

# FEM Mesh preparation and quality inspection

dr. Borut Černe

University of Ljubljana, Faculty of Mechanical Engineering

Univerza v Ljubljani

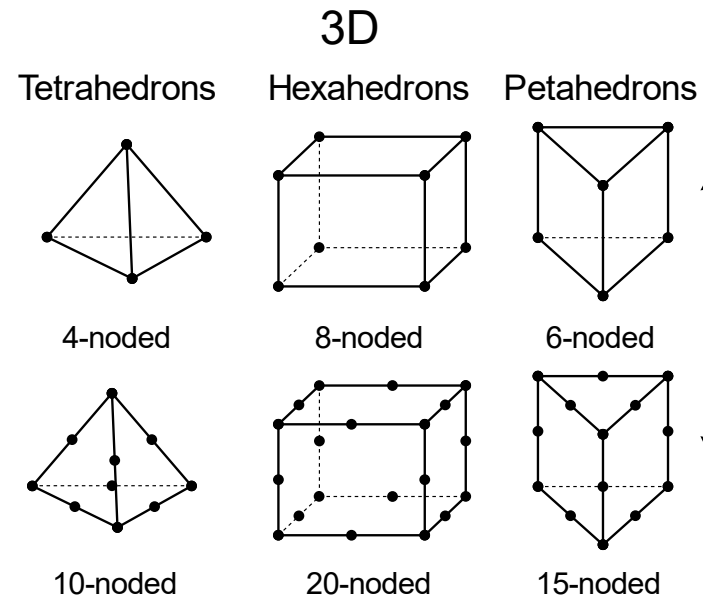
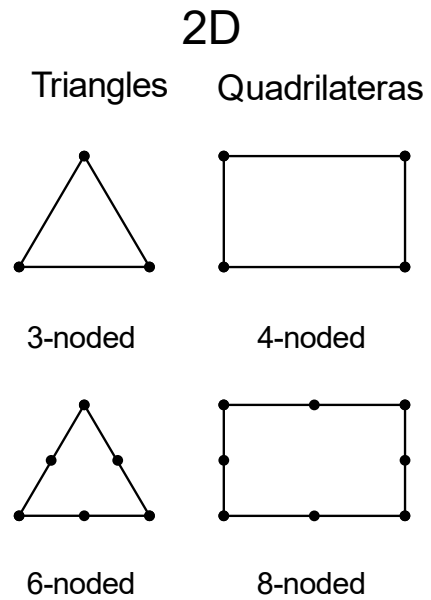
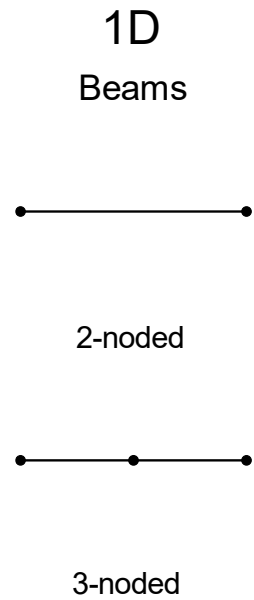


Co-funded by the  
Erasmus+ Programme  
of the European Union

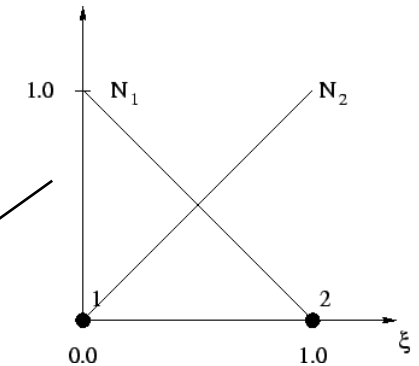
This project has been funded with support from the European Commission.

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

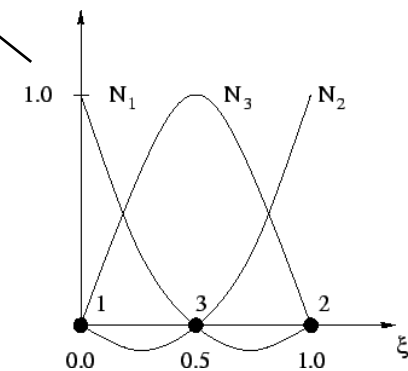
## Finite element (FE) types



Linear shape functions (SF):



Quadratic SF:



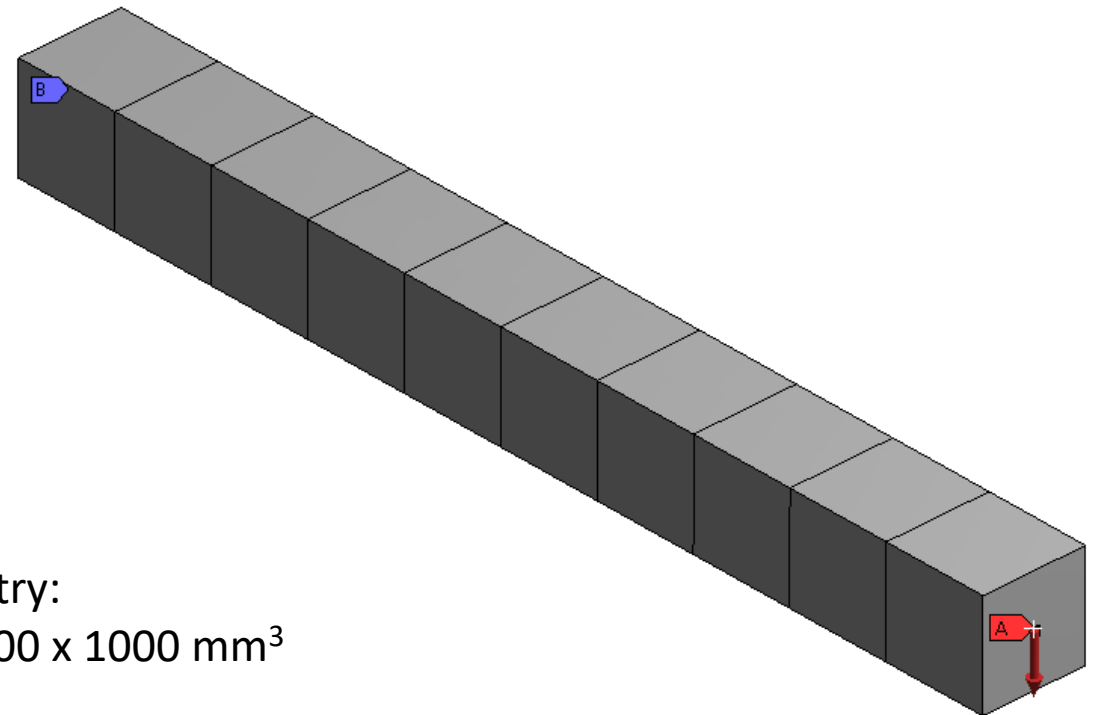
## Case 1: 1D FEs in 3D space

Details View	
[-] Details of Point2	
Point	Point2
Type	Construction Point
Definition	Manual Input
# Points generated	1
[-] Point Group 1 (RMB)	
<input type="checkbox"/> FD8, X Coordinate	1 m
<input type="checkbox"/> FD9, Y Coordinate	0 m
<input type="checkbox"/> FD10, Z Coordinate	0 m

Details of "Rect1"	
[-] Definition	
Type	RECT
Import Type	Imported
[-] Dimensions	
<input type="checkbox"/> B	100. mm
<input type="checkbox"/> H	100. mm
[-] Physical Properties	
Beam Section	Rect1
A	10000 mm <sup>2</sup>
Iyy	8.3333e+006 mm <sup>2</sup> .mm <sup>2</sup>
Izz	8.3333e+006 mm <sup>2</sup> .mm <sup>2</sup>

**A: beam 1D**  
Static Structural  
Time: 1. s  
24. 06. 2021 10:32

**A** Force: 300. N  
**B** Fixed Support



Geometry:  
100 x 100 x 1000 mm<sup>3</sup>

## Case 2: 3D FEs (quadratic element order) in 3D space

**Project\***

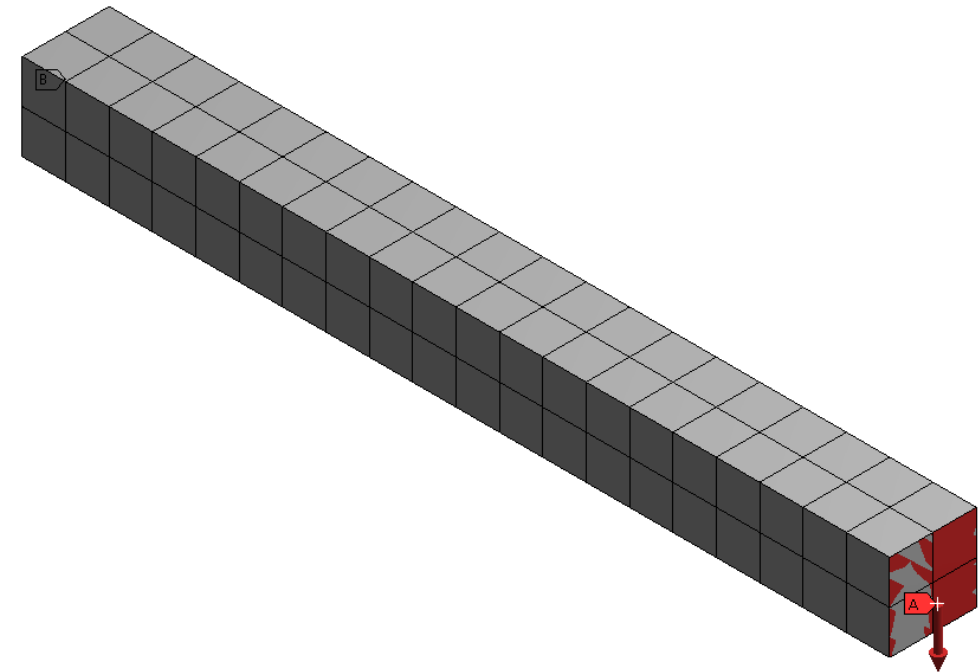
- Model (B4)
  - Geometry
  - Materials
  - Coordinate Systems
  - Mesh
  - Static Structural (B5)
    - Analysis Settings
    - Remote Displacement
    - Force
  - Solution (B6)
    - Solution Information
    - Equivalent Stress

Details of "Mesh"

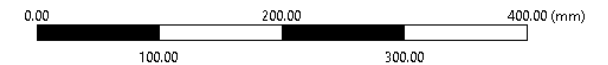
<b>Display</b>	
Display Style	Use Geometry Setting
<b>Defaults</b>	
Physics Preference	Mechanical
Element Order	Quadratic
<input type="checkbox"/> Element Size	Default
<b>Sizing</b>	
<b>Quality</b>	
<b>Inflation</b>	
<b>Advanced</b>	
<b>Statistics</b>	

**B: beam 3d**  
Static Structural  
Time: 1. s  
24.06.2021 10:37

**A** Force: 300. N  
**B** Remote Displacement



Geometry:  
 $100 \times 100 \times 1000 \text{ mm}^3$



# FE Mesh preparation and quality

## Case 3: 2D shell FEs in 3D space

**Project\***

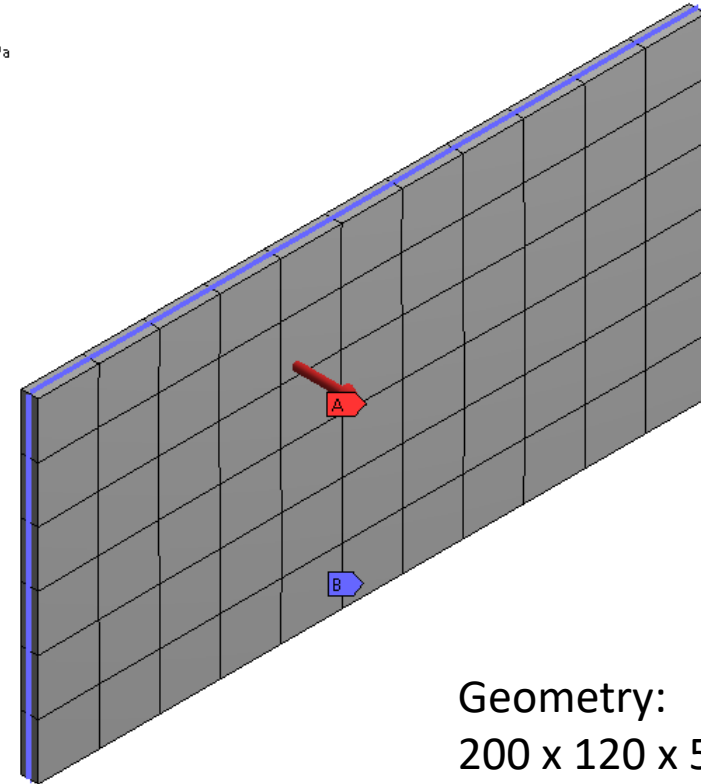
- Model (C4)
  - Geometry
  - SYS-3\Surface
  - Materials
    - Structural Steel
  - Coordinate Systems
    - Global Coordinate System
  - Mesh
  - Static Structural (C5)
    - Analysis Settings
    - Fixed Support
    - Pressure
  - Solution (C6)
    - Solution Information
    - Directional Deformation

Details of "SYS-3\Surface"

<b>Graphics Properties</b>	
<b>Definition</b>	
<input type="checkbox"/> Suppressed	No
Dimension	3D
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
<input type="checkbox"/> Thickness	5. mm
Thickness Mode	Manual
Offset Type	Middle
Treatment	None

C: plate - 3D shell  
Static Structural  
Time: 1. s  
24. 06. 2021 10:40

A Pressure: 0.1 MPa  
B Fixed Support



Geometry:  
200 x 120 x 5 mm<sup>3</sup>

## Case 4: 2D FEs in 2D space (plane stress and plane strain)

**Project**

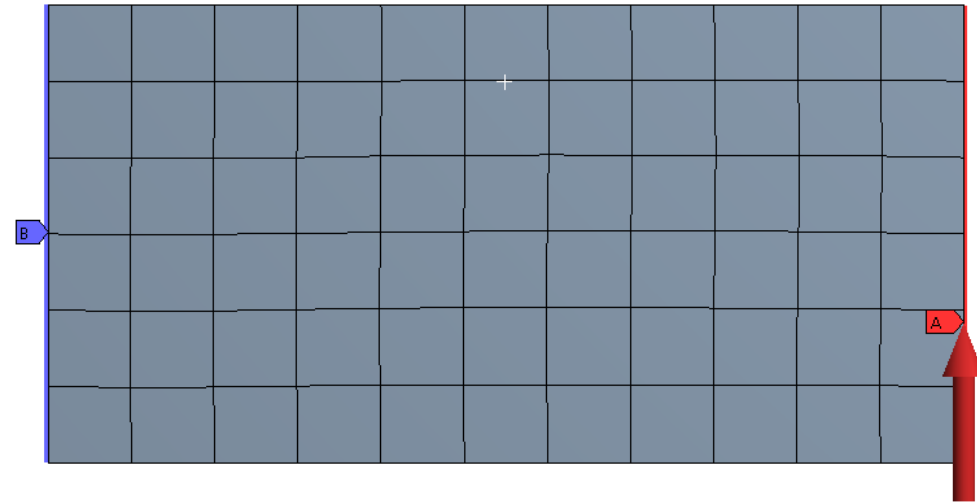
- Model (D4)
  - Geometry
  - SYS-4\Surface
  - Materials
  - Coordinate Systems
  - Mesh
  - Static Structural (D5)
    - Analysis Settings
    - Pressure
    - Fixed Support
  - Solution (D6)
    - Solution Information
    - Directional Deformation

**Details of "Geometry"**

Definition	
Source	E:\Projekti\2021_Erasmus_plu...
Type	SpaceClaim
Length Unit	Meters
Element Control	Program Controlled
2D Behavior	Plane Stress
Display Style	Body Color
Bounding Box	
Properties	
Statistics	
Update Options	
Basic Geometry Options	
Advanced Geometry Options	
CAD Attributes	
Model Assembly Output Version	1

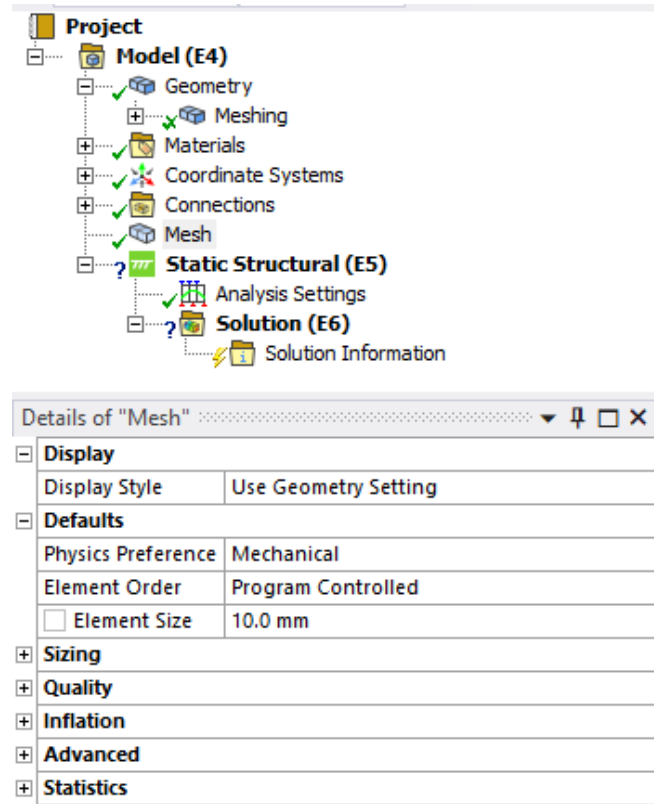
D: plate - 2D plane stress  
Static Structural  
Time: 1. s  
24. 06. 2021 10:42

A Pressure: 0.1 MPa  
B Fixed Support



Geometry:  
200 x 120 x 5 mm<sup>3</sup>

## Case 5: Structured 3D FE mesh



The screenshot shows the ANSYS Workbench interface. On the left is the Project tree, and on the right is the 'Details of "Mesh"' panel.

**Project Tree:**

- Project
  - Model (E4)
    - Geometry
    - Meshing
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
    - Static Structural (E5)
      - Analysis Settings
      - Solution (E6)
        - Solution Information

**Details of "Mesh" Panel:**

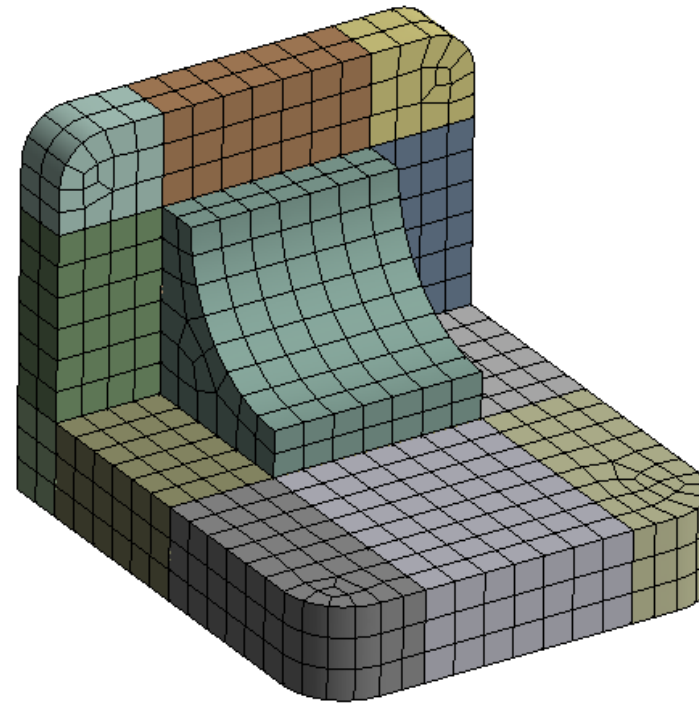
Display	
Display Style	Use Geometry Setting

Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	10.0 mm

<b>Sizing</b>
<b>Quality</b>
<b>Inflation</b>
<b>Advanced</b>
<b>Statistics</b>



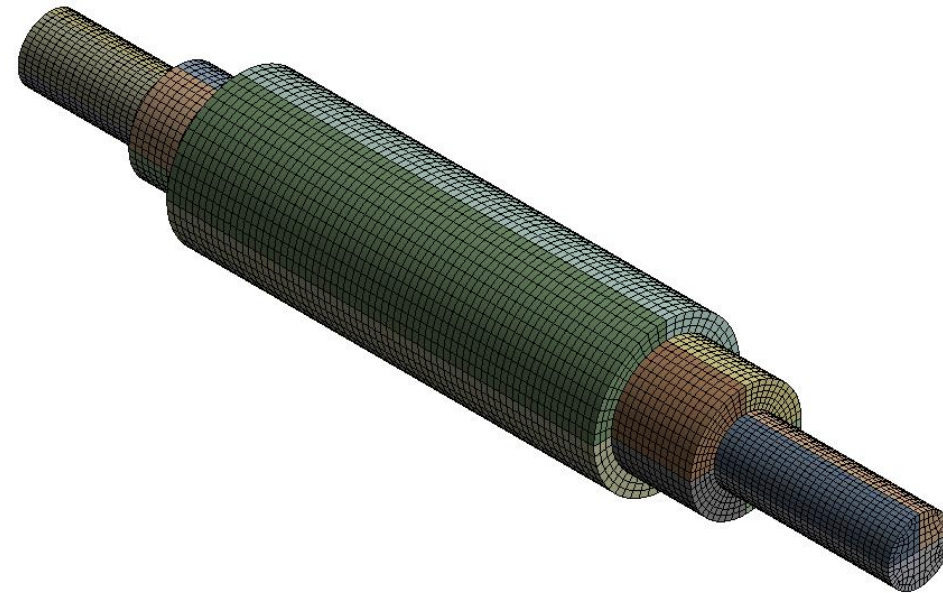
- Case 6: Structured 3D FE mesh (cylindrical bodies)

The screenshot shows the ANSYS Workbench project tree on the left and the 'Details of "Mesh"' panel on the right. The project tree includes:

- Project
  - Model (H4)
    - Geometry
    - Materials
    - Coordinate Systems
    - Connections
    - Mesh
  - Static Structural (H5)
    - Analysis Settings
    - Solution (H6)
      - Solution Information

The 'Details of "Mesh"' panel shows the following settings:

Details of "Mesh"	
<b>Display</b>	
Display Style	Use Geometry Setting
<b>Defaults</b>	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	2.0 mm
<b>Sizing</b>	
<b>Quality</b>	
<b>Inflation</b>	
<b>Advanced</b>	
<b>Statistics</b>	





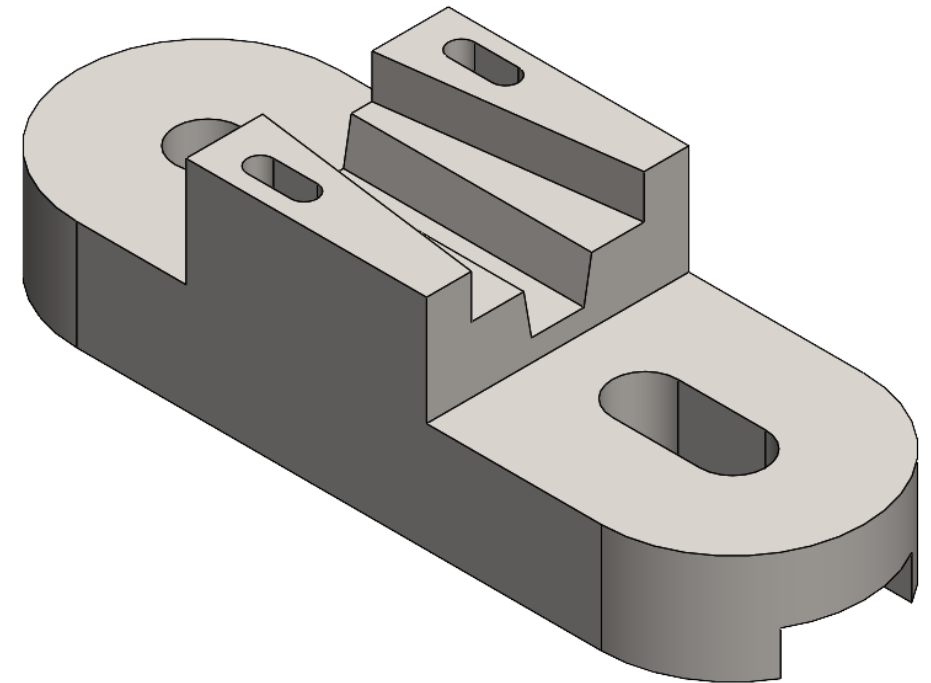
## Individual work case 1: Structured 3D FE mesh (wheel)

- Upload geometry from file  
*Wheel\_Meshing\_defeatured.x\_t*
- Try to reach at least 80 % element quality using  
(predominantly) hexagonal FEs
- Hint – use either
  - Geometry slicing or
  - Mesh method – Hex dominant or
  - Both methods combined



Individual work case 1: Structured 3D FE mesh (3D body)

- Upload geometry from file *Meshing\_block\_2.x\_t*
- Try to reach at least 80 % element quality using (predominantly) hexagonal FEs
- Hint – use either
  - Geometry slicing or
  - Mesh method – Hex dominant or
  - Both methods combined



Thank you for your attention!

<http://sctrain.eu/>

Univerza v Ljubljani



TECHNISCHE  
UNIVERSITÄT  
WIEN



VSB TECHNICAL  
UNIVERSITY  
OF OSTRAVA

IT4INNOVATIONS  
NATIONAL SUPERCOMPUTING  
CENTER



Co-funded by the  
Erasmus+ Programme  
of the European Union

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

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