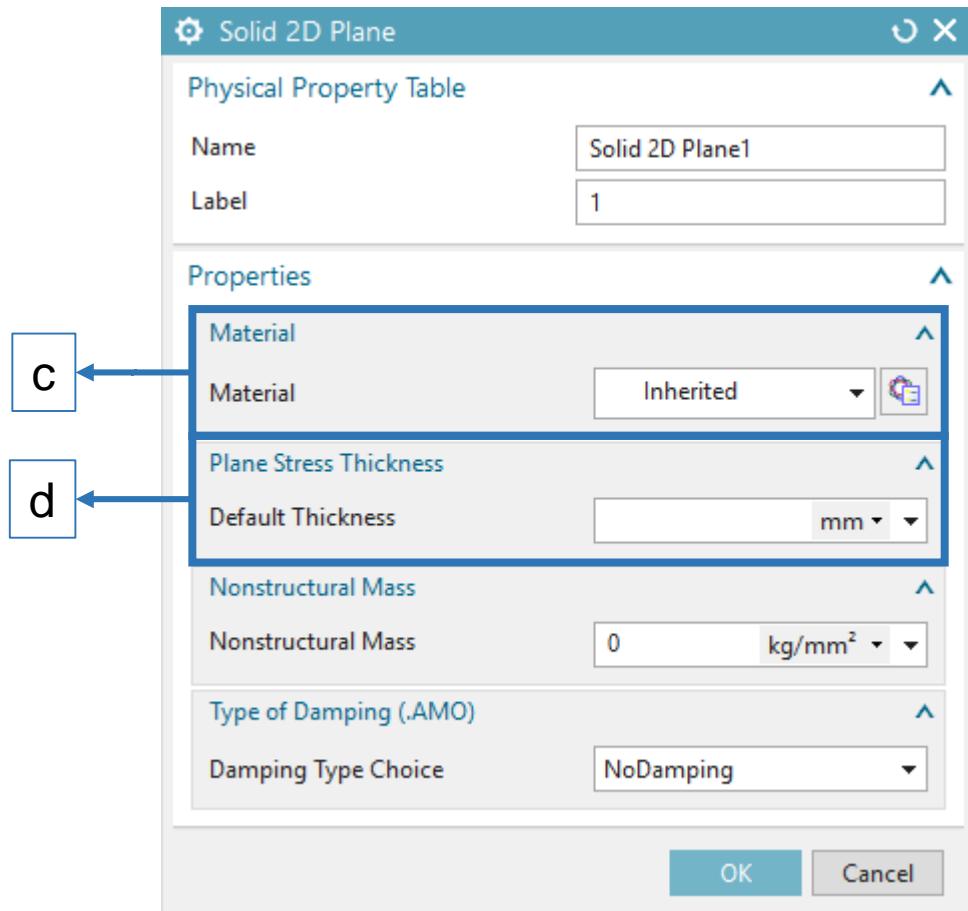


2. Set mesh collector properties:

- Click on “Physical Properties”.
- For each face, double click on its physical property “Solid 2D Plane1” or “Solid 2D Plane2”.
- Set the material properties to “Inherited”.
- Set the “Default Thickness” according to your model for **plane stress** problems.



5.6 Mesh generation (.fem)



Plane strain = “Infinite thickness”.

No need to define a “Default Thickness” for **plane strain** problems. Physical values will be expressed per unit of thickness.

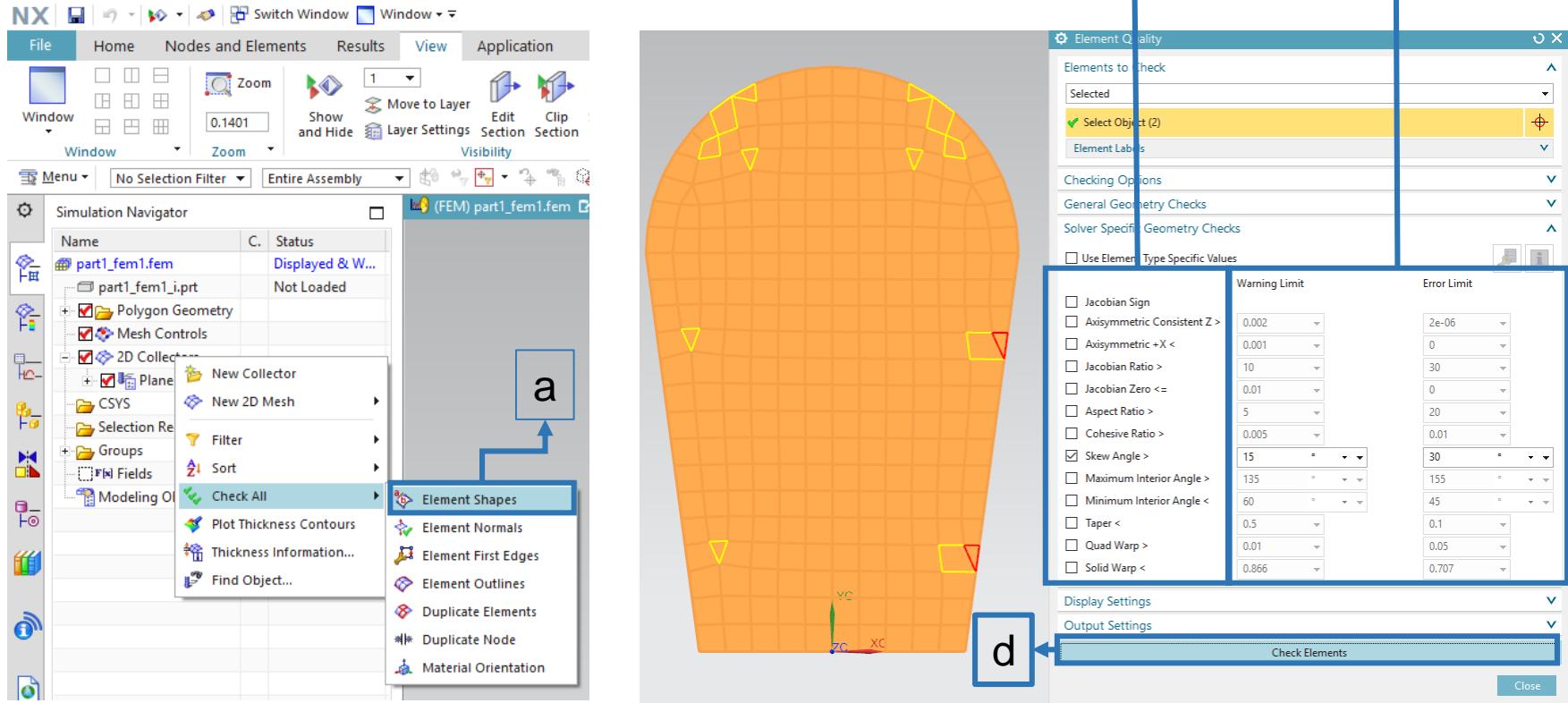
Therefore, leave “Default Thickness” blank.

For **plane stress** problems, you need to define a “Default Thickness”!

5.6 Mesh generation (.fem)

3. Checking the mesh quality, example:

- Right click on “2D Collectors” → “Check All” → “Element Shapes”
- Select the variable(s) to check.
- Specify the “warning” and “error” limits.
- Click on “Check elements”.



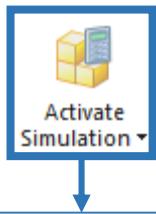
Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

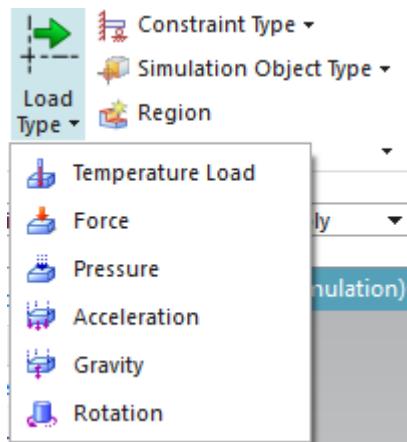
5. Analysis with NX 12

1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 - 7. Boundary conditions and loads (.sim)**
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
 7. General remarks
 8. General project instructions

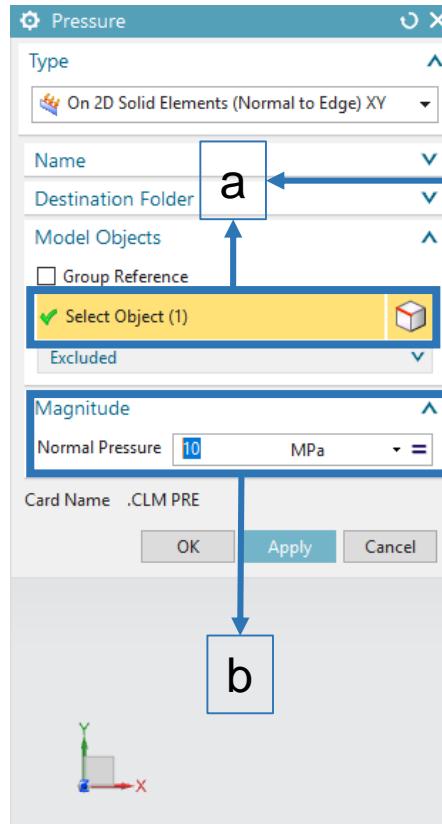
5.7 Boundary conditions and loads (.sim)



1. Click on “Activate Simulation”.



2. Click on “Load Type”
→ “Pressure” or
“Acceleration”.



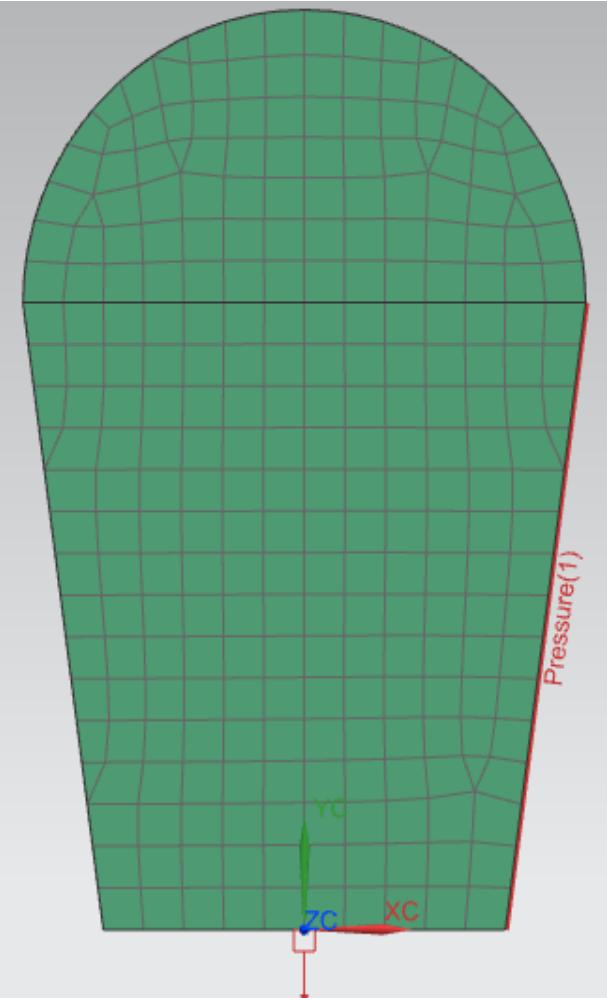
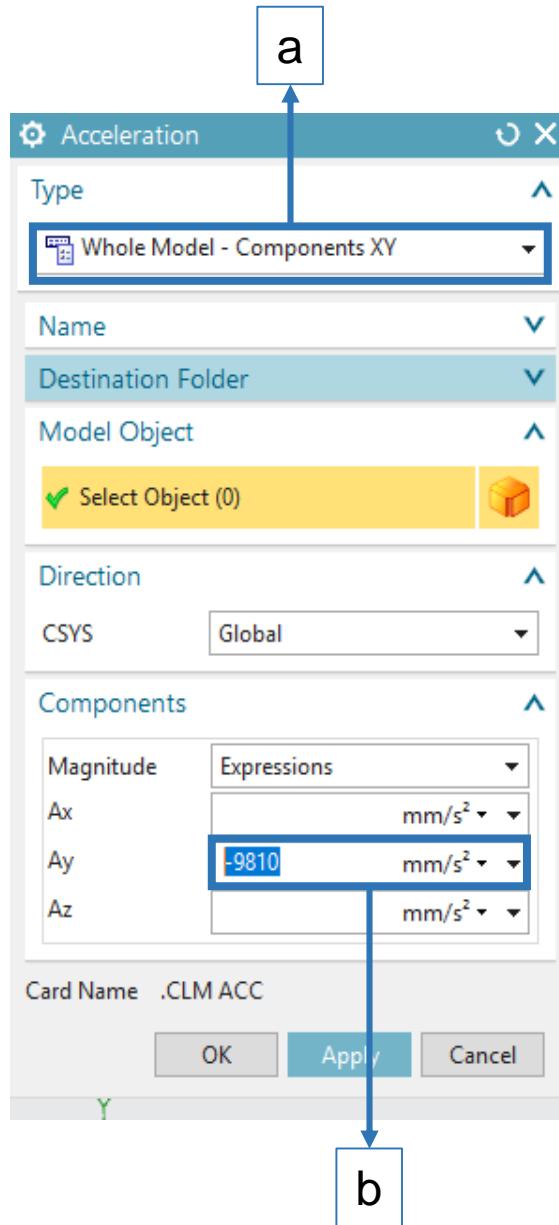
3. Apply the pressure:
a. Select the right edge.
b. Pressure of 10 MPa.

5.7 Boundary conditions and loads (.sim)

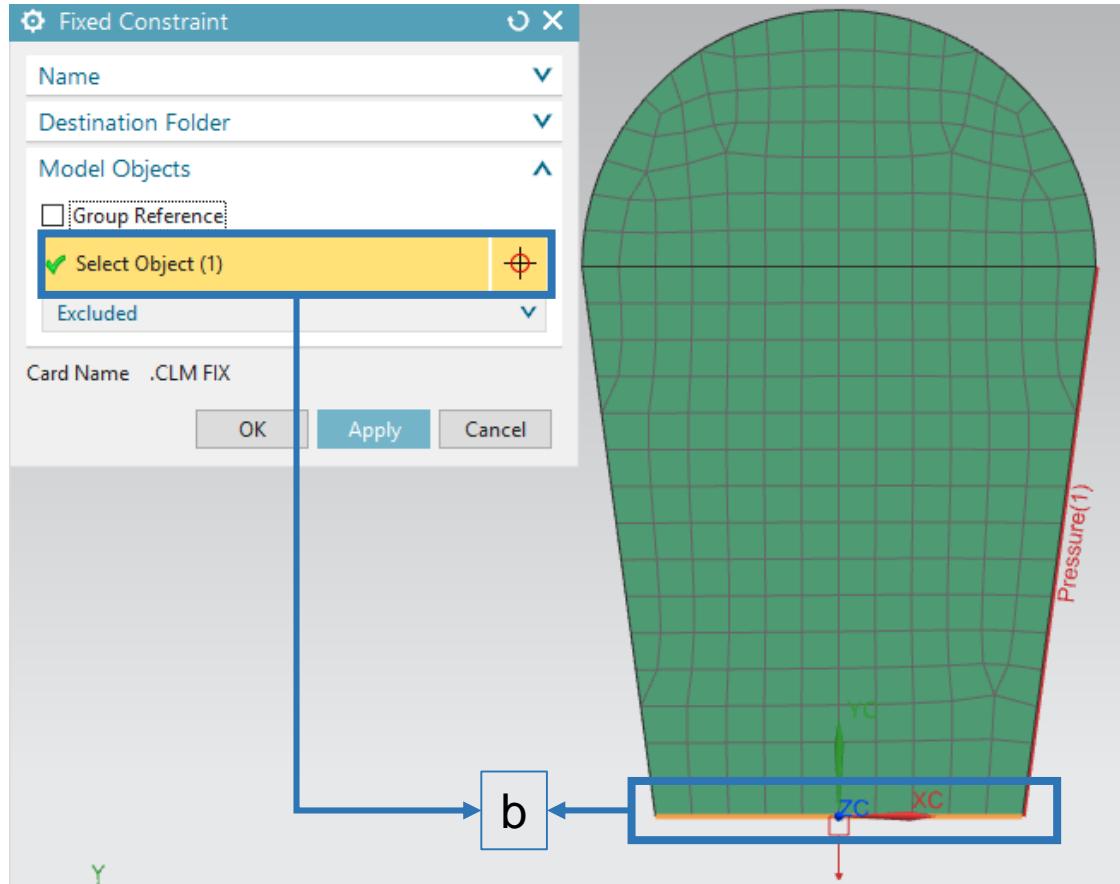
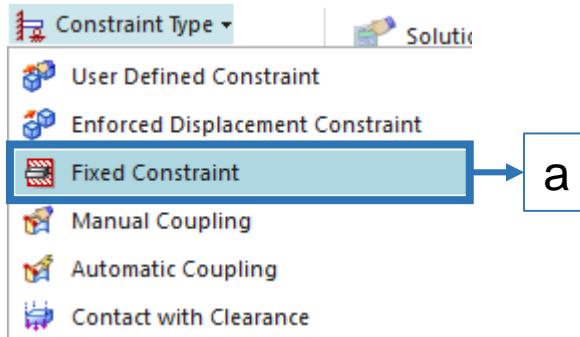
4. Apply the acceleration:
 - a. "Whole Model – Components XY".
 - b. Acceleration: -9810 mm/s².



Units!



5.7 Boundary conditions and loads (.sim)



5. Apply the clamped constraint:
 - a. Click on “Constraint Type” → “Fixed constraint”.
 - b. Select the lower edge.

➤ “OK”.

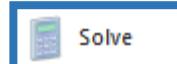
Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

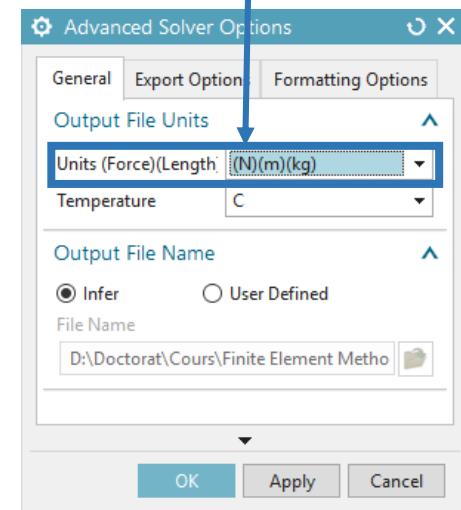
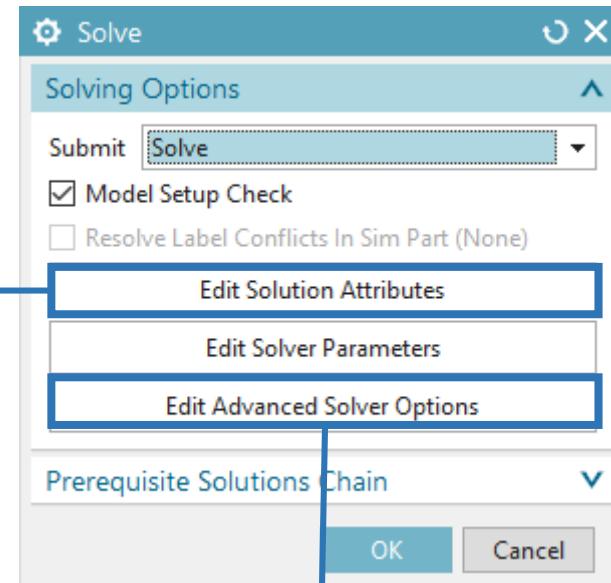
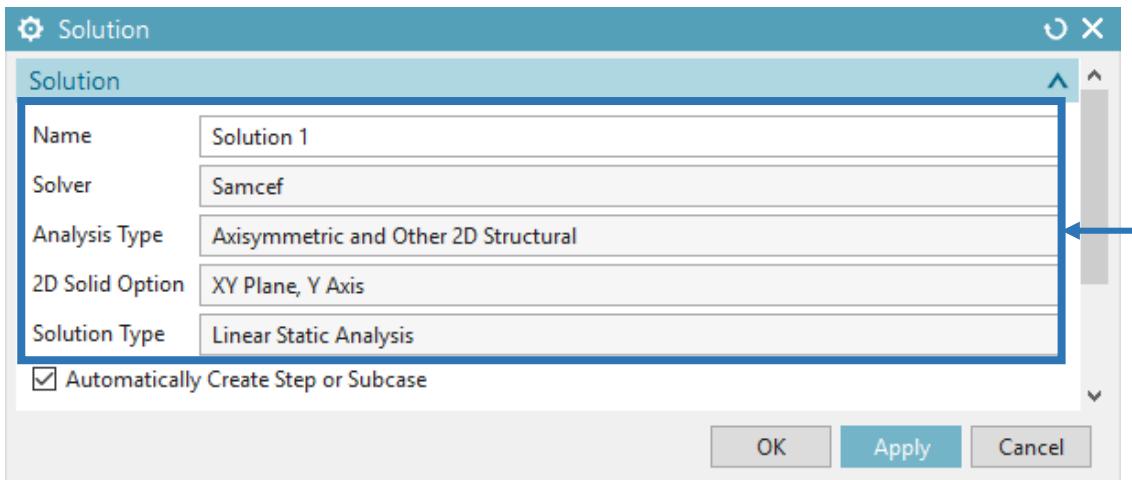
5. Analysis with NX 12

1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 - 8. Launch a linear static analysis (.sim)**
 9. Post-processing of the results
6. Computed results
 1. Singularities
 7. General remarks
 8. General project instructions

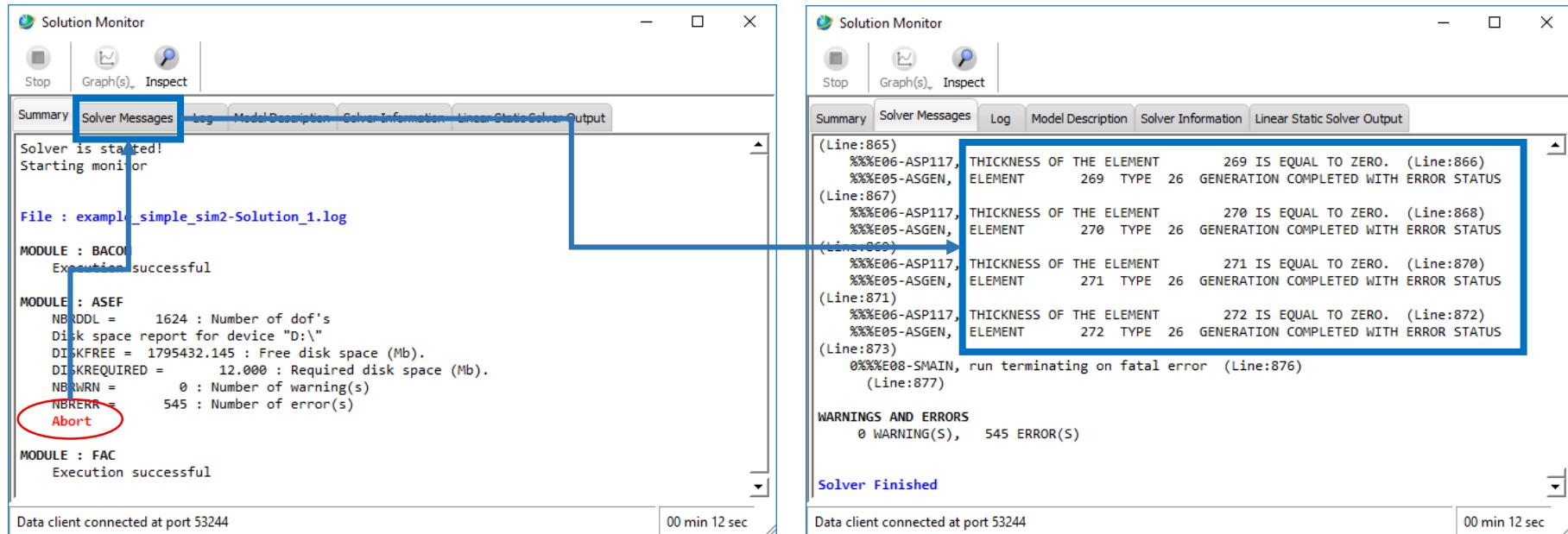
5.8 Launch a Linear Static Analysis (.sim)



1. To Launch the simulation, click on “Solve”.
2. You can check solution and solver parameters:
 - Edit Solution Attributes.
 - Edit Advanced Solver Options:
 - Check units sent to Samcef Solver.
 - “OK”.



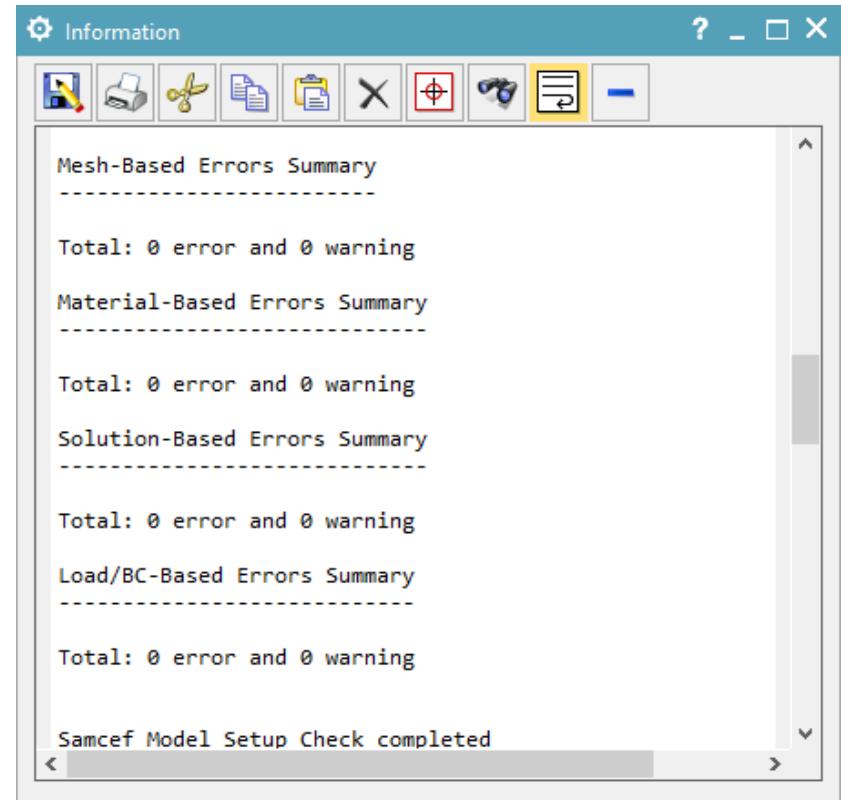
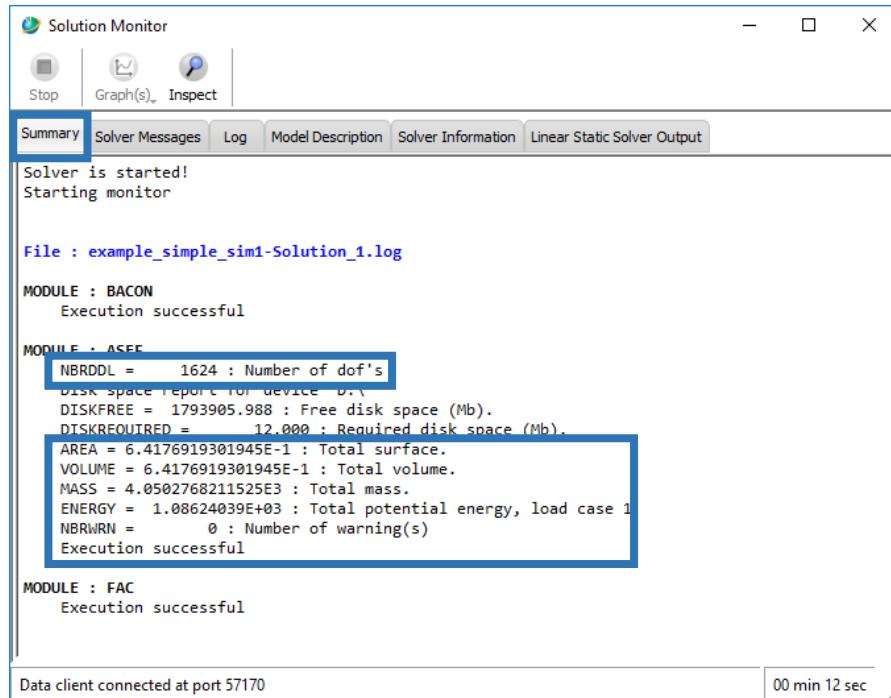
5.8 Launch a Linear Static Analysis (.sim)



Useful summary in the “Solution Monitor” and “Information” windows.

In this example, no “Default Thickness” was defined for a **plane stress** problem: **ERROR !**

5.8 Launch a Linear Static Analysis (.sim)



Useful summary in the “Solution Monitor” and “Information” windows.

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results

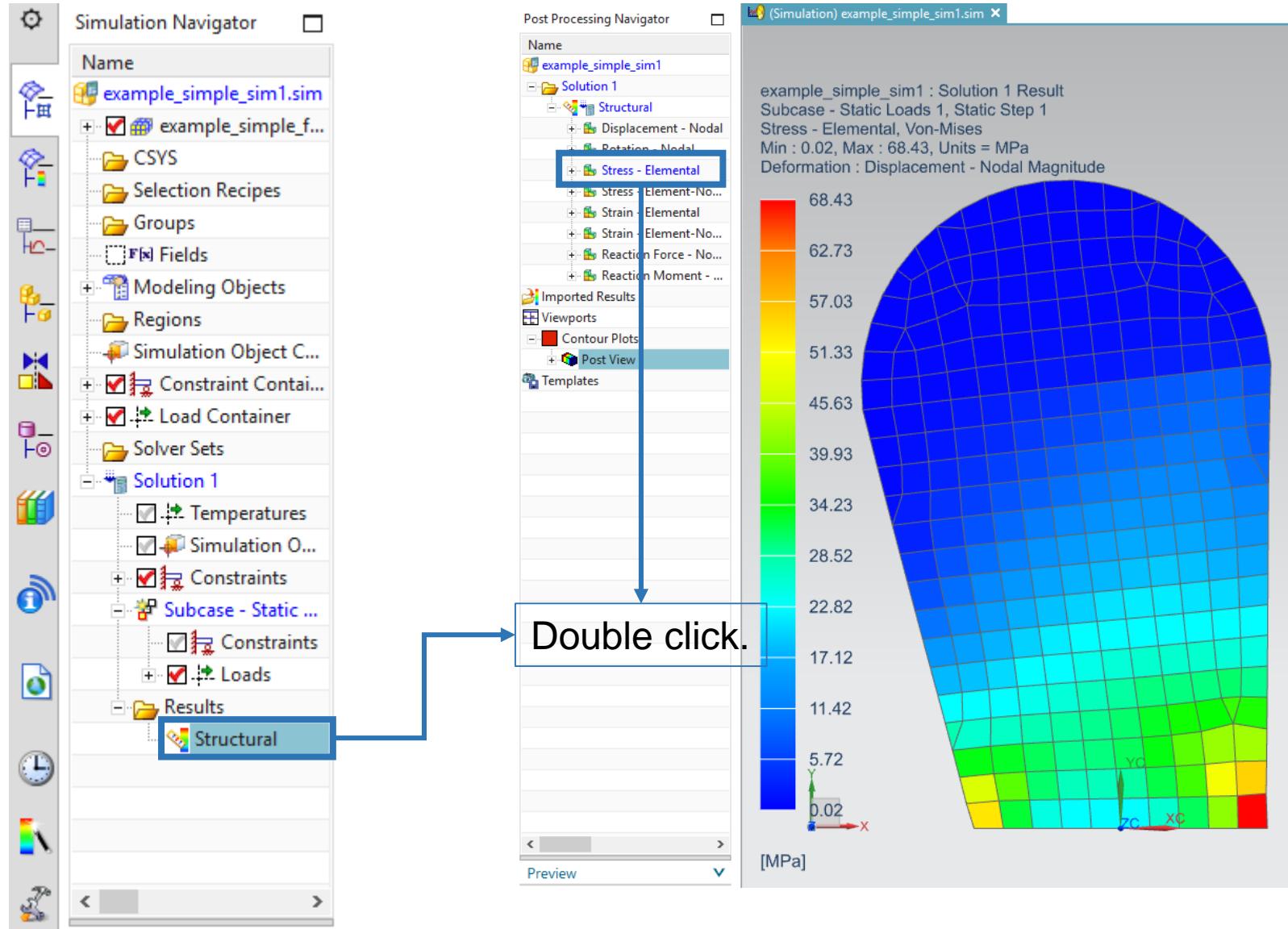
5. Analysis with NX 12

1. Introduction
2. Moving around in NX
3. Geometry drawing (.prt)
4. Generation of .fem and .sim files
5. Material properties (.fem)
6. Mesh generation(.fem)
7. Boundary conditions and loads (.sim)
8. Launch a linear static analysis (.sim)

9. Post-processing of the results

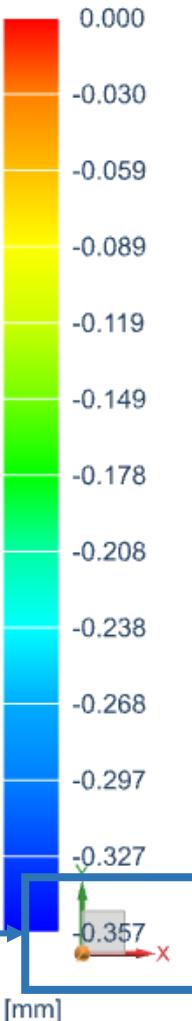
6. Computed results
 1. Singularities
7. General remarks
8. General project instructions

5.9 Post-processing of the results



5.9 Post-processing of the results

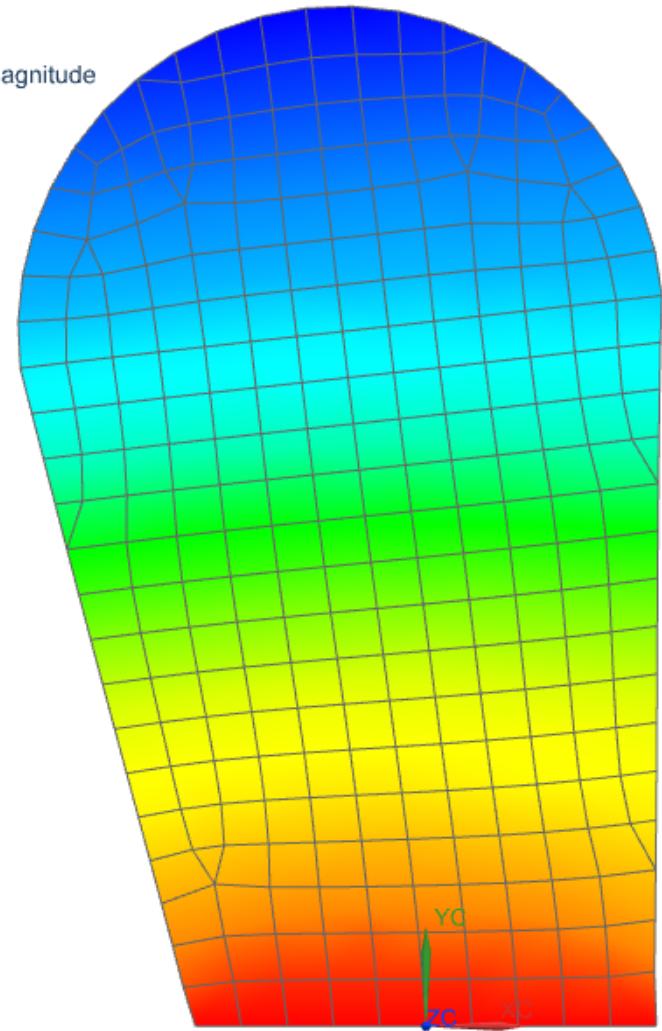
Problem_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, X
Min : -0.357, Max : 0.000, Units = mm
Deformation : Displacement - Nodal Magnitude



Similar results between
strength of materials and finite
element method results!

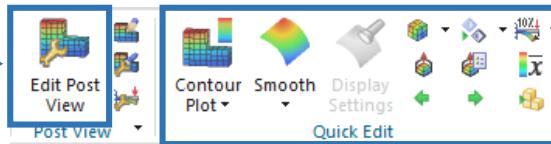
$$v_{max, SOM} = 0.322 \text{ mm}$$

$$v_{max, FEM} = 0.357 \text{ mm}$$



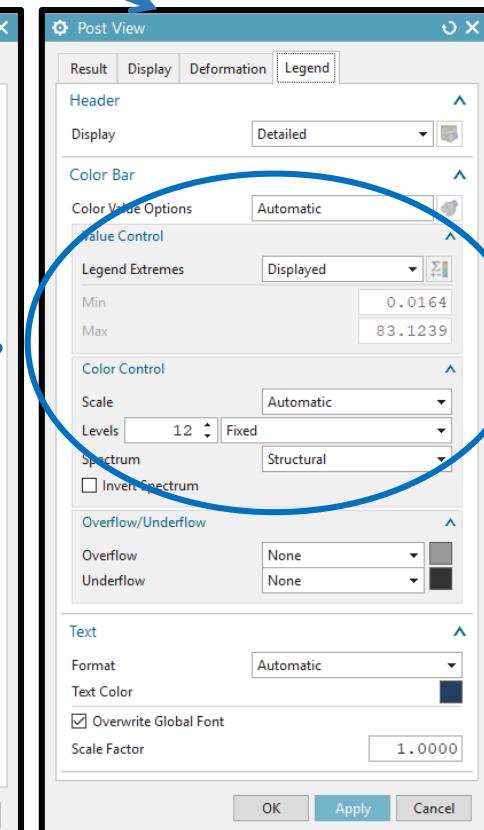
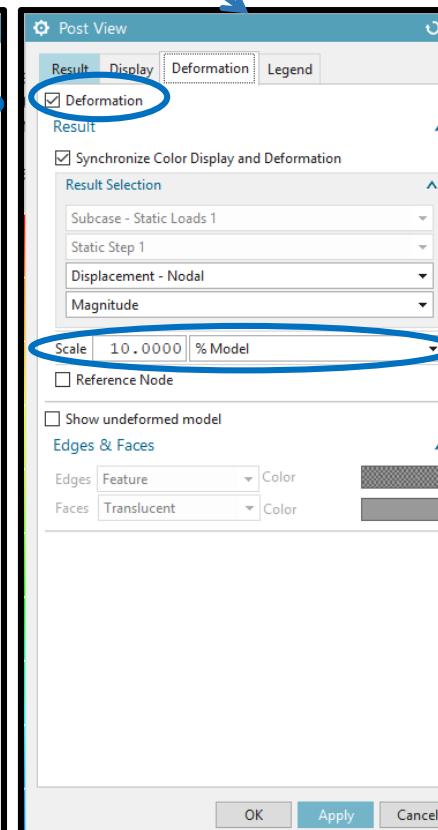
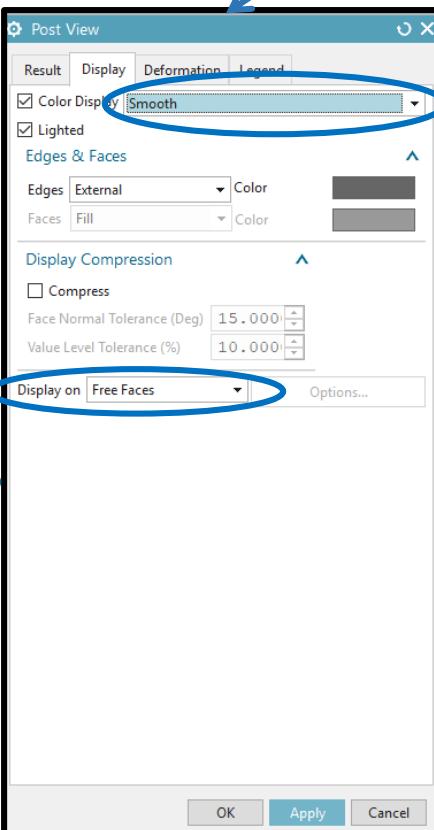
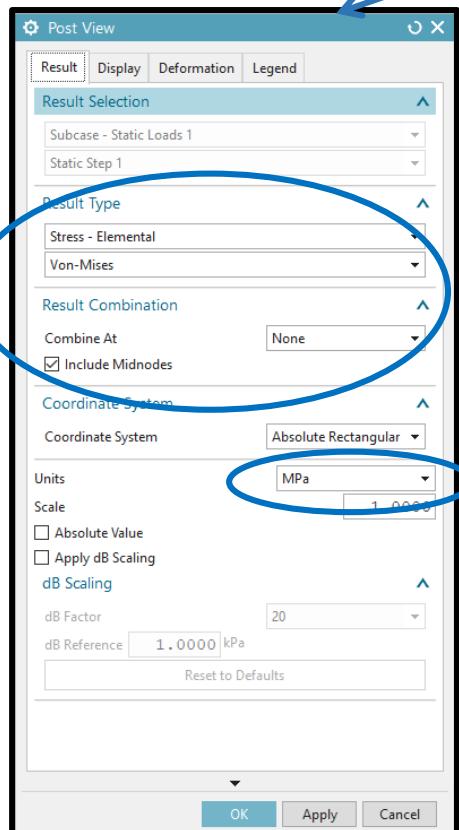
5.9 Post-processing of the results

Edit the post view.



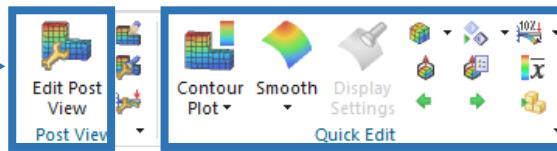
Shortcuts.

Useful commands.

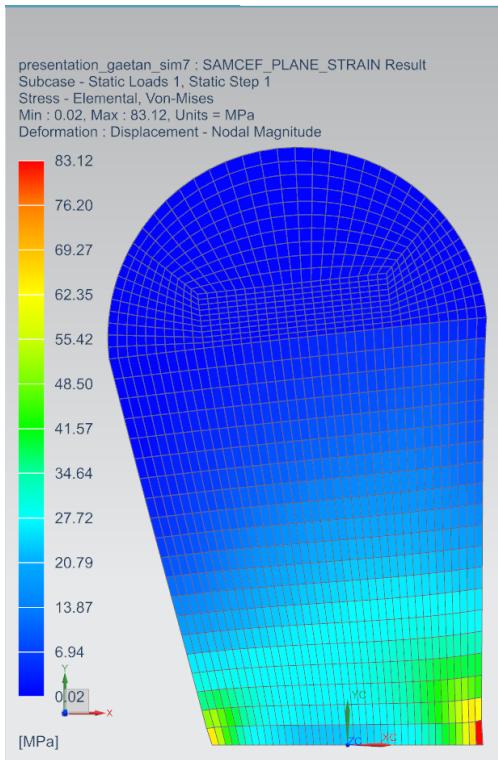


5.9 Post-processing of the results

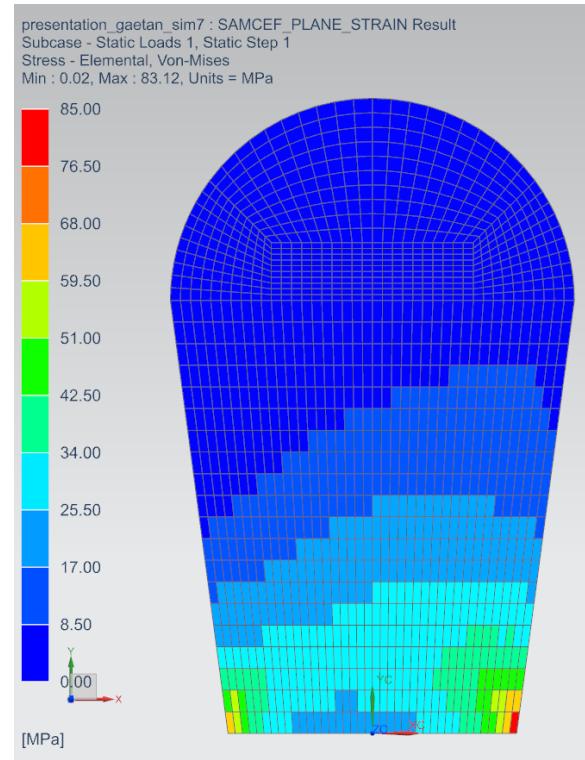
Edit the post view.



Shortcuts.

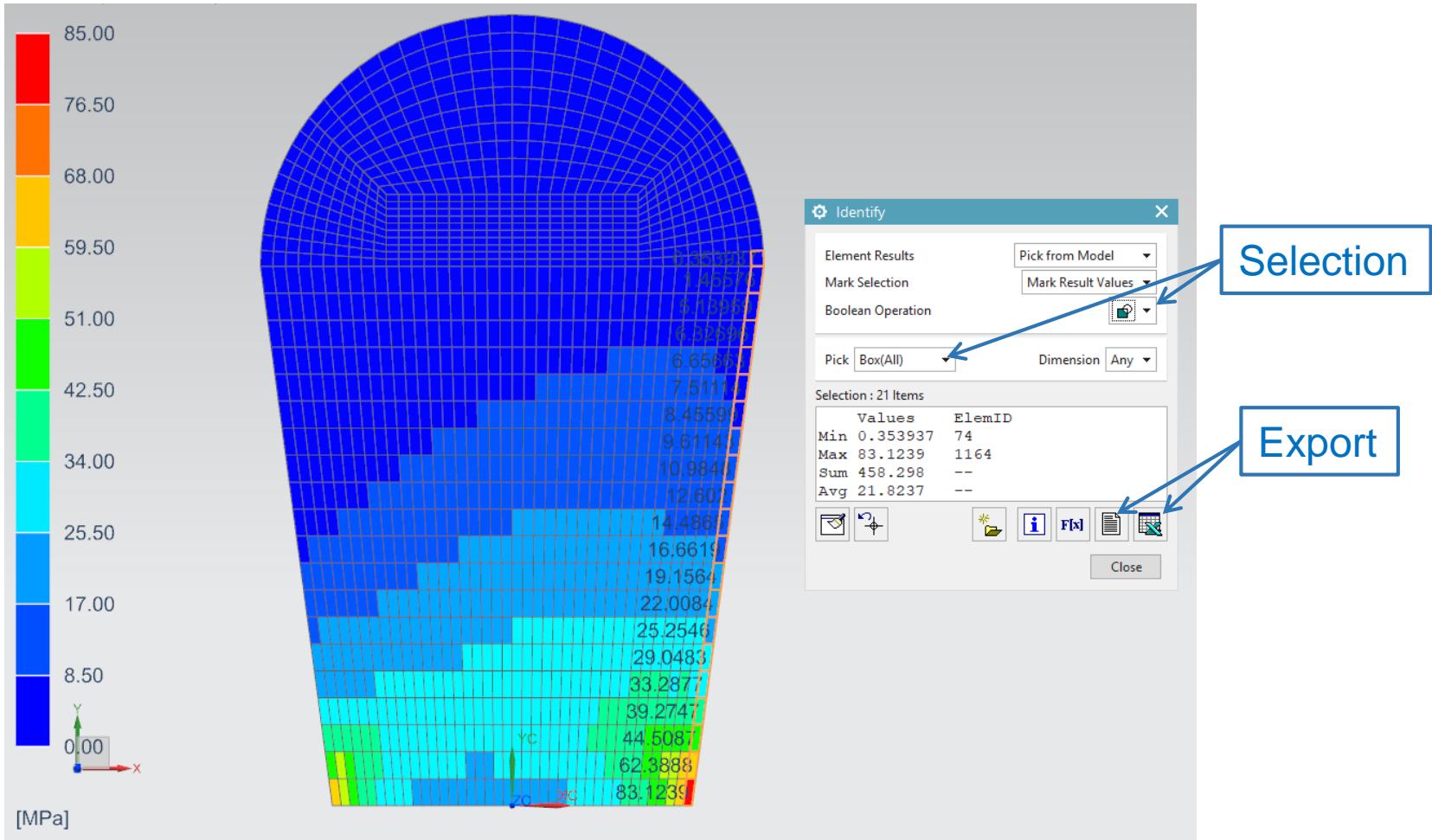
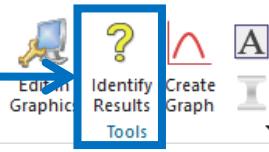


- Example:
- Banded colors.
 - Change of legend.
 - No deformation.



5.9 Post-processing of the results

- Export Data:
 - Identify Results.



5.9 Post-processing of the results

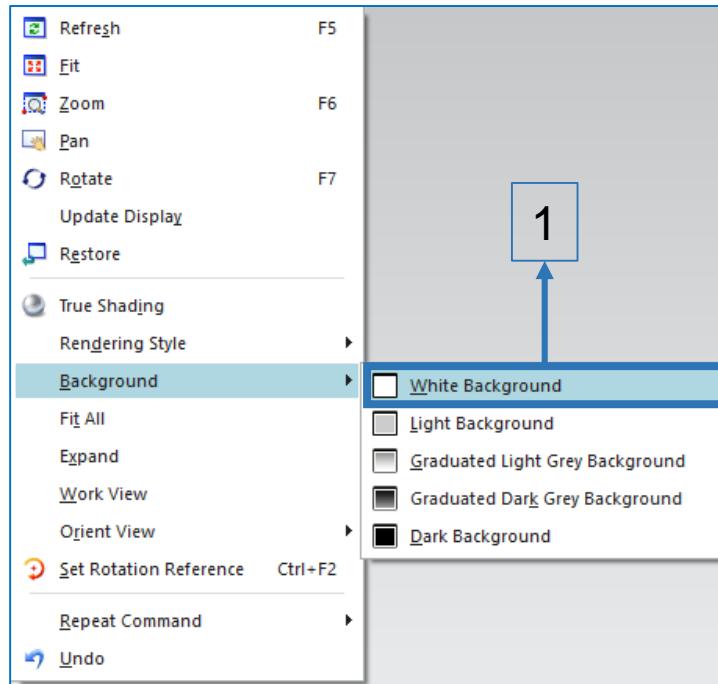
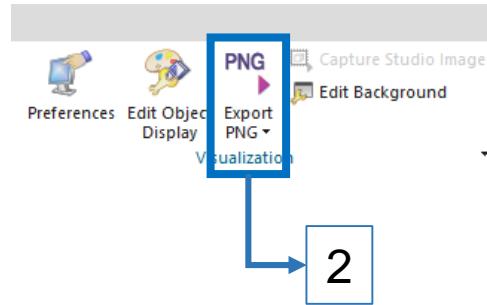
- Export Data:
 - Created .csv file.

Feuille de calcul dans UG PostProcessor Results - presentation_gaetan_sim7.sim															
A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P
1															
2	Solution Name :	presentation_gaetan_sim7 : SAMCEF_PLANE_STRAIN													
3															
4	Load Case :	Subcase - Static Loads 1													
5	Iteration :	Static Step 1													
6	Result :	Stress - Elemental, Unaveraged													
7	Units :	MPa													
8	Coordinate System :	Absolute Rectangular													
9															
10	Elem ID	X Coord	Y Coord	Z Coord	XX	YY	ZZ	XY	YZ	ZX	Von-Mises				
11															
12	74	340,917E+0	760,771E+0	0	349,825E-3	367,382E-3	215,162E-3	-186,606E-3	000,000E+0	000,000E+0	353,937E-3				
13	1145	338,813E+0	731,250E+0	0	-1,621E+0	-1,090E+0	-813,302E-3	733,518E-3	000,000E+0	000,000E+0	1,456E+0				
14	1146	333,938E+0	693,750E+0	0	-8,124E+0	-3,532E+0	-3,497E+0	1,313E+0	000,000E+0	000,000E+0	5,140E+0				
15	1147	329,063E+0	656,250E+0	0	-10,334E+0	-3,880E+0	-4,264E+0	488,853E-3	000,000E+0	000,000E+0	6,327E+0				
16	1148	324,188E+0	618,750E+0	0	-9,856E+0	-2,913E+0	-3,831E+0	737,500E-3	000,000E+0	000,000E+0	6,657E+0				
17	1149	319,313E+0	581,250E+0	0	-9,939E+0	-1,872E+0	-3,543E+0	821,802E-3	000,000E+0	000,000E+0	7,511E+0				
18	1150	314,438E+0	543,750E+0	0	-9,906E+0	-647,863E-3	-3,166E+0	959,533E-3	000,000E+0	000,000E+0	8,456E+0				
19	1151	309,563E+0	506,250E+0	0	-9,895E+0	774,738E-3	-2,736E+0	1,107E+0	000,000E+0	000,000E+0	9,611E+0				
20	1152	304,688E+0	468,750E+0	0	-9,877E+0	2,425E+0	-2,235E+0	1,283E+0	000,000E+0	000,000E+0	10,985E+0				
21	1153	299,813E+0	431,250E+0	0	-9,856E+0	4,331E+0	-1,657E+0	1,489E+0	000,000E+0	000,000E+0	12,602E+0				
22	1154	294,938E+0	393,750E+0	0	-9,832E+0	6,517E+0	-994,332E-3	1,727E+0	000,000E+0	000,000E+0	14,487E+0				
23	1155	290,063E+0	356,250E+0	0	-9,801E+0	9,015E+0	-235,779E-3	2,006E+0	000,000E+0	000,000E+0	16,662E+0				
24	1156	285,188E+0	318,750E+0	0	-9,771E+0	11,853E+0	624,551E-3	2,317E+0	000,000E+0	000,000E+0	19,156E+0				
25	1157	280,313E+0	281,250E+0	0	-9,725E+0	15,079E+0	1,606E+0	2,695E+0	000,000E+0	000,000E+0	22,008E+0				
26	1158	275,438E+0	243,750E+0	0	-9,691E+0	18,742E+0	2,715E+0	3,065E+0	000,000E+0	000,000E+0	25,255E+0				
27	1159	270,563E+0	206,250E+0	0	-9,647E+0	22,953E+0	3,992E+0	3,634E+0	000,000E+0	000,000E+0	29,048E+0				
28	1160	265,688E+0	168,750E+0	0	-9,490E+0	27,913E+0	5,527E+0	3,883E+0	000,000E+0	000,000E+0	33,288E+0				
29	1161	260,813E+0	131,250E+0	0	-10,053E+0	33,750E+0	7,109E+0	5,196E+0	000,000E+0	000,000E+0	39,275E+0				
30	1162	255,938E+0	93,750E+0	0	-7,378E+0	42,783E+0	10,622E+0	3,821E+0	000,000E+0	000,000E+0	44,509E+0				
31	1163	251,063E+0	56,250E+0	0	-18,004E+0	50,570E+0	9,770E+0	10,379E+0	000,000E+0	000,000E+0	62,389E+0				
32	1164	246,188E+0	18,750E+0	0	18,543E+0	107,835E+0	37,913E+0	-9,849E+0	000,000E+0	000,000E+0	83,124E+0				
33	Minimum				-18,004E+0	-3,880E+0	-4,264E+0	-9,849E+0	000,000E+0	000,000E+0	353,937E-3				
34	Elem ID				1163	1147	1147	1164	74	74	74				
35	Maximum				18,543E+0	107,835E+0	37,913E+0	10,379E+0	000,000E+0	000,000E+0	83,124E+0				
36	Elem ID				1164	1164	1164	1163	74	74	1164				
37	Column Sum				-163,905E+0	340,974E+0	53,121E+0	37,619E+0	000,000E+0	000,000E+0	458,298E+0				
38	Column Avg				-7,805E+0	16,237E+0	2,530E+0	1,791E+0	000,000E+0	000,000E+0	21,824E+0				
39															
40															
41															

5.9 Post-processing of the results

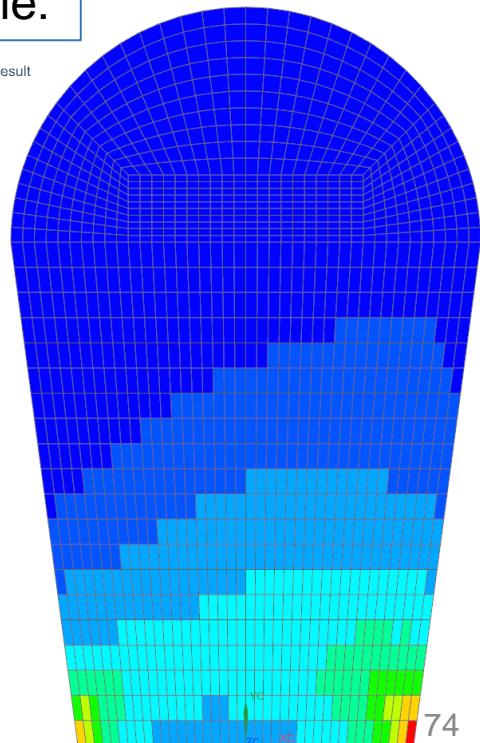
Export results to .PNG format:

1. Right click on the background
→ “Background” → “White Background”.
2. Export the view: → “View” → “Export to PNG”.



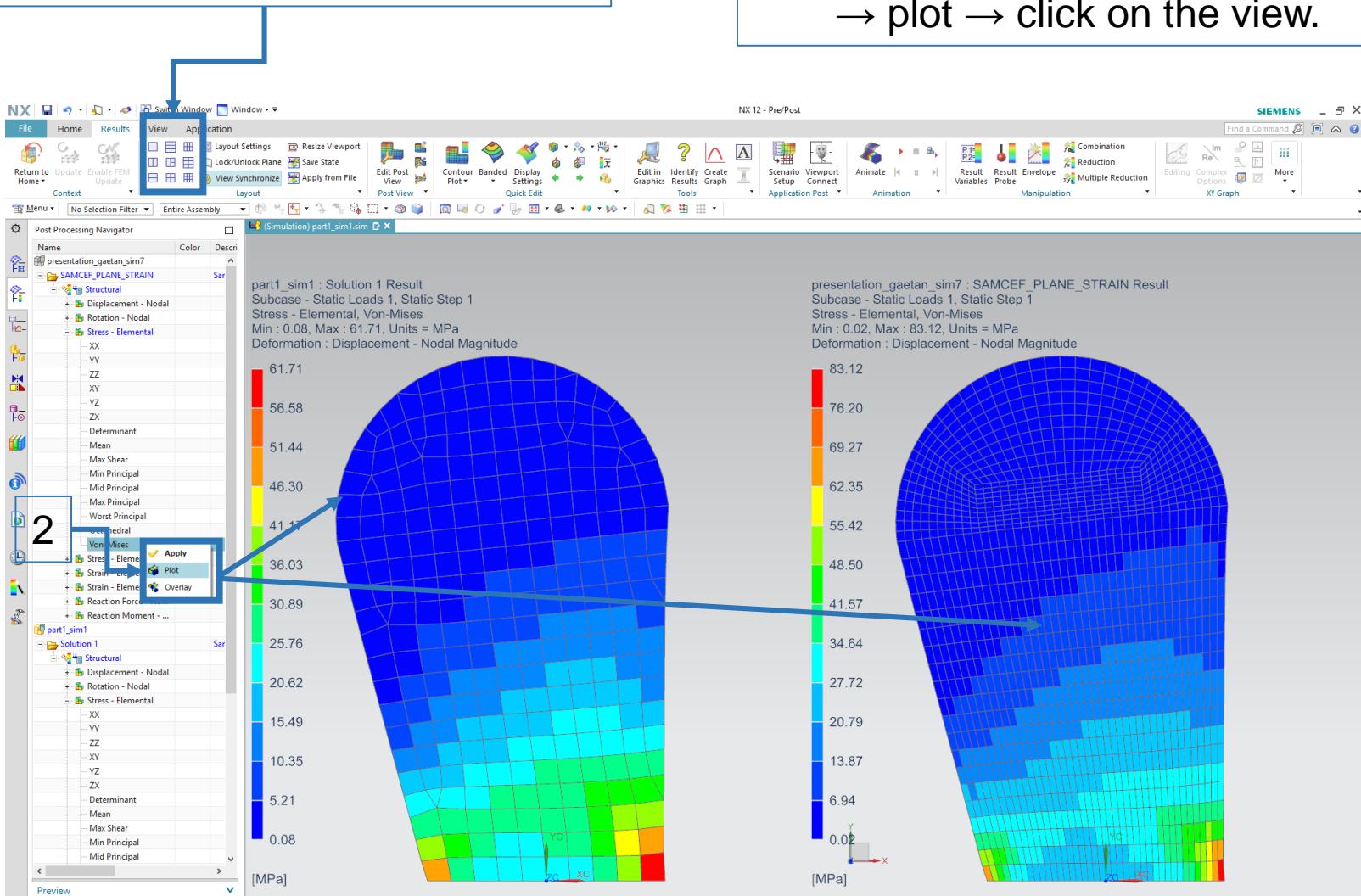
Resulting PNG file:

presentation_gaelan_sim7 : SAMCEF_PLANE_STRAIN Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.02, Max : 83.12, Units = MPa



5.9 Post-processing of the results

1. Use of Multiple View Layout.



2. Right click on wanted results → plot → click on the view.

5.9 Post-processing of the results

Files generated in the working directory:

.prt, .fem & .sim files.

.dat file generated by NX and that contains all the information relative to the model (mesh, mechanical properties, loads applied, etc.).

.res file generated by the SAMCEF solver based on the .dat file and that contains the solution of the analysis.

bacon	17/01/2018 14:58	Text Document	1 KB
example_simple	11/01/2018 17:38	Siemens Part File	98 KB
example_simple_fem1	12/01/2018 16:55	Simcenter FEM File	1,148 KB
example_simple_sim1	12/01/2018 16:58	Simcenter SIM File	2,394 KB
example_simple_sim1-Solution_1.dat	17/01/2018 14:58	DAT File	30 KB
example_simple_sim1-Solution_1	17/01/2018 14:58	Text Document	50 KB
example_simple_sim1-Solution_1.sam	17/01/2018 14:58	SAM File	55 KB
example_simple_sim1-Solution_1.sdb	17/01/2018 14:58	SDB File	1,287 KB
example_simple_sim1-Solution_1.sdb#1	17/01/2018 14:58	SDB#1 File	1,080 KB
example_simple_sim1-Solution_1.spy	17/01/2018 14:58	SPY File	2 KB
example_simple_sim1-Solution_1_as.des	17/01/2018 14:58	DES File	5 KB
example_simple_sim1-Solution_1_as.fac	17/01/2018 14:58	FAC File	88 KB
example_simple_sim1-Solution_1_as.res	17/01/2018 14:58	RES File	23 KB
example_simple_sim1-Solution_1_as.u18	17/01/2018 14:58	U18 File	456 KB
example_simple_sim1-Solution_1_sr.w80	17/01/2018 15:01	W80 File	0 KB
example_simple_sim1-Solution_1_srap1	17/01/2018 15:01	Text Document	4 KB



A lot of useful information is contained in these files, it is recommended to take a look at them to see how they are structured and what you can deduce from them.

5.9 Post-processing of the results

.dat file

```
! Samcef Bank File: UNITS.  
!  
.UNIT  
LENGTH 1 ! Unit = Meter  
MASS 1 ! Unit = Kilogram  
CELSIUS  
TIME 1
```

Units system
used by the
Samcef Solver.

```
! Samcef Bank File: MATERIALS.  
!  
.MAT  
! Aluminium::example_simple_feml::[1]  
I 1 BEHA "ELASTIC"  
YT 7e+10 ! Young's modulus (E)  
NT 0.3 ! Poisson's ratio (NU)  
M 2700 ! Mass Density (Rho)  
  
! Steel::example_simple_feml::[2]  
I 2 BEHA "ELASTIC"  
YT 2.05e+11 ! Young's modulus (E)  
NT 0.3 ! Poisson's ratio (NU)  
M 7850 ! Mass Density (Rho)  
  
! Samcef Bank File: HYPOTHESES  
!  
.HYP DEFO PLAN GROUP "2d_mesh(1)_0"  
.HYP DEFO PLAN GROUP "2d_mesh(2)_1"  
! Samcef Bank File: CONSTRAINTS.  
!
```

Material properties.

```
.CLM ADD  
!  
! Fixed(1)  
GROUP "Fixed(1)" FIX C 1 2 3 4 5 6  
  
! Samcef Bank File: LOADS.  
!  
.CLM ADD  
STRU ACC VAL 0 -9.81 0  
  
ARETE I 182 3 PRE V 1e+07  
ARETE I 183 3 PRE V 1e+07  
ARETE I 186 3 PRE V 1e+07  
ARETE I 187 3 PRE V 1e+07  
ARETE I 190 1 PRE V 1e+07  
ARETE I 191 1 PRE V 1e+07  
ARETE I 192 1 PRE V 1e+07  
ARETE I 193 1 PRE V 1e+07  
ARETE I 214 3 PRE V 1e+07  
ARETE I 215 3 PRE V 1e+07  
ARETE I 216 3 PRE V 1e+07  
ARETE I 219 3 PRE V 1e+07  
ARETE I 220 1 PRE V 1e+07  
ARETE I 221 1 PRE V 1e+07  
ARETE I 272 1 PRE V 1e+07
```

Loads and constraints.

5.9 Post-processing of the results

.res file

```
STRUCTURE CHARACTERISTICS
=====
TOTAL LENGTH      0.000000E+00
TOTAL AREA        6.417692E-01
TOTAL VOLUME       6.417692E-01
TOTAL MASS         4.050277E+03
```

```
END OF STRESS STORAGE ON U18 FILE
CPU= 0 H 0 Min 1.27 Sec.    Elapsed= 0 H 0 Min 2.00 Sec.
```

Computation time

Model characteristics

CHARACTERISTICS OF THIS PROBLEM

NUMBER OF NODES	299
MAXIMUM NODE NUMBER	299
MINIMUM NODE NUMBER	1
NUMBER OF INTERFACES	570
NUMBER OF LOCAL AXIS SYSTEMS	0
TOTAL NUMBER OF ELEMENTS	272
MAXIMUM ELEMENT NUMBER	272
NUMBER OF GENERAL PROPERTY ENTRIES	0
NUMBER OF ELEMENT PROPERTY ENTRIES	0
NUMBER OF ELEMENT ATTRIBUTE ENTRIES	0
NUMBER OF ELEMENTS OF TYPE	15
NUMBER OF ELEMENTS OF TYPE	26

```
TOTAL POTENTIAL ENERGY 1.086240E+03
```

TPE

```
ODIAGNOSTICS: 0 WARNING(S), 0 ERROR(S)
```

Status of
the analysis

Mesh characteristics

5.9 Post-processing of the results

BASE UNITS			DERIVED UNITS				
LENGTH	MASS	TIME	FORCE	STRESS	ENERGY	VELOCITY	ACCELER.
m	kg	s	N	Pa	J	m/s	m/s^2
mm	kg	s	mN	kPa	$10^{-3} mJ$	$10^{-3} m/s$	$10^{-3} m/s^2$
mm	<i>tonne</i>	s	N	MPa	mJ	$10^{-3} m/s$	$10^{-3} m/s^2$



Results in the .res and .dat files are expressed in consistent units !

If you are in plane strain, results are expressed per 1 unit of length (i.e. 1mm or 1m)!

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
- 6. Computed results**
 1. Singularities
7. General remarks
8. General project instructions

6. Computed results

- In general, the equations are **not exactly** satisfied:

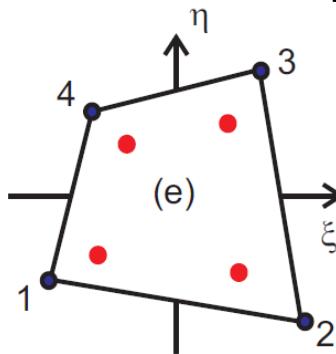
$$\frac{\partial \sigma_{ij}^{F.E.}}{\partial x_j} + \rho_0 \bar{b}_i \neq 0$$
$$\sigma_{ij}^{F.E.} n_j - \bar{t}_i \neq 0$$

- The finite element solution is **an approximation**:
 - Equilibrium/compatibility requirements of the **exact solution** are NOT satisfied.
- Strong knowledge on displacements, weak knowledge on stresses.
 - Computed **displacement field** is continuous over the structure,
 - Computed **stress** is known at discrete points of the structure.

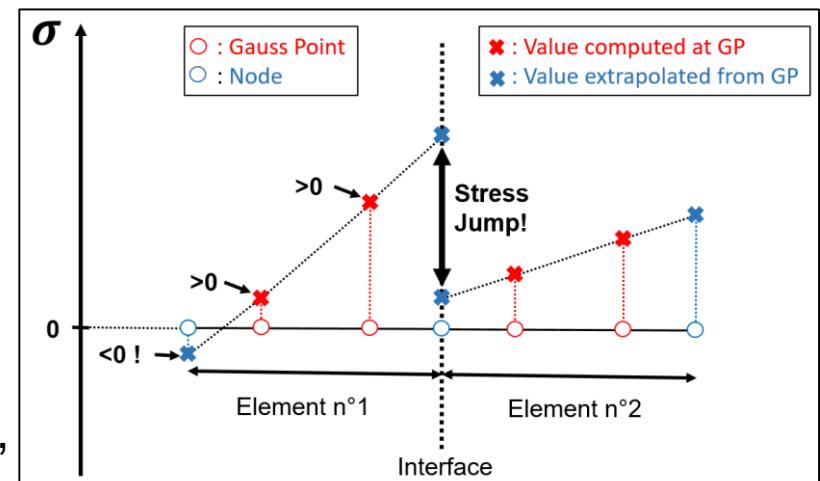
Pay attention : In FEM, one can only trust at best the average stress over element!

6. Computed results

- In isoparametric elements, the stress field is computed at **Gauss points**:

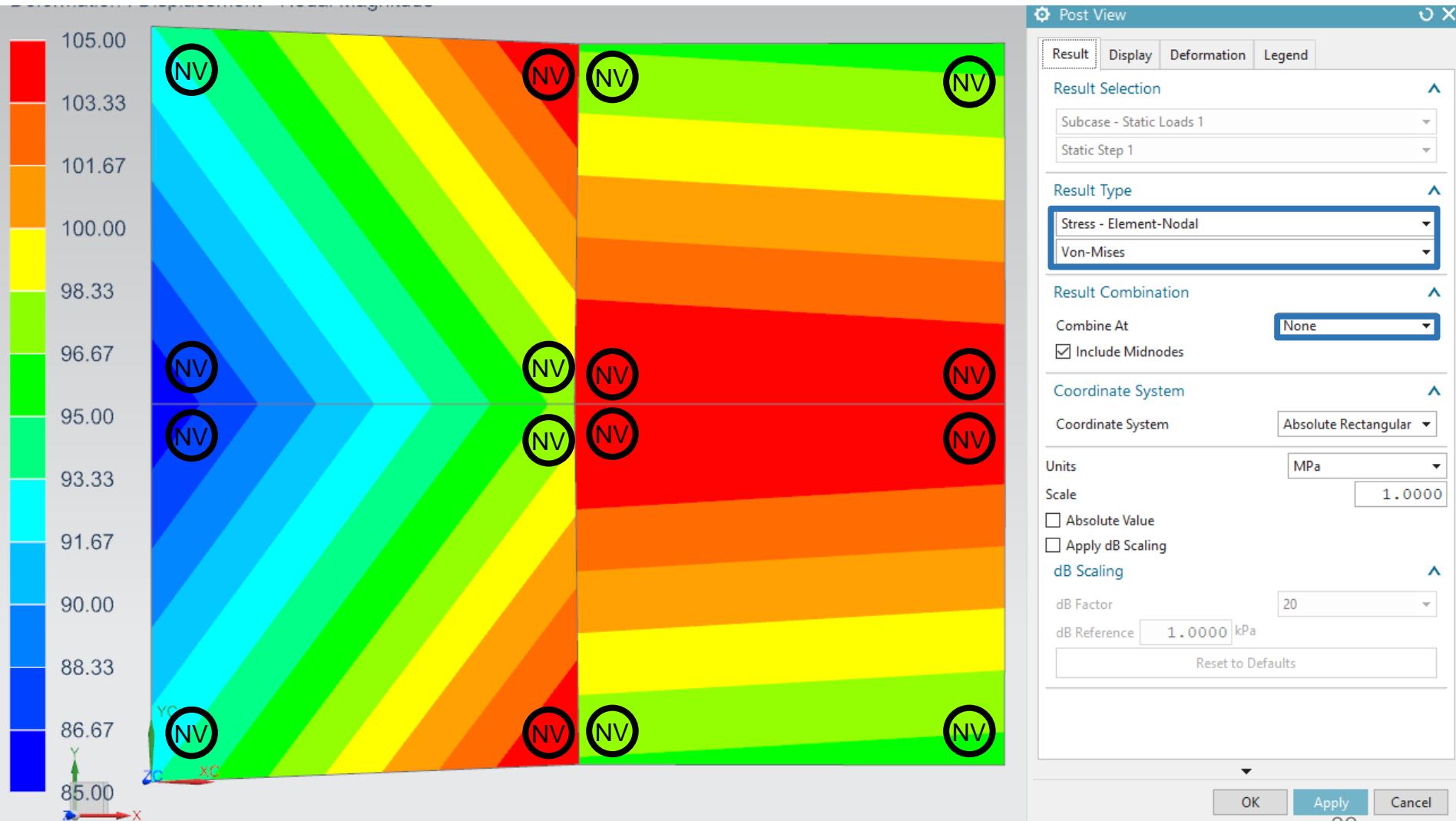


- Then, the field is extrapolated from Gauss points to corners:
- This can lead to large inter-element jumps.
- The obtained stress field is discontinuous:
 - Converges to a smooth field when the number of elements increases.
- **Best practice rules** : Try to avoid:
 - Large stress gradient within an element,
 - Large stress jumps between elements.



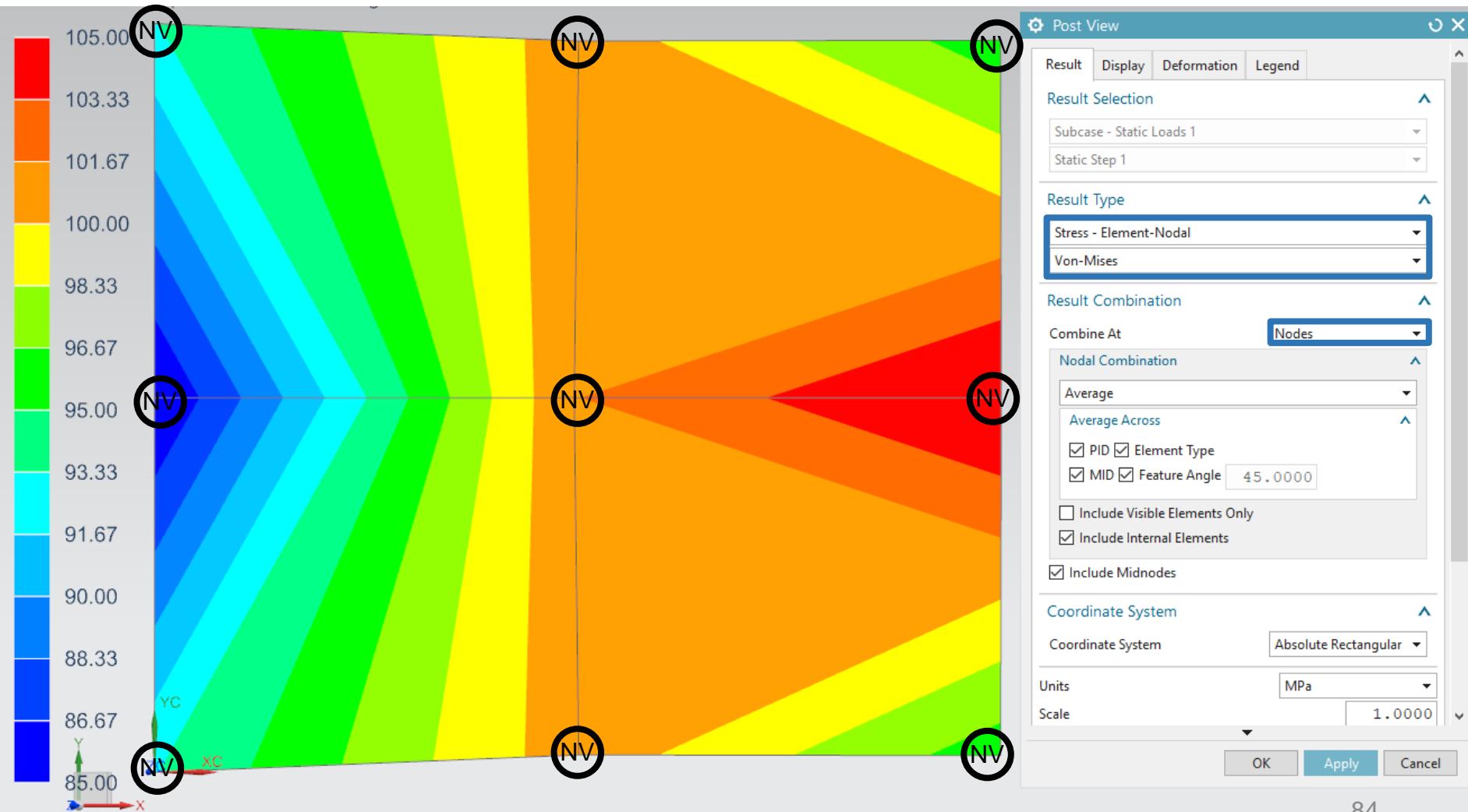
6. Computed results

- Elemental-nodal: 1 value per node per element.
 - Multiple nodal values (NV) per node (one per connected element).



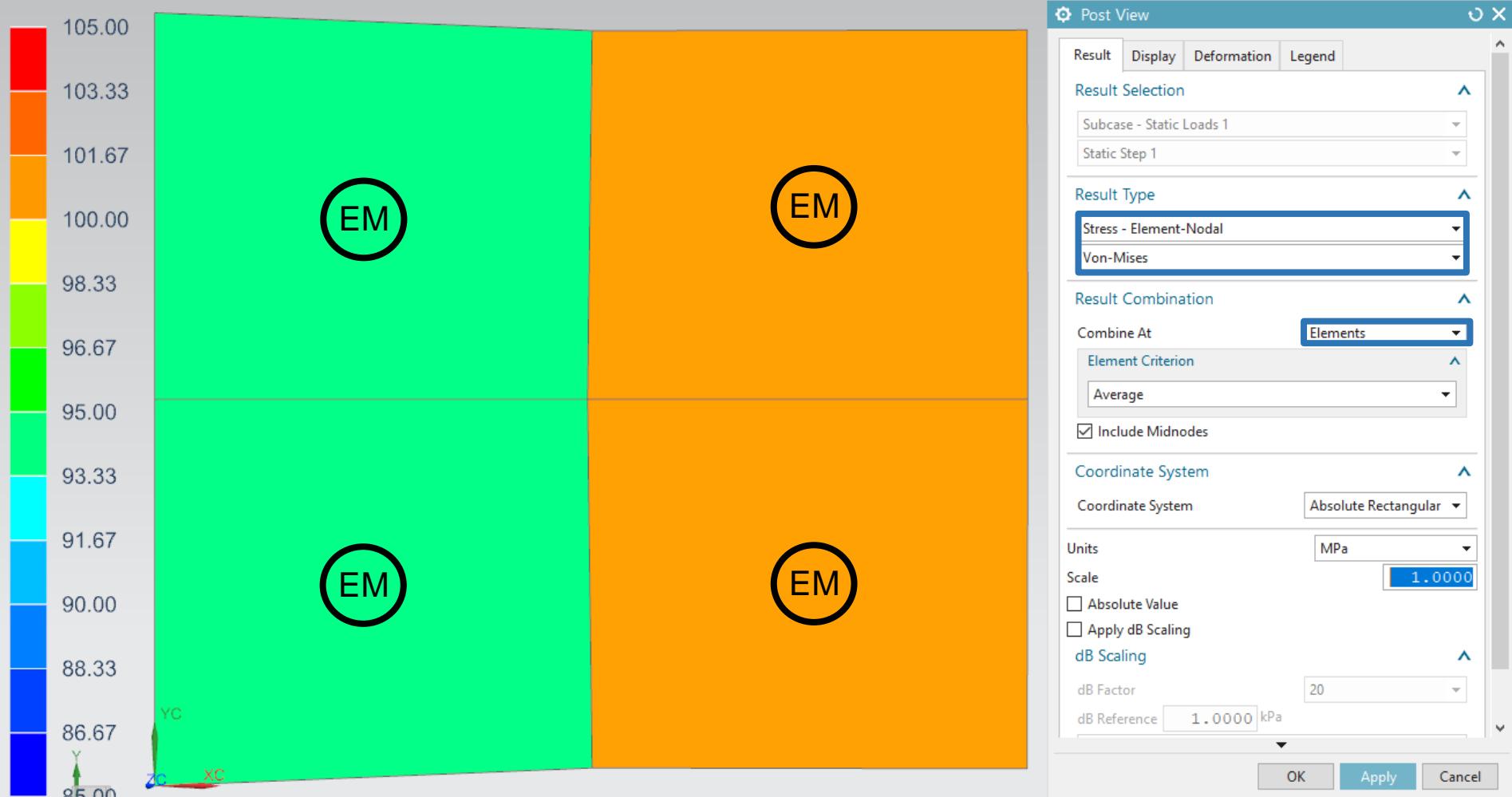
6. Computed results

- Elemental-nodal, averaged at nodes: 1 value per node.
 - A unique Nodal Value (NV) is obtained by averaging the nodal values per node.

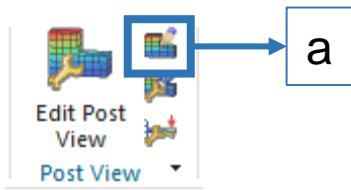


6. Computed results

- Elemental-nodal, **averaged at elements**: 1 value per element.
 - A unique **Elemental Value (EV)** is obtained by **averaging** NV's **per element**.
 - Same results as « Stress – Element » result type.



6. Computed results

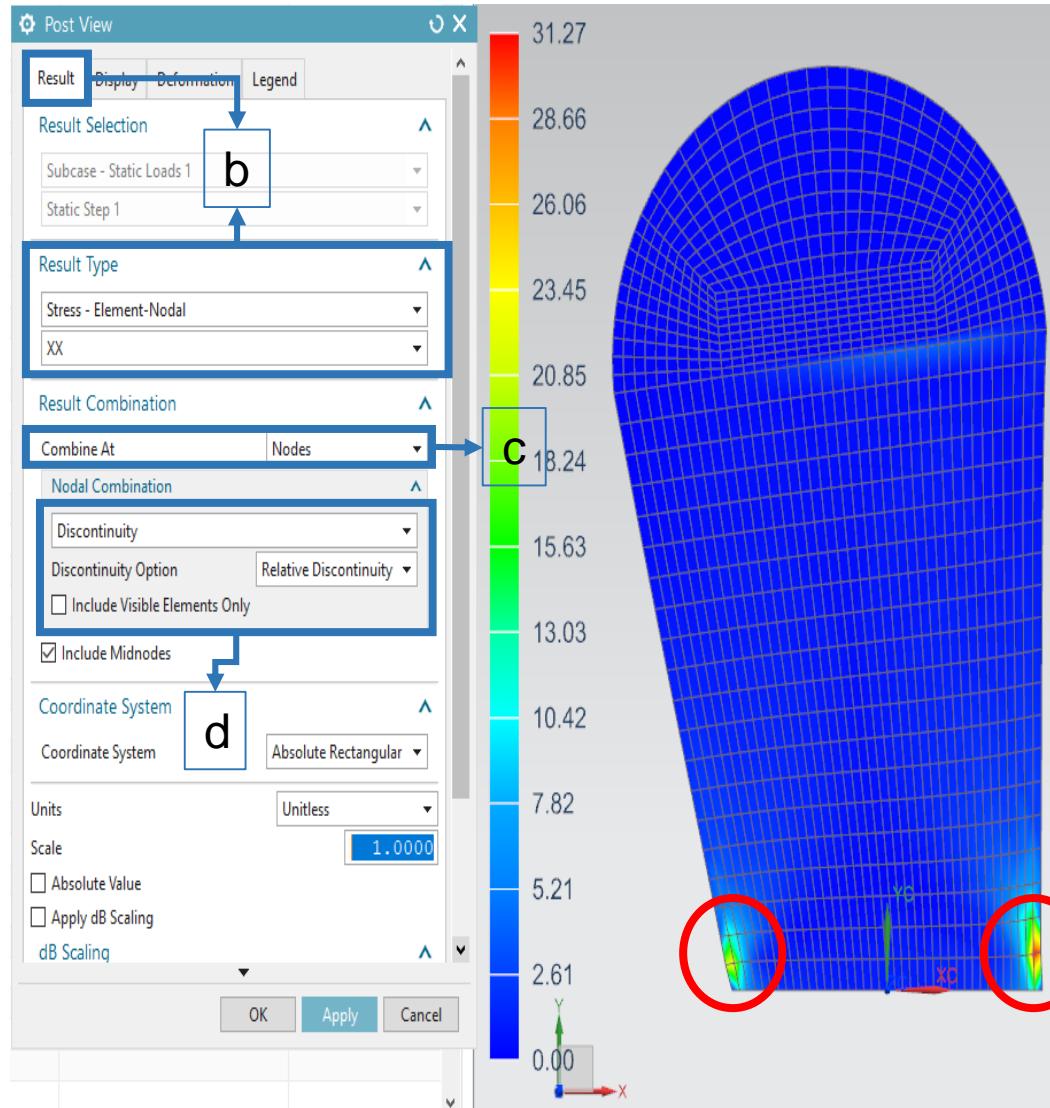


➤ Check for stress discontinuities in NX:

- Select “set results”
- Result → Result Type
→ Stress – Elemental-Nodal
- Combine at → Nodes
- Nodal combination
→ Discontinuity
→ Relative Discontinuity

Relative Discontinuity	$100 \cdot x \cdot (VMAX_{node} - VMIN_{node}) / (VMAX_{model} - VMIN_{model})$
------------------------	---

Possible singularity!

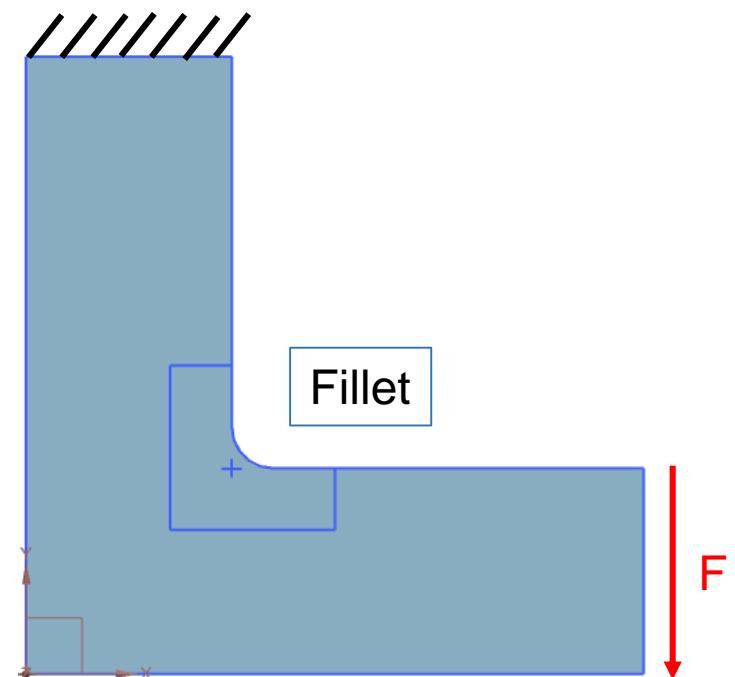
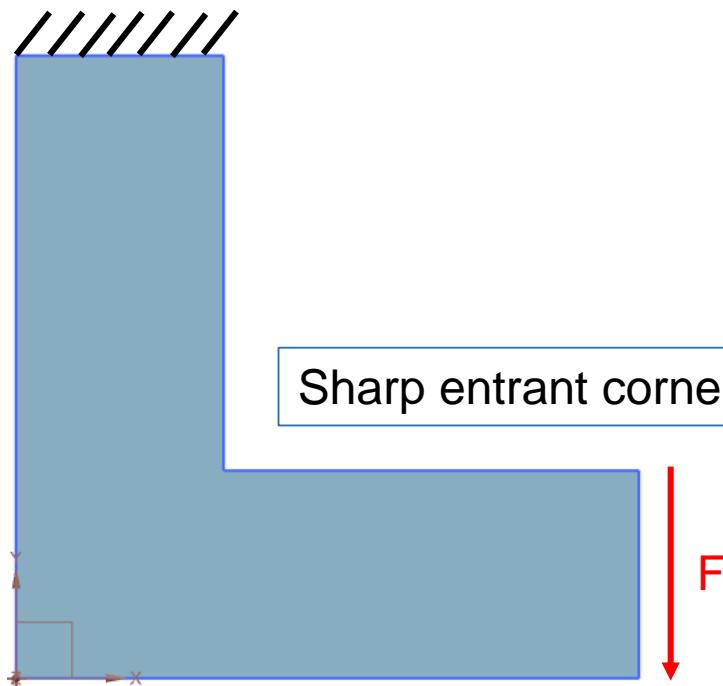


Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
- 6. Computed results**
 - 1. Singularities**
7. General remarks
8. General project instructions

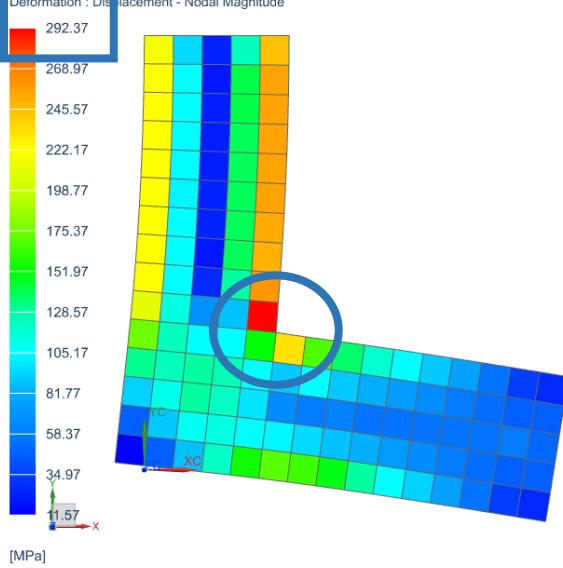
6.1 Singularities

Pay attention to singularities!

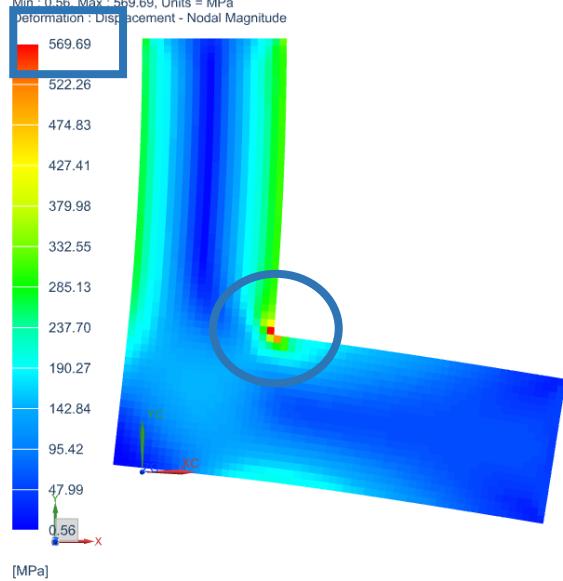


6.1 Singularities

singularity_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 11.57, Max : 292.37, Units = MPa
Deformation : Displacement - Nodal Magnitude

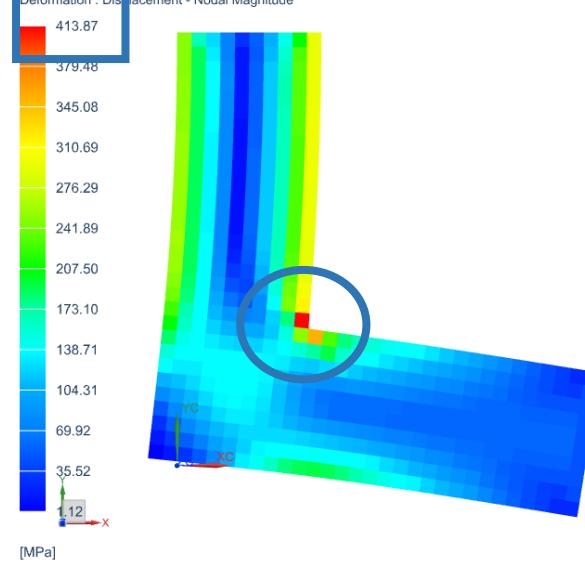


singularity_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.56, Max : 569.69, Units = MPa
Deformation : Displacement - Nodal Magnitude

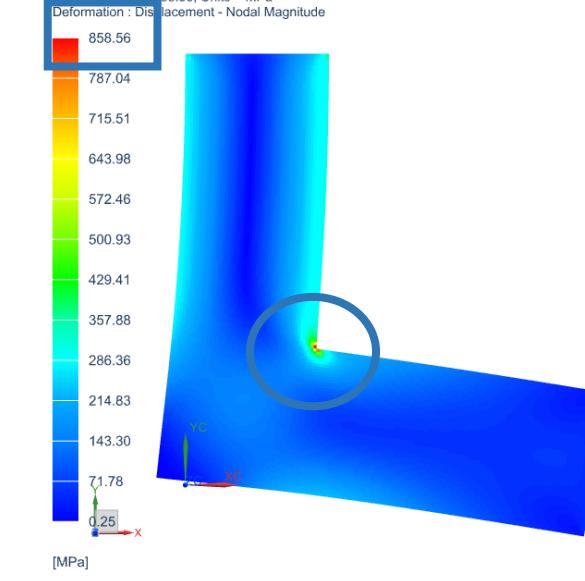


Entrant corner

singularity_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 1.12, Max : 413.87, Units = MPa
Deformation : Displacement - Nodal Magnitude

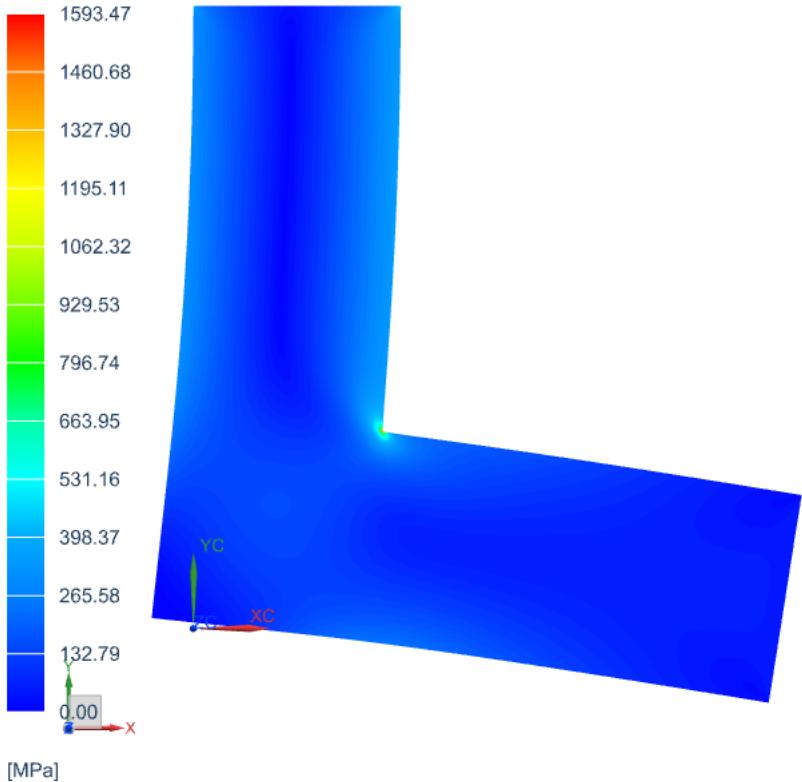


singularity_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.25, Max : 858.56, Units = MPa
Deformation : Displacement - Nodal Magnitude

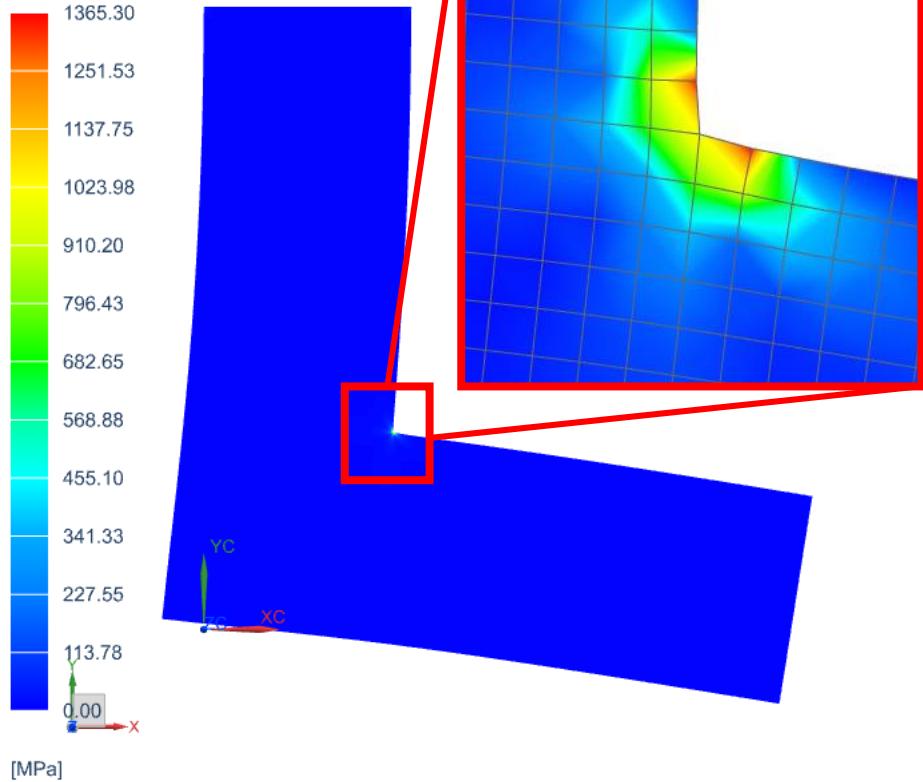


6.1 Singularities

singularity_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.00, Max : 1593.47, Units = MPa
Deformation : Displacement - Nodal Magnitude



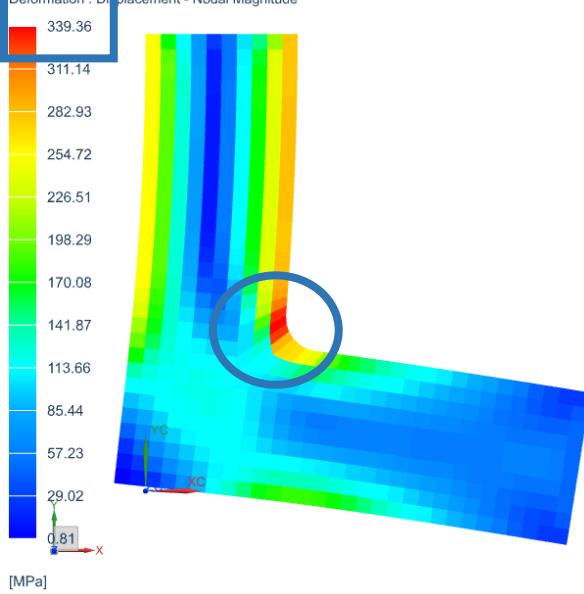
singularity_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Local Discontinuity, Von-Mises
Min : 0.00, Max : 1365.30, Units = MPa
Deformation : Displacement - Nodal Magnitude



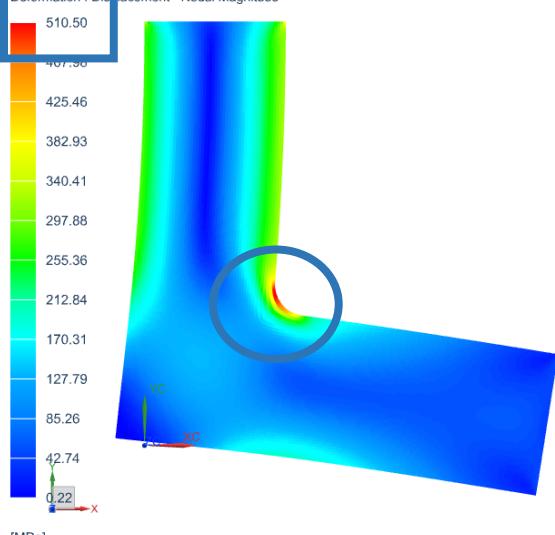
Local discontinuity of stresses very high!

6.1 Singularities

filleted_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.81, Max : 339.36, Units = MPa
Deformation : Displacement - Nodal Magnitude

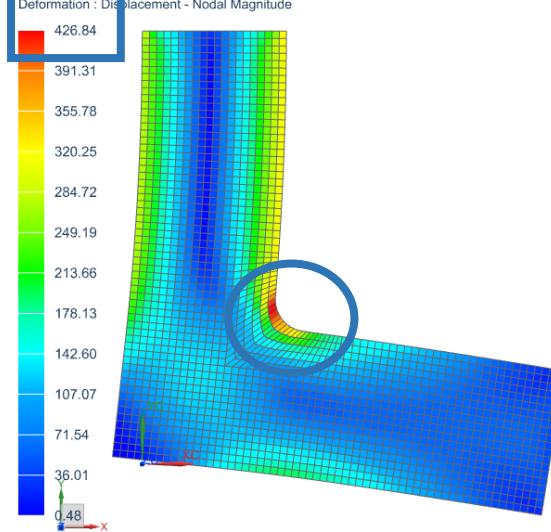


filleted_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.22, Max : 510.50, Units = MPa
Deformation : Displacement - Nodal Magnitude

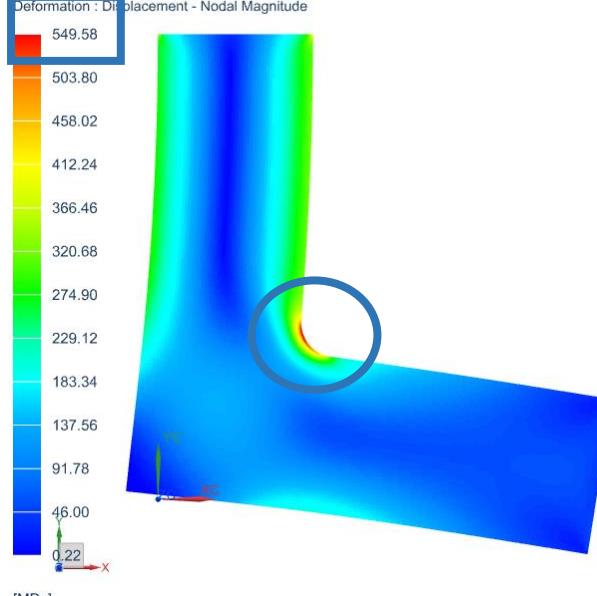


Fillet

filleted_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.48, Max : 426.84, Units = MPa
Deformation : Displacement - Nodal Magnitude

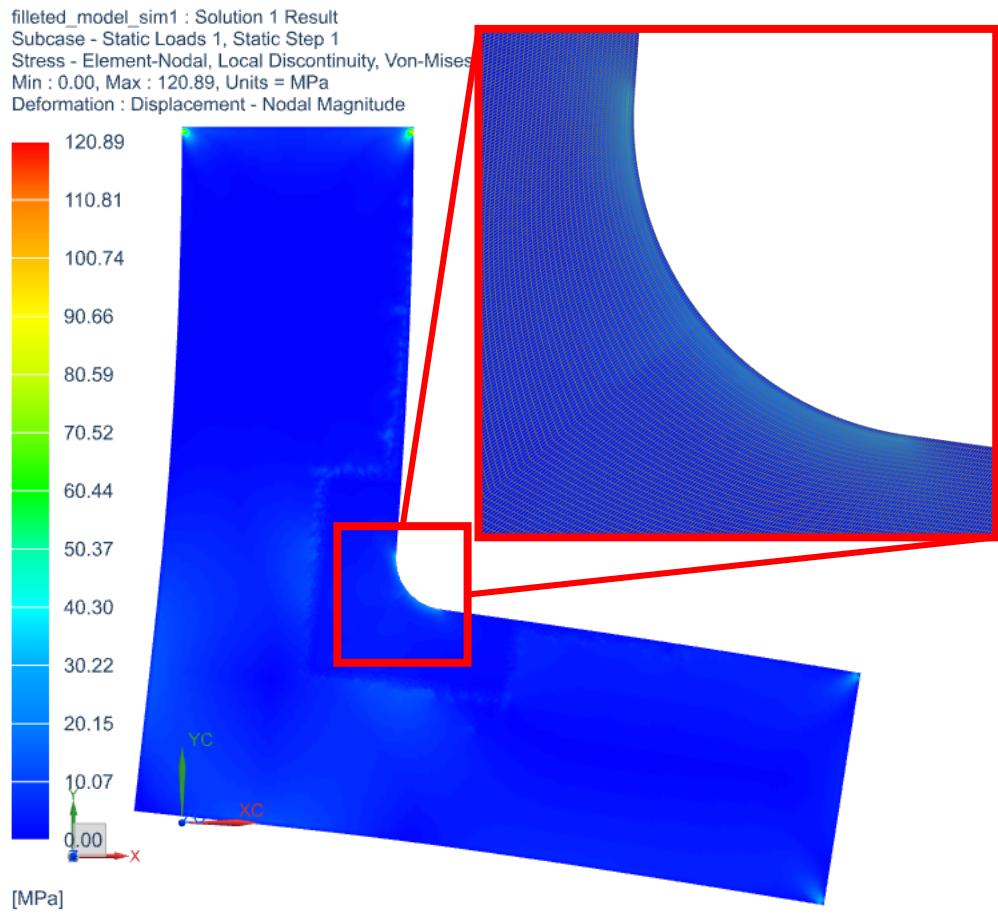
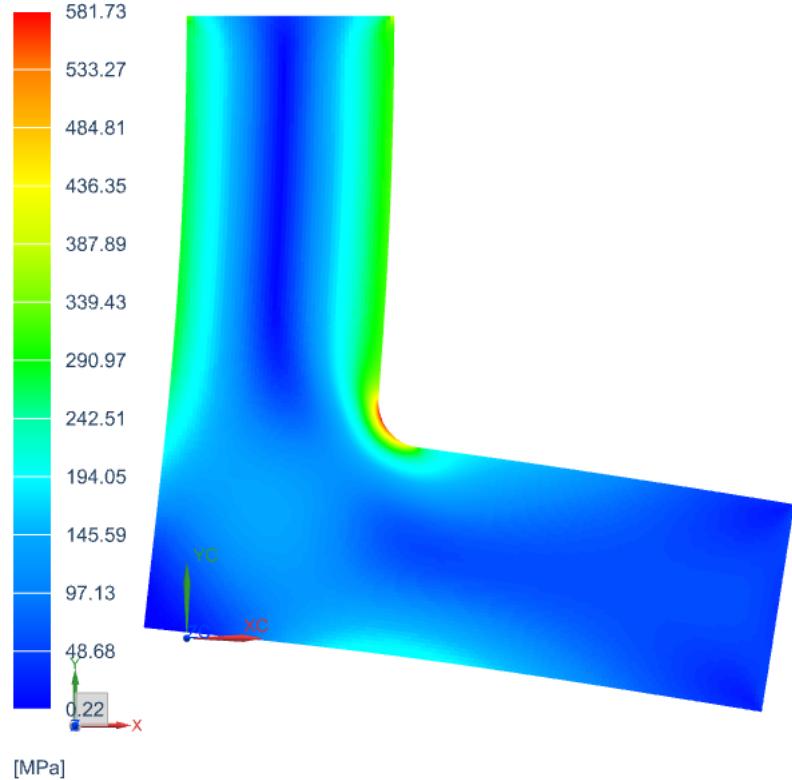


filleted_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.22, Max : 549.58, Units = MPa
Deformation : Displacement - Nodal Magnitude



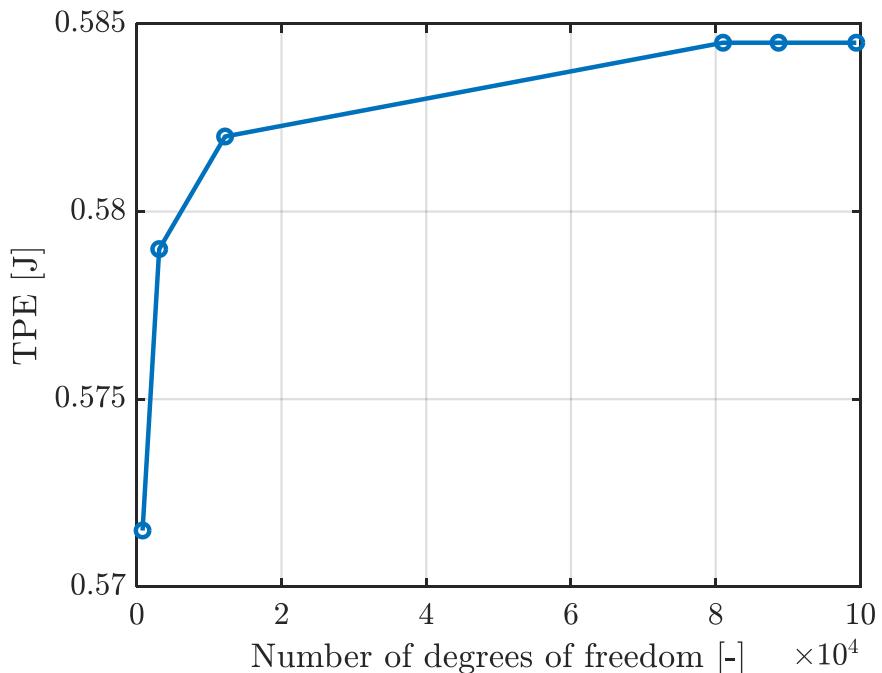
6.1 Singularities

filleted_model_sim1 : Solution 1 Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.22, Max : 581.73, Units = MPa
Deformation : Displacement - Nodal Magnitude

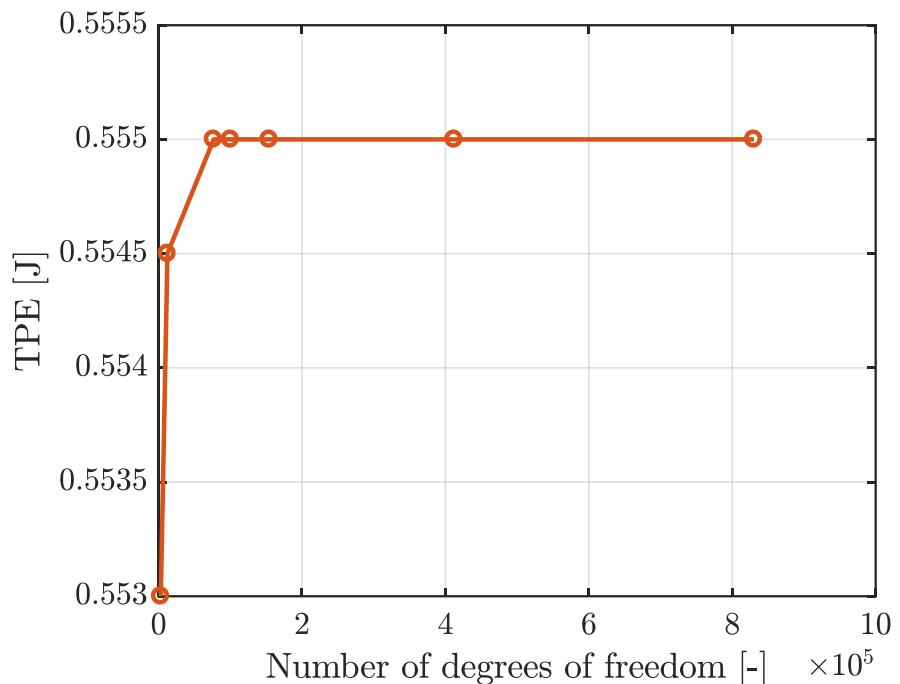


Stresses uniformly distributed

6.1 Singularities



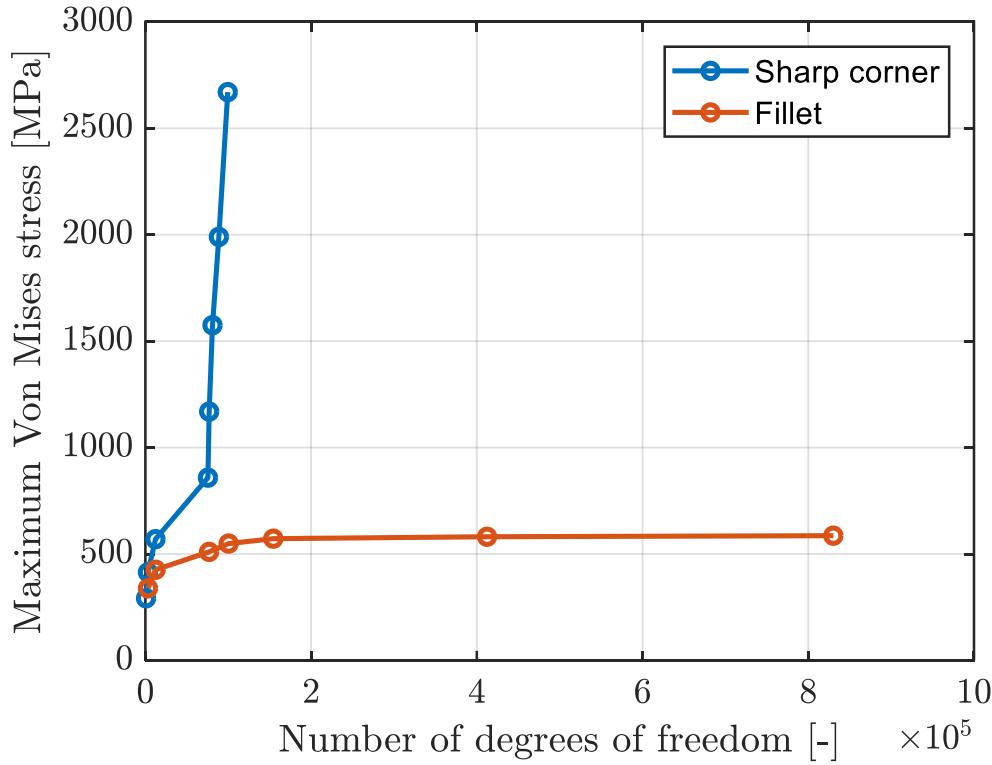
With sharp corner



With fillet

Global convergence in both cases!

6.1 Singularities



Global convergence does not mean local convergence!

Deal with your singularities (if necessary)
before doing any finite element analysis!!!

6.1 Singularities

Possible causes of singularity:

1. Sharp corners
2. Point loads
3. Constraints
4. Welds
5. Cracks



Singularity
 \neq
Stress concentration

It is **not always** possible to avoid them nor necessary to suppress them. If the location of the singularity is not of interest, the solution can still be valid at certain distance from it.

Be critical with your results!!!

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
- 7. General remarks**
8. General project instructions

General remarks

Type of element (Samcef)	Type of element (Finite Element Method Course)
[T26] (3)	Constant Strain Triangle - T3
[T26] (6)	Linear Strain Triangle - T6
[T15] (4)	Wilson Quadrangles – Q6
[T15] (8)	Biquadratic Quadrangles – Q8

Use only one type of element per simulation (if possible).

Remark: see the definition of the element types in the theoretical course, ch.10.

7. General remarks

Tips

- Try different meshing tools other than “2D mesh”, e.g. “2D mapped”;
- Use “Mesh Control”;



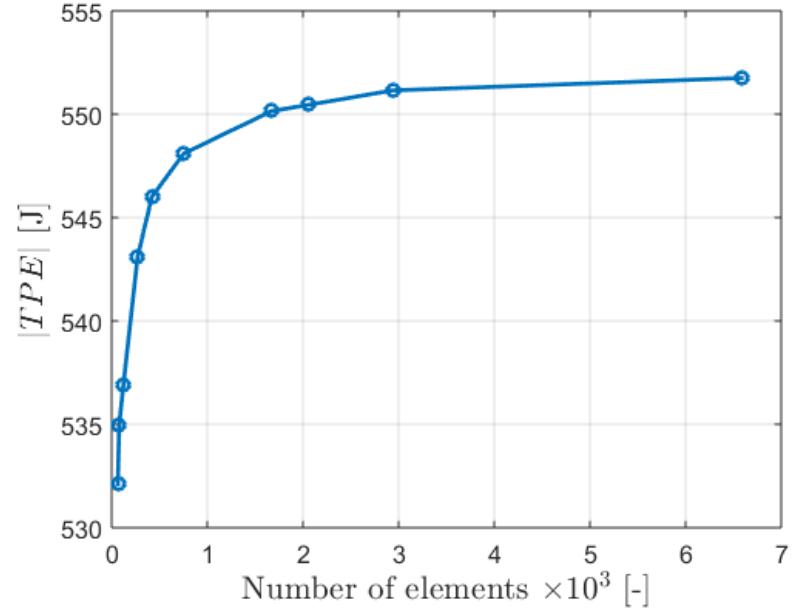
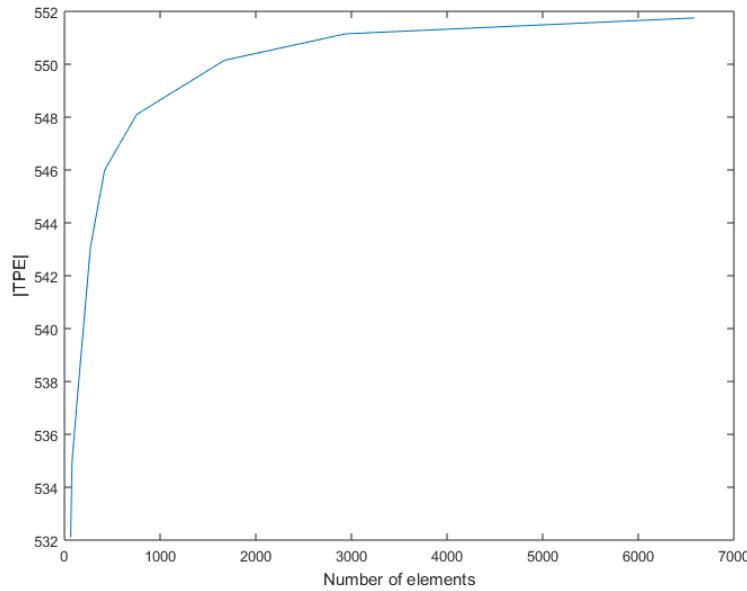
Possible bugs (they might have been corrected in NX 1859)

- The “Gravity” load type can induce a bug, prefer the “Acceleration” load type when gravity has to be considered;
- The “Pressure – On 2D elements XY” might need a correction factor of 1000 to get the right value in the .dat file.
 - If you find other errors, please let us know!
- The “Force” loads might need a correction factor of 100 to get the right value in the .dat file.

➤ **Check your .dat and .res files!**

7. General remarks

- Make your graphs and schemes readable!



Bad graph

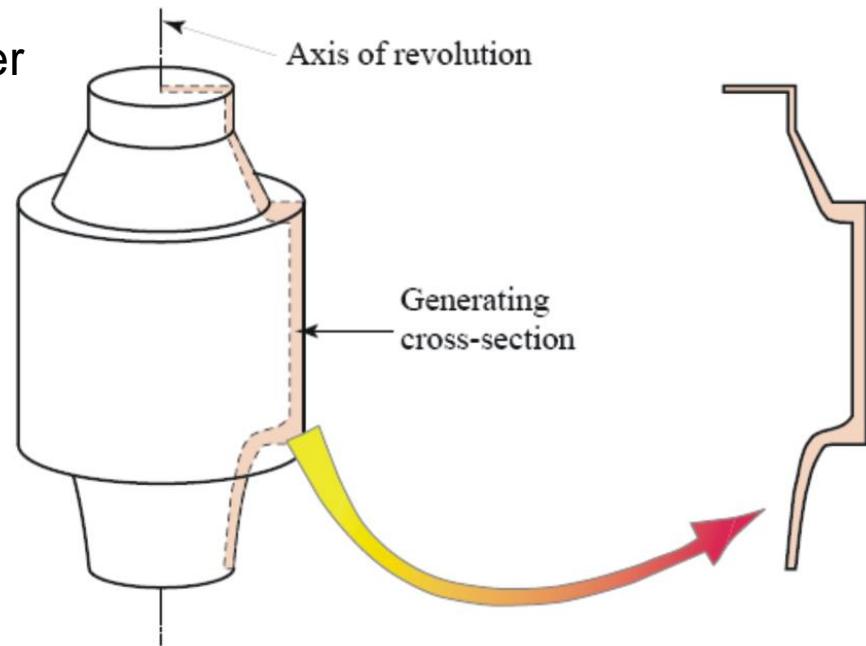


Good graph

7. General remarks

➤ Axisymmetric modelling:

- Typical Problem: Thick cylinder under pressure, rotating disk.
- Requirements:
 - Axisymmetric geometry,
 - Axisymmetric loading.
- Basics: invariance w.r.t. θ of:
 - Displacement field,
 - Strain field,
 - Stress field.
- In a FE software, a slice of one **radian** is studied.



Remark: A uniform **radial** displacement is no longer a rigid body motion, but produces a **circumferential** strain.

7. General remarks

Questions?

1. Check this presentation!
2. The “NX help” is available in:
 - File → Help → Help NX
3. For other questions forums have been created on eCampus and the answers will be given by the monitoring students:
 - louis.denis@student.uliege.be
 - nicolas.dujardin@student.uliege.be
 - ali sezgin@student.uliege.be
 - clement.moureau@student.uliege.be

Presentation Plan

1. Installation of the softwares
2. Quick introduction to FEM
3. Mechanical problem description
4. Analysis with strength of materials
 1. Idealization of the problem
 2. Relevant results
5. Analysis with NX 12
 1. Introduction
 2. Moving around in NX
 3. Geometry drawing (.prt)
 4. Generation of .fem and .sim files
 5. Material properties (.fem)
 6. Mesh generation(.fem)
 7. Boundary conditions and loads (.sim)
 8. Launch a linear static analysis (.sim)
 9. Post-processing of the results
6. Computed results
 1. Singularities
7. General remarks
- 8. General project instructions**

8. General project instructions

➤ Project Goals :

- Model a mechanical problem in linear elasticity and small strain.
- Discuss and analyze the results obtained with a FE software.
- Perform an optimization of the mechanical element.

➤ Group :

- Form groups of six.
- The problem statement is related to the group number.
- All files will be available when the group enrolment is performed.

➤ Workload balancing file :

- Mention all the sections of the report that you have written.
 - Mention concisely what you have done.
 - All members should agree with the content of the document and sign it.
- **No document signed, no grade (0/20 for each member)!**

8. General project instructions

➤ Key Dates:

- Deadline for temporary NX data set + first attempt at solving the problem with mechanics of material (March 31st 2022 before 8:00pm).
- Deadline for report (May 5th 2022 before 2:00pm).

➤ File/Document Uploading's (Temporary Data):

- Temporary NX data set files:
 - *.prt, .fem & .sim* files for at least 1 type of element (that run without problem).
 - The associated *.dat & .res* files.
- Temporary documents on the first attempt at solving the problem with mechanics of material.

If you do not send all the requested temporary NX data set files and documents on the first attempt at solving the problem with the mechanics of material before the deadline, all members of the group will receive a grade of 0/20 for the project.

Respect the file naming conventions!

Read the general project instruction document!

8. General project instructions

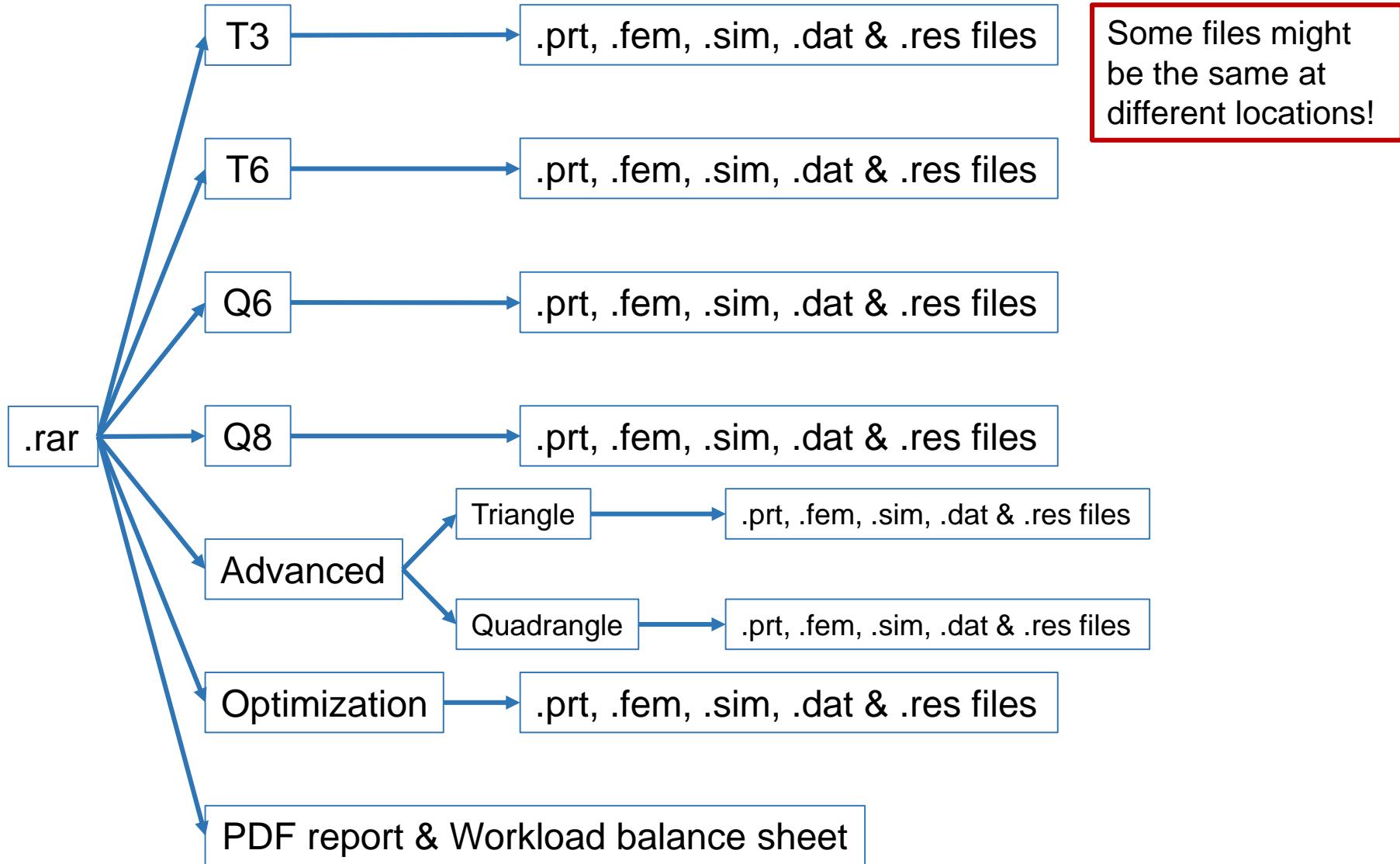
➤ File/Document Uploading's (Final Data):

- NX data set files:
 - For each type of finite element and type of mesh (coarse and advanced), the associated .prt, .fem, .sim, .dat & .res files.
 - For the optimization, the associated .prt, .fem, .sim, .dat & .res files.
- The PDF report.
- The workload balance sheet (electronically) signed by each member of the group.



The folder containing every document and file must be well structured! See next slide

8. General project instructions



8. General project instructions

- Project Q&A:
 - Room to be determined later.
 - Make a reservation according to the online schedule:
 - More information later.
 - Priority to the students who have made a reservation!
 - The student attendance will be taken at the beginning of each session.
- Sanctions:
 - Each group must attend at least **2** sessions!
 - Contact the supervising students, if you could not attend the reserved session!
Otherwise, you **could** be individually penalized!

Read the supervising session document and the session schedule document!

8. General project instructions

➤ Project Contents

- The project is divided into three main parts :
 1. Strength of Materials (10%)
 2. Finite Element Study (70%)
 3. Optimization Study (10%)
 4. Report Quality (10%)
- **Read the project instructions** (same for all the project statements) and the **project statement** (different for all the groups). Check the degree of relevance!
- The goals of the study as well as the optimization constraints are clearly precised in the project statement!

Read the project instruction document and your project statement document!

8. General project instructions

➤ Report Contents :

- A peculiar attention will be paid to the quality of the written report, to the ability to summarize, and to the scientific rigor of the words :
 - Introduction – Conclusion - Bibliography
 - Page header
 - Report structure
 - Equations
 - Graphs and Tables
 - Results on Mesh
- Grammar and Spelling Check
- **Do not forget to explain what you are doing!**
- **Read the Report instructions!**

Plagiarism of reports, the first attempt at solving the problem with the mechanics of material, final and temporary NX files will not be tolerated!

8. General project instructions

GOOD WORK!

9. References

- C. Leu Ming, Ghazanfari Amir, Kolan Krishna, NX 10 for Engineering Design, https://web.mst.edu/~mleu/nx_manuals/nx10.pdf
- <http://www.stressebook.com/finite-element-analysis-in-a-nut-shell/>
- <https://www.comsol.com/blogs/singularities-in-finite-element-models-dealing-with-red-spots/>

Annex

A1. Governing equations

➤ Governing equations:

The exact solution satisfies the following equations:

1. Equilibrium equations:

$$\frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} = 0, \quad \text{along X axis} \quad \forall (x, y) \in V_a \cup V_b$$

$$\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} - \rho_b g = 0, \quad \text{along Y axis} \quad \forall (x, y) \in V_b$$

$$\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y} - \rho_a g = 0, \quad \text{along Y axis} \quad \forall (x, y) \in V_a$$

A1. Governing equations

➤ Governing equations:

The exact solution satisfies the following equations:

1. Strain-displacement equations:

$$\varepsilon_{xx} = \frac{\partial u_x}{\partial x}, \quad \varepsilon_{yy} = \frac{\partial u_y}{\partial y}, \quad \gamma_{xy} = \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} \quad \forall (x, y) \in V_a \cup V_b$$

$$\varepsilon_{zz} = \gamma_{xz} = \gamma_{yz} = 0$$

2. Constitutive equations:

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{pmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{pmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{(1-2\nu)}{2} \end{pmatrix} \begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{pmatrix} \quad \forall (x, y) \in V_a \cup V_b$$

$$\sigma_{zz} = \nu (\sigma_{xx} + \sigma_{yy})$$

A1. Governing equations

➤ Governing equations:

The exact solution satisfies the following equations:

1. Kinematic (= Essential) boundary conditions:

- Lower edge (restraint)

$$u_x(x = 0, y) = 0 \quad \text{and} \quad u_y(x = 0, y) = 0 \quad \forall y \in \left[-\frac{B}{2}, \frac{B}{2}\right]$$

2. Static (= Natural) boundary conditions:

- Left edge (load free)

$$\forall s \in [0, 1] : \quad x(s) = Ls \quad \text{and} \quad y(s) = -(1 - s)\frac{B}{2} - Rs$$

$$\sigma_{xx}n_x + \tau_{xy}n_y = 0, \quad \text{along X axis}$$

$$\text{and} \quad \vec{n} = [-\sin \alpha \ \cos \alpha]$$

$$\tau_{xy}n_x + \sigma_{yy}n_y = 0, \quad \text{along Y axis}$$



$$-\sigma_{xx} \sin \alpha + \tau_{xy} \cos \alpha = 0, \quad \text{along X axis}$$

$$-\tau_{xy} \sin \alpha + \sigma_{yy} \cos \alpha = 0, \quad \text{along Y axis}$$

A1. Governing equations

➤ Governing equations:

The exact solution satisfies the following equations:

1. Static (= Natural) boundary conditions:

- Right edge (pressure)

$$\forall s \in [0, 1] : \quad x(s) = Ls \quad \text{and} \quad y(s) = (1 - s) \frac{B}{2} + Rs$$

$$\sigma_{xx} n_x + \tau_{xy} n_y = -P n_x, \quad \text{along X axis}$$

$$\text{and} \quad \vec{n} = [-\sin \alpha \quad -\cos \alpha]$$

$$\tau_{xy} n_x + \sigma_{yy} n_y = -P n_y, \quad \text{along Y axis}$$

$$-\sigma_{xx} \sin \alpha - \tau_{xy} \cos \alpha = P \sin \alpha, \quad \text{along X axis}$$

$$-\tau_{xy} \sin \alpha - \sigma_{yy} \cos \alpha = P \cos \alpha, \quad \text{along Y axis}$$



2. Upper edge (load free):

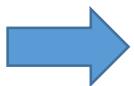
$$\forall \theta \in \left[-\frac{\pi}{2}, \frac{\pi}{2} \right] : \quad x(\theta) = L + R \cos \theta \quad \text{and} \quad y(\theta) = R \sin \theta$$

$$\sigma_{xx} n_x + \tau_{xy} n_y = 0, \quad \text{along X axis}$$

$$\text{and} \quad \vec{n} = [\sin \theta \quad -\cos \theta]$$

$$\tau_{xy} n_x + \sigma_{yy} n_y = 0, \quad \text{along Y axis}$$

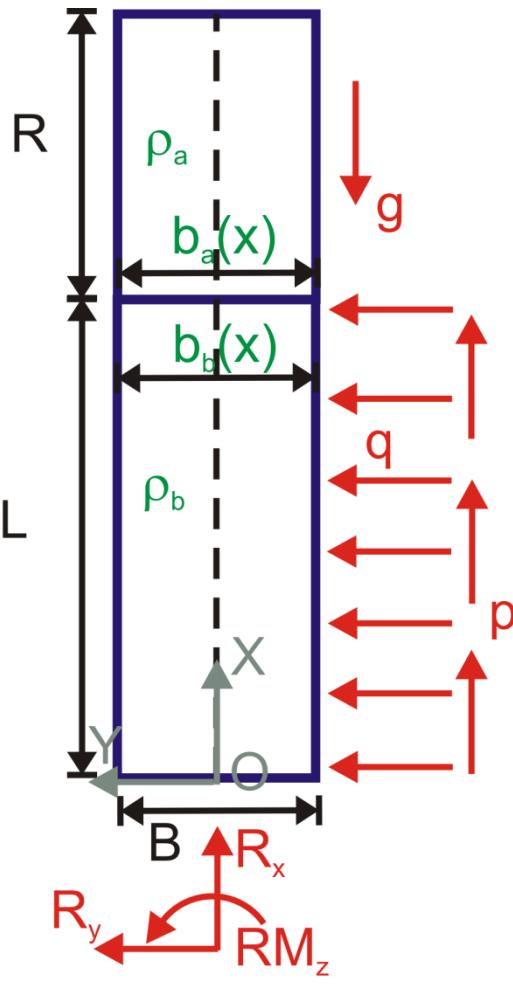
$$\sigma_{xx} \sin \theta - \tau_{xy} \cos \theta = 0, \quad \text{along X axis}$$



$$\tau_{xy} \sin \theta - \sigma_{yy} \cos \theta = 0, \quad \text{along Y axis}$$

A2. Stress resultants and displacement field

➤ Reaction Forces:



- Write down the 3 static equilibrium equations:

- Horizontal equilibrium equation:

$$\Rightarrow R_y + qL = 0 \Leftrightarrow R_y = -qL$$

- Vertical equilibrium equation:

$$\Rightarrow R_x + pL - (\rho_a V_a + \rho_b V_b)g = 0$$

$$\Leftrightarrow R_x = -pL + (\rho_a V_a + \rho_b V_b)g$$

- Moment equilibrium equation around O :

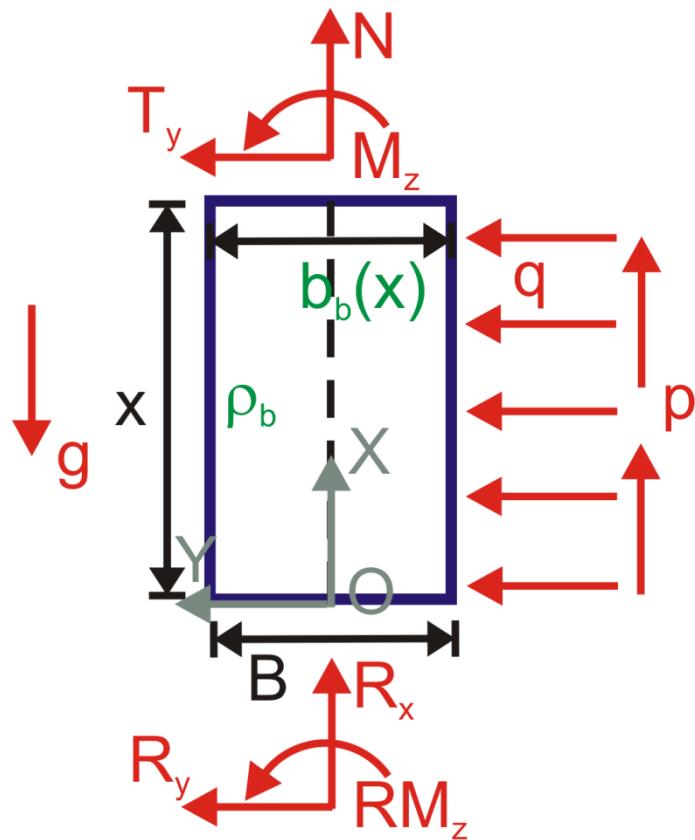
$$\Rightarrow RM_z + \frac{qL^2}{2} + \int_0^L p \frac{b_b(x)}{2} dx = 0$$

$$\Leftrightarrow RM_z = -\frac{qL^2}{2} - \frac{pV_b}{2}$$

➤ Statically determinate structure

A2. Stress resultants and displacement field

➤ Stress resultants: Beam b



- Write down the 3 static equilibrium equations:

- Horizontal equilibrium equation:

$$\Rightarrow T_y(x) + R_y + qx = 0$$

$$\Leftrightarrow T_y(x) = -R_y - qx$$

$$\Leftrightarrow T_y(x) = q(L - x)$$

- Vertical equilibrium equation:

$$\Rightarrow N(x) + R_x + px - \rho_b g \int_0^x A_b(\xi) d\xi = 0$$

$$\Leftrightarrow N(x) = -R_x - px + \rho_b g \frac{(b_b(x) + B)}{2} x$$

$$\Leftrightarrow N(x) = p(L - x) - \left[\rho_a V_a + \rho_b \left(V_b - \frac{(b_b(x) + B)}{2} x \right) \right] g$$

- Moment equilibrium equation around O:

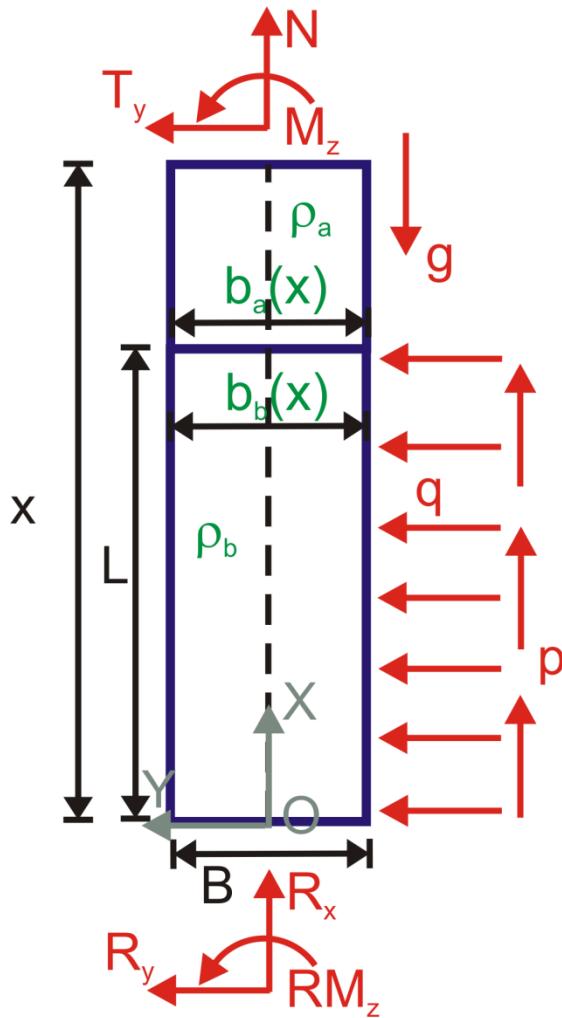
$$\Rightarrow M_z(x) + RM_z + T_y(x)x + \frac{qx^2}{2} + \int_0^x p \frac{b_b(\xi)}{2} d\xi = 0$$

$$\Leftrightarrow M_z(x) = -RM_z - T_y(x)x - \frac{qx^2}{2} - p \frac{(b_b(x) + B)}{2} x$$

$$\Leftrightarrow M_z(x) = \frac{q(L^2 - 2Lx + x^2)}{2} + \frac{p}{2} [V_b - (b_b(x) + B) \frac{x}{2}]$$

A2. Stress resultants and displacement field

➤ Stress resultants: Beam a



- Write down the 3 static equilibrium equations:

- Horizontal equilibrium equation:

$$\Rightarrow T_y(x) + R_y + qL = 0$$

$$\Leftrightarrow T_y(x) = -R_y - qL \Leftrightarrow T_y(x) = 0$$

- Vertical equilibrium equation:

$$\Rightarrow N(x) + R_x + pL - \rho_b V_b g - \rho_a g \int_L^x A_a(\xi) d\xi = 0$$

$$\Leftrightarrow N(x) = -R_x - pL + \rho_b V_b g + \rho_a g \left(R^2 \arcsin \left(\frac{x-L}{R} \right) + (x-L) \sqrt{R^2 - (x-L)^2} \right)$$

$$\Leftrightarrow N(x) = -\rho_a g \left[V_a - \left(R^2 \arcsin \left(\frac{x-L}{R} \right) + (x-L) \sqrt{R^2 - (x-L)^2} \right) \right]$$

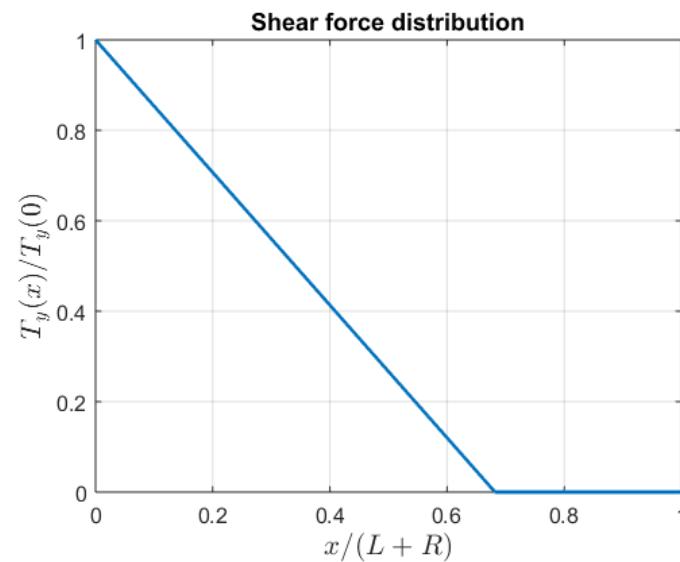
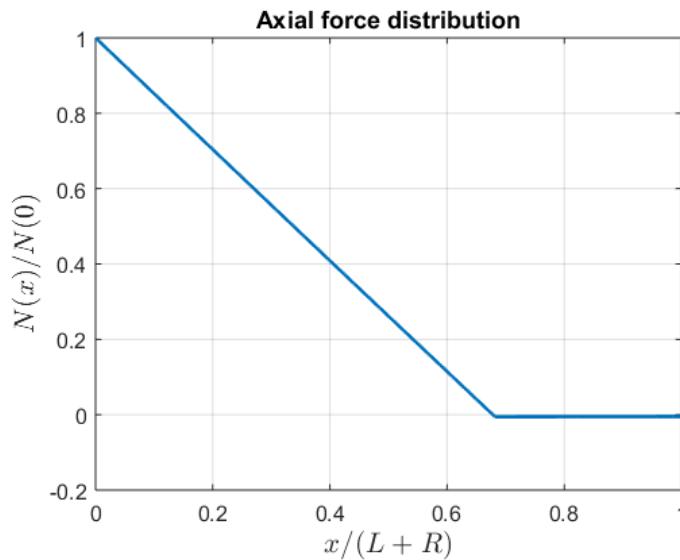
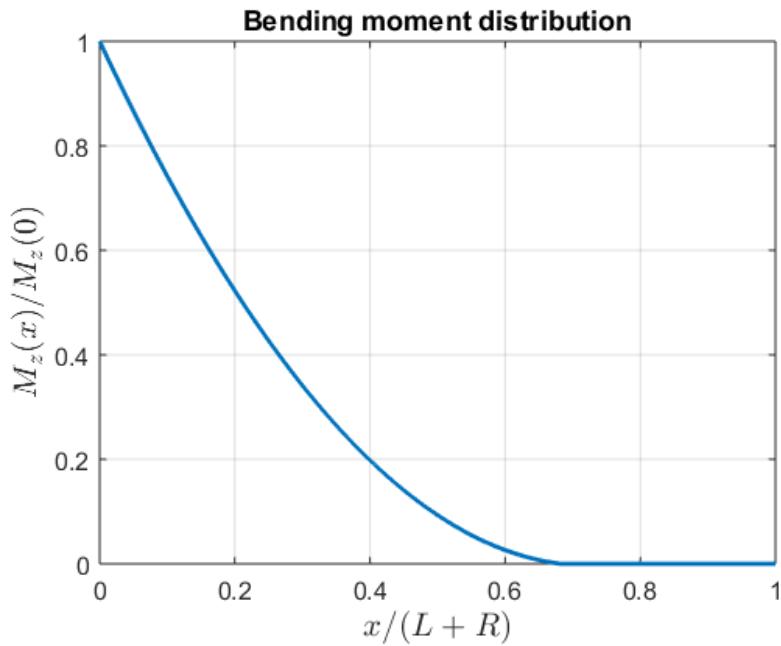
- Moment equilibrium equation around O:

$$\Rightarrow M_z(x) + RM_z + T_y(x)x + \frac{qL^2}{2} + \int_0^L p \frac{b_b(\xi)}{2} d\xi = 0$$

$$\Leftrightarrow M_z(x) = -RM_z - T_y(x)x - \frac{qL^2}{2} - \frac{pV_b}{2} \Leftrightarrow M_z(x) = 0$$

A2. Stress resultants and displacement field

➤ Stress Resultants: Sum up



The maximum stresses will occur at the restraint!

A2. Stress resultants and displacement field

➤ Effective von Mises stress at the restraint ($x = 0$) :

- Stress resulting from axial force N : $\sigma_{xx}^{Tension} = \frac{N(0)}{A_b(0)}$
 - Stress resulting from shear force T_y : $\tau_{xy}^{Shear}(y) = \frac{3}{2} \frac{T_y(0)}{A_b(0)} \left(1 - \left(\frac{2y}{b_b(0)} \right)^2 \right)$
 - Stress resulting from bending moment M_z : $\sigma_{xx}^{Bending}(y) = -\frac{M_z(0)y}{I_{z_b}(0)}$
 - Effective von Mises stress:
 - i. Beam: $\sigma_{yy} = 0$
 - ii. Plane strain: $\sigma_{zz} = \nu_b \sigma_{xx}$ & $\tau_{zx} = \tau_{yz} = 0$
- $\bar{\sigma}^{VM} = \sqrt{(1 - \nu_b + \nu_b^2) \sigma_{xx}^2 + 3 \tau_{xy}^2}$

➤ Upper layer ($y = \frac{b_b(0)}{2}$) : $\bar{\sigma}^{VM} \left(\frac{b_b(0)}{2} \right) = \sqrt{(1 - \nu_b + \nu_b^2) \left(\sigma_{xx}^{Tension} + \sigma_{xx}^{Bending} \left(\frac{b_b(0)}{2} \right) \right)^2} = 64.121 \text{ MPa}$

➤ Neutral fiber ($y = 0$) : $\bar{\sigma}^{VM}(0) = \sqrt{(1 - \nu_b + \nu_b^2) (\sigma_{xx}^{Tension})^2 + 3 (\tau_{xy}^{Shear}(0))^2} = 38.666 \text{ MPa}$

➤ Lower layer ($y = -\frac{b_b(0)}{2}$) : $\bar{\sigma}^{VM} \left(-\frac{b_b(0)}{2} \right) = \sqrt{(1 - \nu_b + \nu_b^2) \left(\sigma_{xx}^{Tension} + \sigma_{xx}^{Bending} \left(-\frac{b_b(0)}{2} \right) \right)^2} = 67.504 \text{ MPa}$

A2. Stress resultants and displacement field

➤ Strain Energy Density : Reminder

- In plane strain state, the strain energy density is given by:

$$w = \frac{1}{2} \left[\frac{(1-\nu^2)}{E} \left(\sigma_{xx}^2 + \sigma_{yy}^2 - 2 \frac{\nu}{1-\nu} \sigma_{xx} \sigma_{yy} \right) + \frac{\tau_{xy}^2}{G} \right] \quad \overset{\sigma_{yy}=0}{\longrightarrow} \quad w = \frac{1}{2} \left[\frac{(1-\nu^2)}{E} (\sigma_{xx}^2) + \frac{\tau_{xy}^2}{G} \right]$$

- By adding the work done by each stress resultants and by performing the integration over the beam span, the strain energy of a beam is:

$$\mathcal{U} = \frac{1}{2} \int_0^L \left(\frac{(1-\nu^2) N^2(\xi)}{EA(\xi)} + \frac{(1-\nu^2) M_z^2(\xi)}{EI_z(\xi)} + \frac{T_y^2(\xi)}{GA_s(\xi)} \right) d\xi \quad \left\{ \begin{array}{l} A_s(\xi) = \frac{5}{6} A(\xi) \\ G = \frac{E}{2(1+\nu)} \end{array} \right.$$

- In our case, the strain energy is obtained by numerically performing the integration over each beam :

$$\left. \begin{aligned} \mathcal{U}^{(a)} &= \frac{1}{2} \int_L^{L+R} \left(\frac{(1-\nu_a^2) N^2(\xi)}{E_a A_a(\xi)} \right) d\xi \\ \mathcal{U}^{(b)} &= \frac{1}{2} \int_0^L \left(\frac{(1-\nu_b^2) N^2(\xi)}{E_b A_b(\xi)} + \frac{(1-\nu_b^2) M_z^2(\xi)}{E_b I_{z_b}(\xi)} + \frac{T_y^2(\xi)}{G_b A_{s_b}(\xi)} \right) d\xi \end{aligned} \right\} \Rightarrow \mathcal{U} = \mathcal{U}^{(a)} + \mathcal{U}^{(b)} = 459.916 J$$

➤ Pay attention that this result does not come from superposition principle (cf. Energy theorem notes available online).

A2. Stress resultants and displacement field

➤ Total Potential Energy:

- By definition, the total potential energy is : $TPE = \mathcal{U} - \mathcal{P}$
 - \mathcal{P} is the potential energy of external applied loads.
 - \mathcal{U} is the strain energy.
- If the **essential boundary conditions** consist to impose the displacement field to **zero** at some boundary parts, it may be proven **at equilibrium** that :

$$TPE = \mathcal{U} - \mathcal{P} = -\frac{1}{2}\mathcal{P} = -\mathcal{U}$$

Since, $\mathcal{U} = \frac{1}{2}\mathcal{P}$ **at equilibrium**.

- In our example, one has: $TPE = -459.916J$
- In a finite element model, it may be proven that the value of total potential energy at equilibrium increases **monotonically** in absolute value to the exact one, as the number of DOFs increases (Mesh refinement) . In practice, the asymptotic value is only reached for simple mechanical problem.
- In the SAMCEF solver, the internal energy and total potential energy stand for the work done by the external forces $W^{ext} = \mathbf{q}_s^T \mathbf{g}_s$ = the work done by the internal forces $W^{int} = \mathbf{q}_s^T \mathbf{K}_s \mathbf{q}_s$.

A2. Stress resultants and displacement field

➤ Displacement Field: Reminder

- The unit virtual load method is a powerful method for calculating displacements and rotations. The principle of virtual forces applied to a beam states that:

$$\left. \begin{array}{l} \delta P \times \Delta \\ \delta M \times \theta \end{array} \right\} = \int_0^L \left(\frac{N(\xi) \delta N(\xi)}{EA(\xi)} + \frac{M_z(\xi) \delta M_z(\xi)}{EI_z(\xi)} + \frac{T_y(\xi) \delta T_y(\xi)}{GA_s(\xi)} \right) d\xi$$

- In plane strain state, one has:

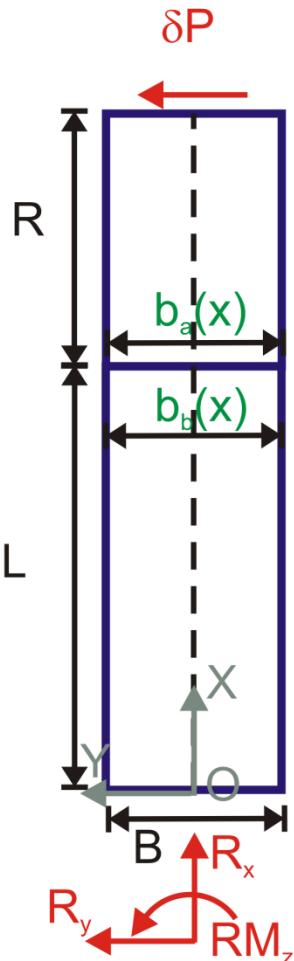
$$\left. \begin{array}{l} \delta P \times \Delta \\ \delta M \times \theta \end{array} \right\} = \int_0^L \left(\frac{(1 - \nu^2) N(\xi) \delta N(\xi)}{EA(\xi)} + \frac{(1 - \nu^2) M_z(\xi) \delta M_z(\xi)}{EI_z(\xi)} + \frac{T_y(\xi) \delta T_y(\xi)}{GA_s(\xi)} \right) d\xi$$

- The procedure to determine displacements by the unit virtual load method is :
 - Determine the stress resultant distribution N, T_y, M_z due to the actual loads.
 - Determine the stress resultant distribution $\delta N, \delta T_y, \delta M_z$ due to a unit virtual force/moment $\delta P/\delta M$ placed where the displacement/rotation Δ/θ is sought. It is set equal to unity for convenience.
 - Evaluate the right-hand side of the equation above by summing contributions from **all members of the structure**.
 - Because the virtual force/moment $\delta P/\delta M$ is set to unity, the result is the sought displacement or rotation Δ/θ .

A2. Stress resultants and displacement field

➤ Displacement field: Maximum horizontal displacement

- A virtual force is applied at the top of the structure



- Reaction forces:

$$R_x = 0 \quad R_y = -\delta P \quad RM_z = -\delta P(L + R)$$

- Stress resultants : $x \in [0, L + R]$

$$\delta N(x) = 0 \quad \delta T_y(x) = \delta P \quad \delta M_z(x) = \delta P(L + R - x)$$

- Displacement: $\delta P = 1$

- Beam a $v_{max}^{(a)} = \int_L^{L+R} \left(\frac{(1 - \nu_a^2) N(\xi) \delta N(\xi)}{E_a A_a(\xi)} \right) d\xi$

- Beam b $v_{max}^{(b)} = \int_0^L \left(\frac{(1 - \nu_b^2) N(\xi) \delta N(\xi)}{E_b A_b(\xi)} + \frac{(1 - \nu_b^2) M_z(\xi) \delta M_z(\xi)}{E_b I_{z_b}(\xi)} + \frac{T_y(\xi) \delta T_y(\xi)}{G_b A_{s_b}(\xi)} \right) d\xi$

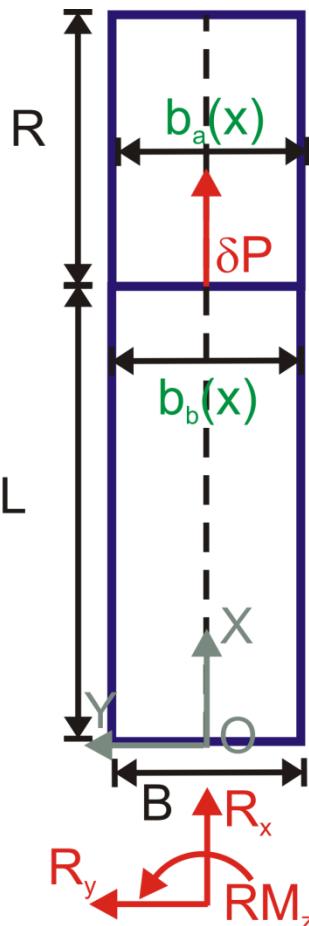
- Maximum horizontal displacement:

$$\Rightarrow v_{max} = v_{max}^{(a)} + v_{max}^{(b)} = 0.322 \text{ mm}$$

A2. Stress resultants and displacement field

➤ Displacement field: Maximum vertical displacement

- A virtual force is applied at the connection between beams a and b



- Reaction forces:

$$R_x = -\delta P \quad R_y = 0 \quad RM_z = 0$$

- Stress resultants: $x \in [0, L]$

$$\delta N(x) = \delta P \quad \delta T_y(x) = 0 \quad \delta M_z(x) = 0$$

- Displacement

$$\delta P = \frac{1}{\int_L^{L+R} \left(\frac{(1 - \nu_a^2) N(\xi) \delta N(\xi)}{E_a A_a(\xi)} \right) d\xi} = 0$$

- Beam a: $u^{(a)} = \int_L^L \left(\frac{(1 - \nu_a^2) N(\xi) \delta N(\xi)}{E_a A_a(\xi)} \right) d\xi = 0$

$$\begin{cases} \delta N(x) = 0 \\ x \in [L, L + R] \end{cases}$$

- Beam b: $u^{(b)} = \int_0^L \left(\frac{(1 - \nu_b^2) N(\xi) \delta N(\xi)}{E_b A_b(\xi)} \right) d\xi$

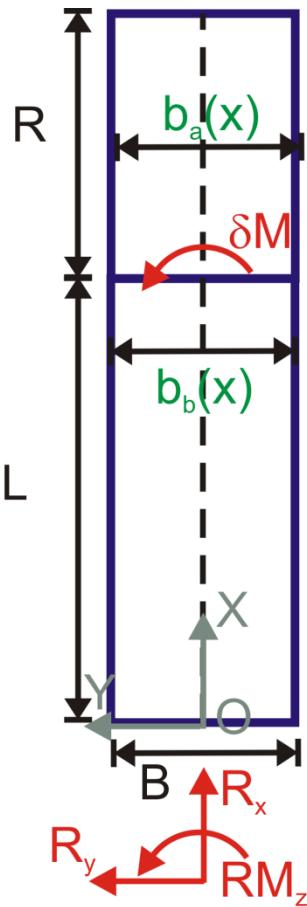
- Vertical displacement:

$$\Rightarrow u(L) = u^{(a)} + u^{(b)} = 0.00278 \text{ mm}$$

A2. Stress resultants and displacement field

➤ Displacement field: Maximum vertical displacement

- A virtual moment is applied at the connection between beams a and b



- Reaction forces:

$$R_x = 0 \quad R_y = 0 \quad RM_z = \delta M$$

- Stress resultants: $x \in [0, L]$

$$\delta N(x) = 0 \quad \delta T_y(x) = 0 \quad \delta M_z(x) = \delta M$$

- Rotation $\delta M = 1$

- Beam a: $\theta^{(a)} = \int_L^{L+R} \left(\frac{(1 - \nu_a^2) N(\xi) \delta N(\xi)}{E_a A_a(\xi)} \right) d\xi = 0 \quad \begin{cases} \delta N(x) = 0 \\ x \in [L, L + R] \end{cases}$

- Beam b: $\theta^{(b)} = \int_0^L \left(\frac{(1 - \nu_b^2) M_z(\xi) \delta M_z(\xi)}{E_b I_{z_b}(\xi)} \right) d\xi$

- Cross section rotation:

$$\Rightarrow \theta(L) = \theta^{(a)} + \theta^{(b)} = 0.000265 \text{ radians}$$

A2. Stress resultants and displacement field

➤ Displacement Field: Results

$$\Rightarrow v_{max} = v_{max}^{(a)} + v_{max}^{(b)} = 0.322 \text{ mm}$$

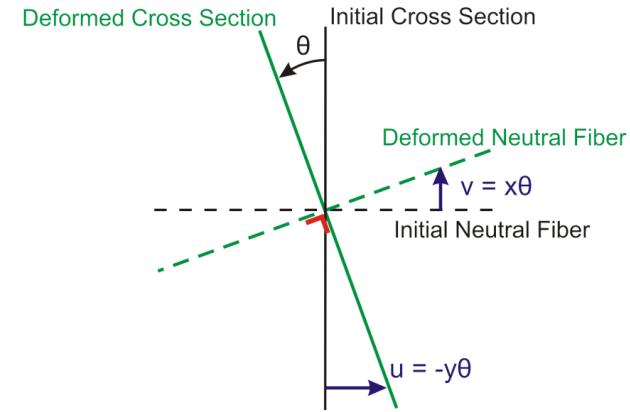
$$\Rightarrow u(L) = u^{(a)} + u^{(b)} = 0.00278 \text{ mm}$$

$$\Rightarrow \theta(L) = \theta^{(a)} + \theta^{(b)} = 0.000265 \text{ radians}$$

➤ Euler Bernoulli beam assumption:

- The cross-section remains plane after deformation.
- This implies an axial displacement field consisting of a rigid body translation $u(x)$, and a rigid body rotation $\theta(x)$ in 2D:

$$u(x, y) = u(x) - y\theta(x)$$



$$\Rightarrow u_{max} = u(x = L, y = -R) = u(L) + R\theta(L) = 0.0955 \text{ mm}$$

A2. Stress resultants and displacement field

➤ Range of validity

- Euler Bernoulli assumptions:
 - The cross section is infinitely rigid in its own plane (No Poisson effect !).
 - The cross section of a beam remains plane after deformation (No Warping).
 - The cross section remains normal to the deformed axis of the beam (Transverse shear strain field has been neglected ! : this effect is taken into account in the Timoshenko Beam Theory.).
- Tapered Beam:
 - The cross section variation has been neglected for stress computation : comparison between beam solution and exact solution for the wedge problem shows that the beam solution still gives good results as long as the cross section variation is relatively moderate.
- Stress Gradient/Concentration:
 - In our study, the stress gradient in the y direction has been totally neglected.
 - The Poisson effect has been totally neglected => a stress concentration will occur in the upper and lower beam layers at the restraint, because it prevents the cross section dilatancy or contraction.
- St. Venant's Principle:
 - The difference between the stress fields caused by statically equivalent load systems is insignificant at distances greater than the largest dimension of the area over which the loads are acting.
 - **Beam theory yields a 1D model along the beam longitudinal axis !**