

# Prutová úloha – příhrada

Ing. Petr Lehner

# Co se dozvíme a naučíme?

*Tvorba geometrie pomocí kombinace střednic a průřezů.*

*Provázání různých komponent (úloh).*

*Možnosti změny zobrazení jednotek a škály výsledků.*

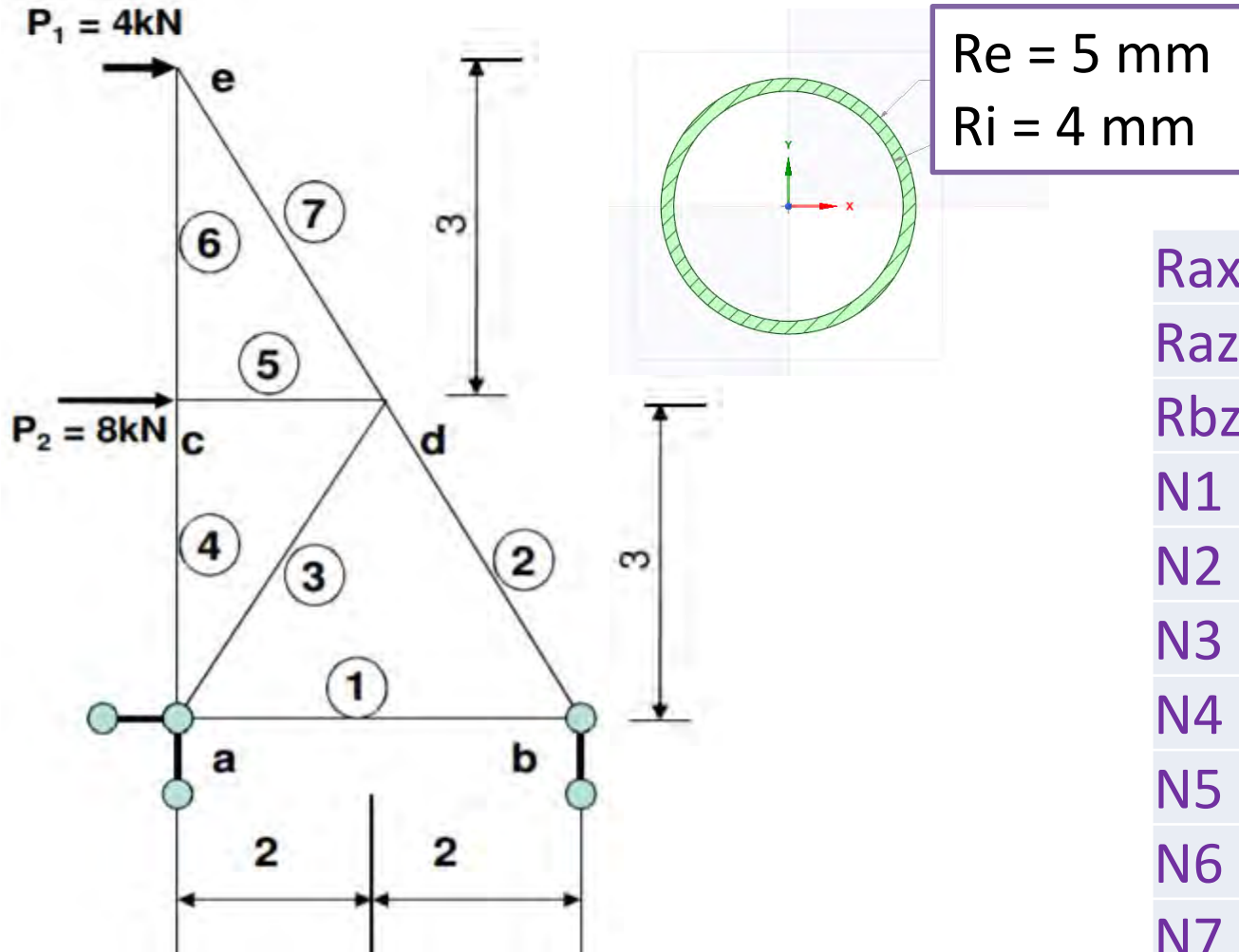
*Využití MKP analýzy pro návrh profilů.*

*Jaké jsou rozdíly výsledků ručního výpočtu a MKP analýzy.*

# Příhradová konstrukce

Ruční výpočet styčnickovou metodou:

[http://fast10.vsb.cz/michalcova/Statika19/10\\_19\\_prihrady.pdf](http://fast10.vsb.cz/michalcova/Statika19/10_19_prihrady.pdf)



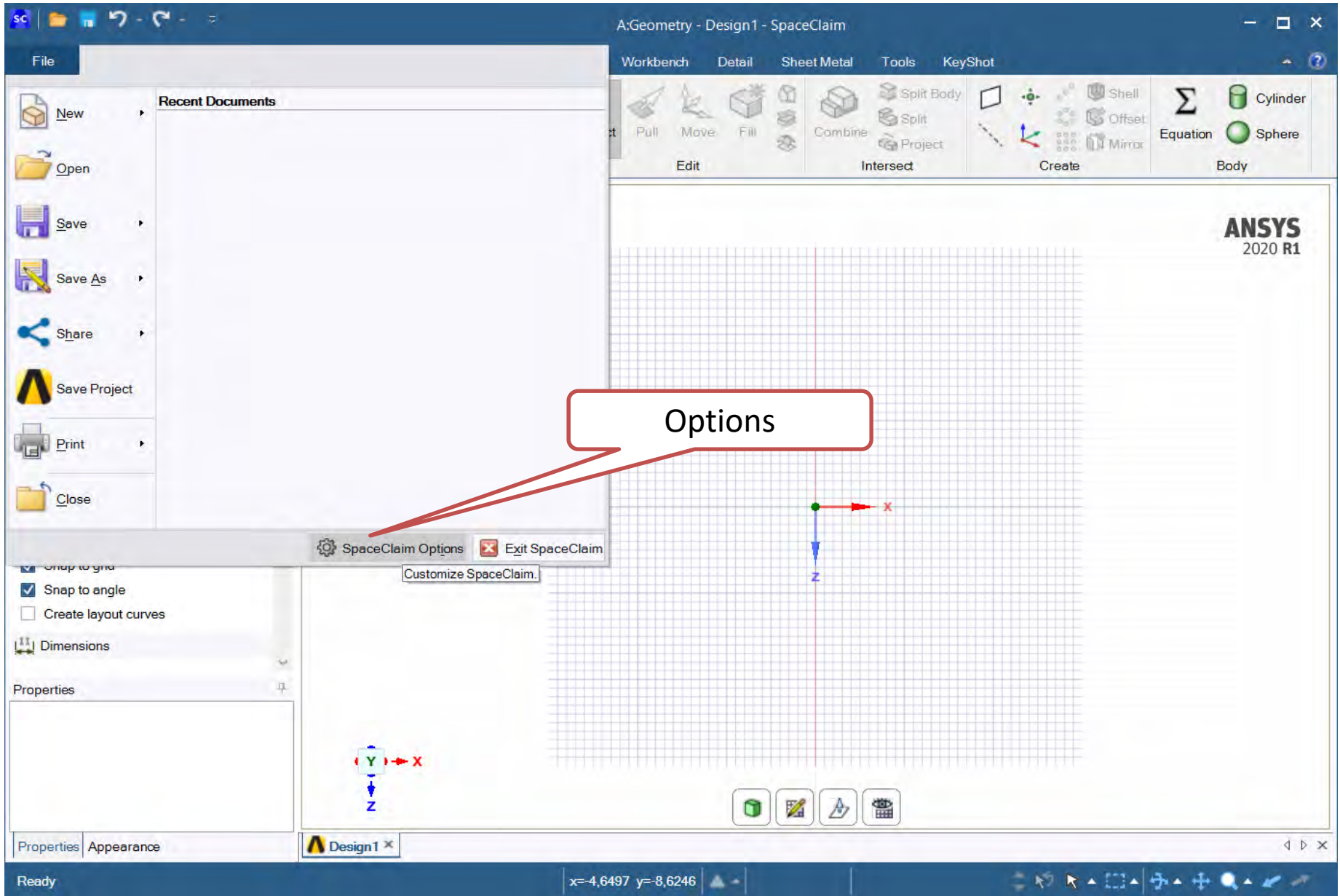
$R_{ax}$	12	kN
$R_{az}$	12	kN
$R_{bz}$	12	kN
$N_1$	8	kN
$N_2$	14,42	kN
$N_3$	7,21	kN
$N_4$	6	kN
$N_5$	8	kN
$N_6$	6	kN
$N_7$	7,21	kN

# Ansys Workbench – Geometry

The screenshot displays the Ansys Workbench interface. On the left, the 'Component Systems' list includes 'Geometry'. A red callout box points to this entry with the text 'Geometry'. In the center, the 'Project Schematic' shows a hierarchy with 'A' containing two 'Geometry' components. A red callout box points to the second 'Geometry' component with the text 'Line Bodies'. On the right, the 'Properties of Schematic A2: Geometry' window is open, showing a table of properties. A red callout box points to the 'Line Bodies' property in row 11, which is checked.

	A	B
1	Property	Value
2	General	
3	Component ID	Geometry
4	Directory Name	Geom
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Basic Geometry Options	
10	Solid Bodies	<input checked="" type="checkbox"/>
11	Surface Bodies	<input checked="" type="checkbox"/>
12	Line Bodies	<input checked="" type="checkbox"/>
13	Parameters	Independent
14	Parameter Key	ANS;DS
15	Attributes	<input type="checkbox"/>
16	Named Selections	<input type="checkbox"/>
17	Material Properties	<input type="checkbox"/>
18	Advanced Geometry Options	
19	Analysis Type	3D
20	Use Associativity	<input checked="" type="checkbox"/>
21	Import Coordinate Systems	<input type="checkbox"/>
22	Import Work Points	<input type="checkbox"/>
23	Reader Mode Saves Updated File	<input type="checkbox"/>
24	Import Using Instances	<input checked="" type="checkbox"/>
25	Smart CAD Update	<input checked="" type="checkbox"/>
26	Compare Parts On Update	No
27	Enclosure and Symmetry Processing	<input checked="" type="checkbox"/>
28	Decompose Disjoint Geometry	<input checked="" type="checkbox"/>
29	Clean Geometry On Import	<input type="checkbox"/>
30	Stitch Surfaces On Import	None
31	Mixed Import Resolution	None

# Space Claim - nastavení jednotek



# Space Claim - nastavení – units - metry

SpaceClaim Options

Change units options.

Units settings for: This Document

**Units**

Type: Metric

Length: Millimeters

Decimal/fraction: Decimal

Angle: Degrees

Mass: Grams

Density: Derived

Grams / mm<sup>3</sup>

Symbol: mm

Primary precision: 2  Show trailing zero

Angular precision: 1  Show trailing zero

Use tight tolerances ⓘ

Show symbol in user interface

Show symbol in annotations

Show "-" separator

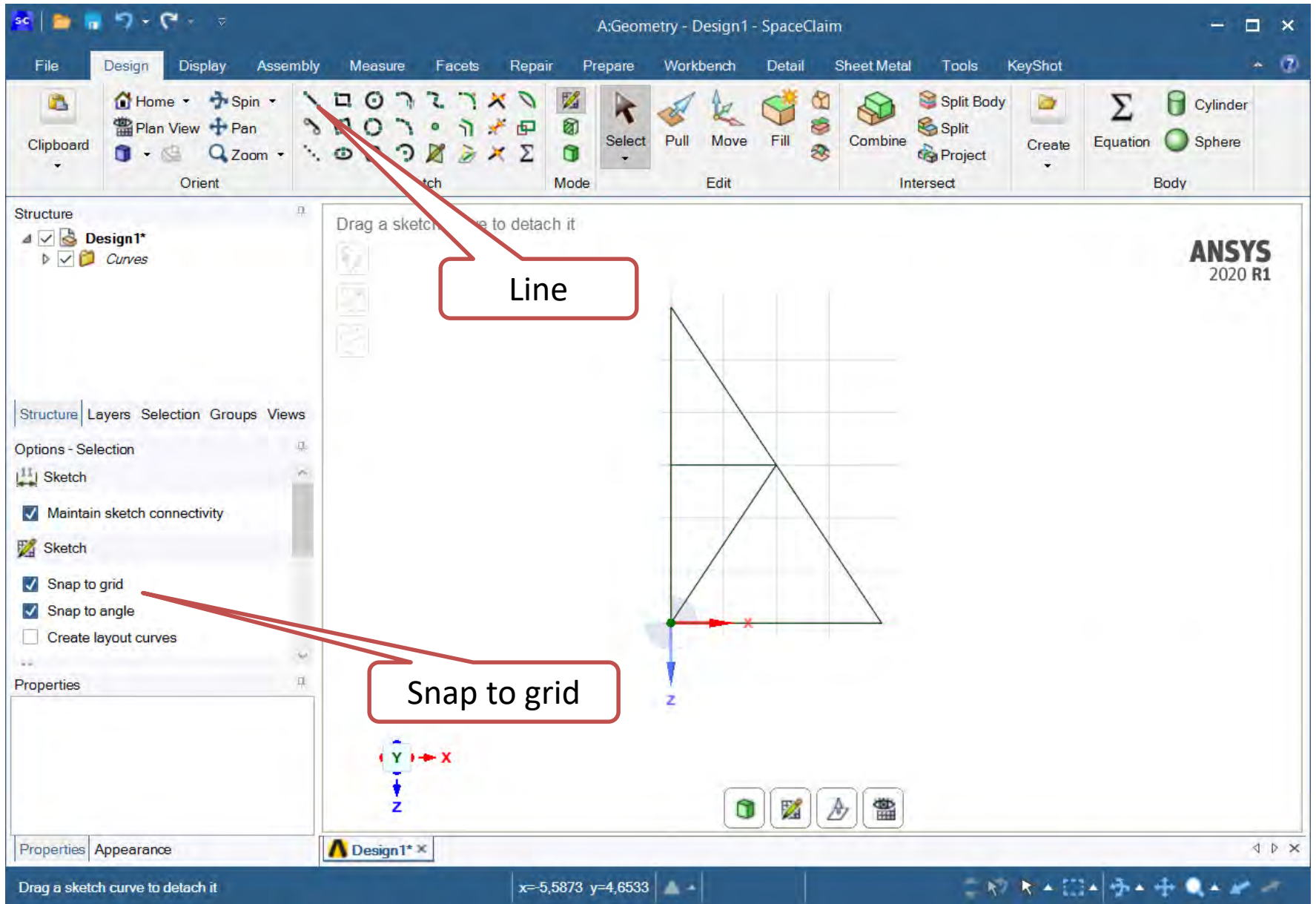
Extended dimensions

Use dual dimensions

OK Cancel



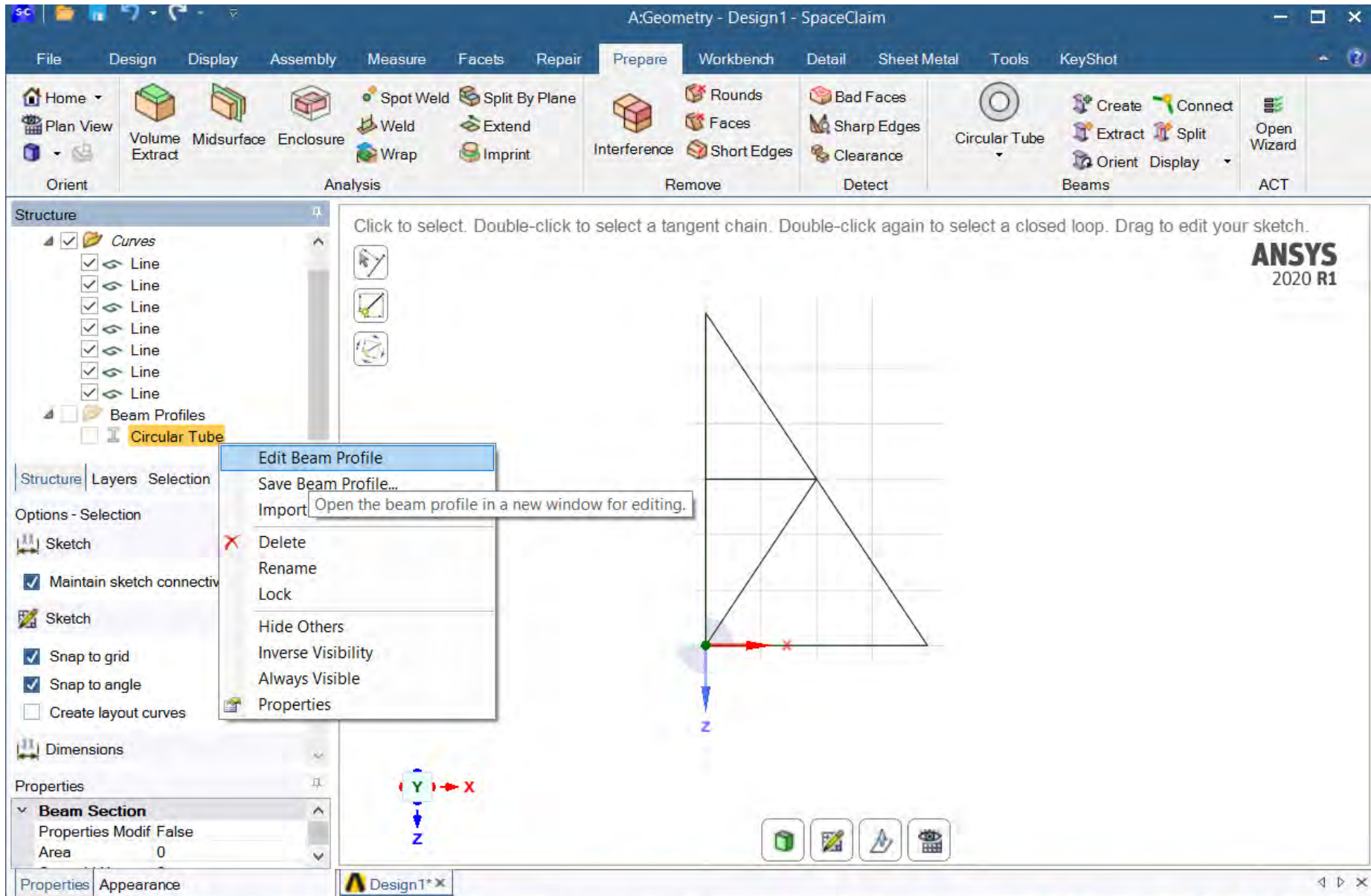
# Space Claim - Line







# Prepare – Beam – Edit



# Ruler dimension Ro – 0.005 a Ri – 0.004

The screenshot shows the ANSYS 2020 R1 software interface. The main window displays a 3D model of a circular ring with a green hatched texture. The ring is centered on a coordinate system with X, Y, and Z axes. The software interface includes a top menu bar with options like File, Design, Display, Assembly, Measure, Facets, Repair, Prepare, Workbench, Detail, Sheet Metal, Tools, and KeyShot. Below the menu bar is a toolbar with various icons for selection, editing, and creating. On the left side, there is a 'Groups' panel showing a tree view of the model's structure. Under 'Driving Dimensions', two dimensions are listed: 'Ro' with a value of '0.01m' and 'Ri' with a value of 'Ruler dimension'. A red callout box points to these dimensions, containing the text: 'Vnější poloměr 0,005 m, poloměr 0,004 m'. Another red callout box at the bottom right contains the text: 'Zavřít'. The bottom status bar shows the text: 'Select and drag a face to offset it. Select and drag an edge to round it.'

Vnější poloměr  
0,005 m,  
poloměr 0,004  
m

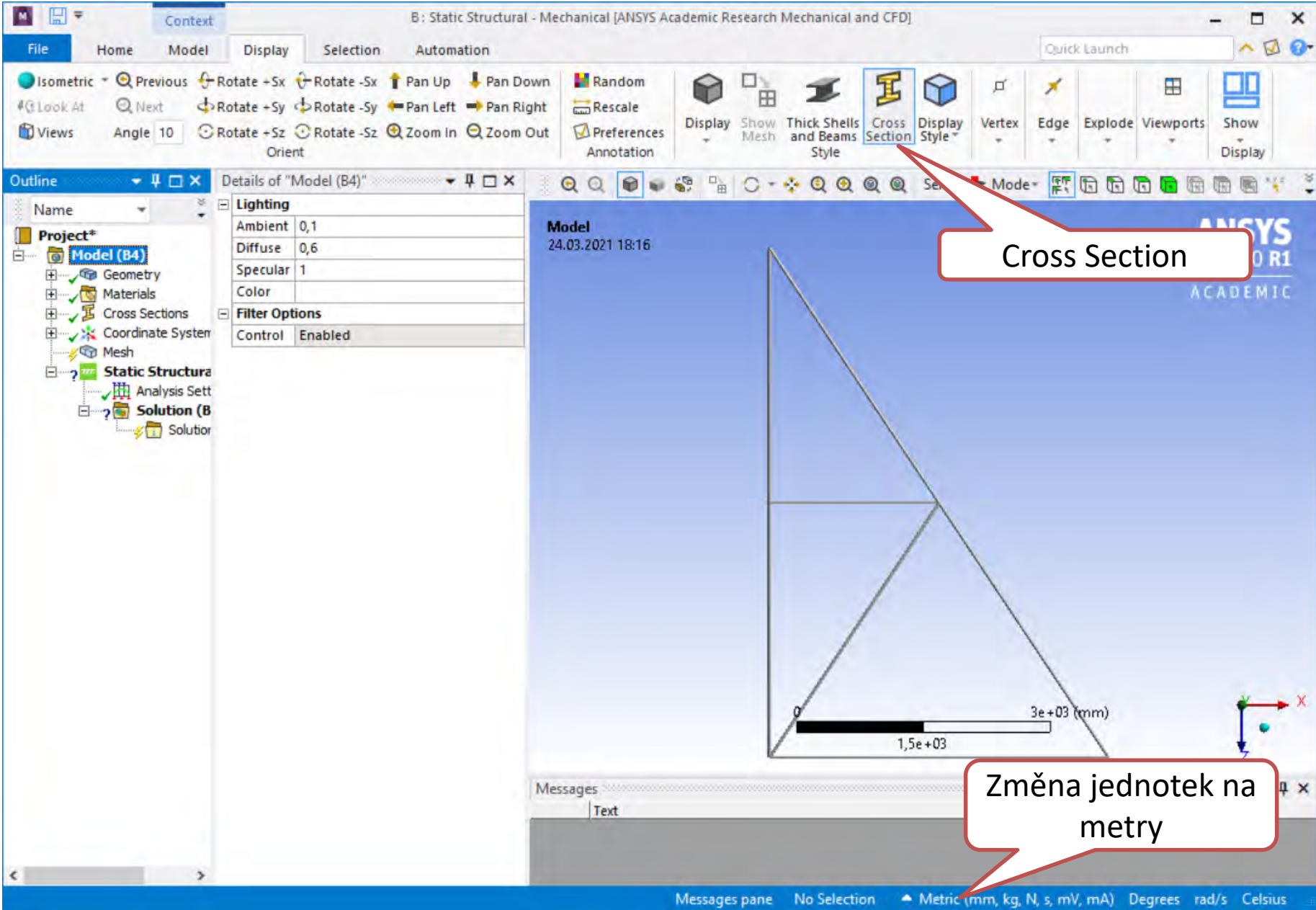
Zavřít



# Prepare – Create

The screenshot displays the SpaceClaim software interface. The ribbon at the top shows the 'Prepare' tab, which includes the 'Create' tool. A red callout box labeled 'Create' points to this tool. On the left, the 'Structure' tree shows a list of 'Line' objects under the 'Curves' folder, with a red callout box labeled 'Označit všechny pruty' (Mark all edges) pointing to this list. The central workspace shows a green triangular sketch on a grid. A dimension of (3,61m) is shown for one of the edges. A coordinate system with X, Y, and Z axes is visible at the bottom left. A tooltip for the 'Create' tool is open on the right, providing instructions: 'Create a beam along an edge, along a curve, or between two points. Points can be vertices, midpoints, or the intersections of planes and edges.' and 'Press F1 for more help.'

# Model – Mechanical – Cross Section



# Mesh – Generate

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window title is "B: Static Structural - Mechanical [ANSYS Academic Research Mechanical and CFD]". The "Mesh" tab is active in the top ribbon, showing various meshing tools like "Generate", "Update", "Surface Mesh", and "Method".

The "Details of 'Mesh'" panel is open, showing the following settings:

- Display:** Display Style: Use Geometry Setting
- Defaults:** Physics Preference: Mechanical; Element Order: Program Controlled;  Element Size: Default
- Sizing:** (Expanded)
- Quality:** (Expanded)
- Inflation:** (Expanded)
- Batch Connections:** (Expanded)
- Advanced:** (Expanded)
- Statistics:** (Expanded)

The 3D view shows a triangular mesh structure. A scale bar at the bottom indicates dimensions from 0,000 to 3,000 (m), with a midpoint at 1,500. The ANSYS 2020 R1 ACADEMIC logo is visible in the top right corner of the 3D view.

The "Messages" panel at the bottom shows a text message: "The default Error Limit for Mechanical Physics Preference has changed to Aggressive Mech Project>Model>Mesh".



# Static Structural – Fixed Support

The screenshot displays the ANSYS Mechanical interface for a static structural analysis. The main window shows a truss structure with a fixed support at the bottom-left vertex. A red callout box points to the 'Fixed Support' icon in the toolbar and contains the text 'Fixed Support > vertex'. The 'Details of "Fixed Support"' panel is open, showing the following information:

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex

Definition	
Type	Fixed Support
Suppressed	No

The 'B: Static Structural' panel shows 'Fixed Support' with a time of 1 s and a date of 24.03.2021 18:20. A scale bar at the bottom of the model indicates dimensions of 0,000, 1,500, and 3,000 (m). The status bar at the bottom shows 'Ready', '1 Message', 'No Selection', and 'Metric (m, kg, N, s, V, A) Degrees rad/s Celsius'.

Fixed Support  
> vertex

# Static Structural – Displacement

**Displacement**  
Components: Free;0,0,0, m

**Displacement > vertex**

X Free  
Y a Z = 0  
(pozor na osy)

Steps	Time [s]	Y [m]	Z [m]
1	0,0	= 0,0	= 0,0
2	1,0	0,0	0,0



# Static Structural – Force

The screenshot displays the ANSYS Mechanical interface for a static structural analysis. The main view shows a truss structure with a force of 4000 N applied at the top vertex. The force is defined by its components: X = 4000 N, Y = 0 N, and Z = 0 N. The structure is supported by a fixed support at the bottom left corner. The dimensions of the structure are 0,000 m, 1,500 m, and 3,000 m.

**Force**

**Define by**  
**> Components**  
**> X = 4000 N**

Steps	Time [s]	X [N]	Y [N]	Z [N]
1	0	= 0	= 0	= 0
2	1	4000	0	0

# Static Structural – Force

The screenshot displays the ANSYS 2020 R1 Academic interface for a static structural analysis. The main view shows a truss structure with a force applied to its top-left node. The force is defined as 8000 N in the X direction. The details panel shows the force definition, and the tabular data table shows the force components over time.

**Force**

**Define by**  
> Components  
> X = 8000 N

Steps	Time [s]	X [N]	Y [N]	Z [N]
1	0.	= 0.	= 0.	= 0.
2	1.	8000.	0.	0.



# Solution – insert – parameter - SOLVE

**Solution:**

- > Total deformation
- > Axial Force
- > Beam Tool
- > Force Reaction (pro obě podpory)

Axis	Value
X Axis	-574,99 N
Y Axis	-4,3087e-010
Z Axis	0, N
Total	574,99 N

Time [s]	Force Reaction 2 (X) [N]
1	-574.99



# Force Reaction – Fixed

The screenshot displays the ANSYS Academic Teaching Mechanical and CFD software interface. The main window shows a 3D model of a structure with a force reaction arrow pointing downwards. The software title is "B: Static Structural - Mechanical [ANSYS Academic Teaching Mechanical and CFD]". The toolbar includes options like Duplicate, Solve, Analysis, and Display. The Outline tree on the left shows the project structure, including Model (B4), Static Structural (B5), and Solution (B6). The Details panel for "Force Reaction" shows the following information:

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Fixed Support
Orientation	Global Coordinate ...
Suppressed	No

Options	
Result Selection	All
<input type="checkbox"/> Display Time	End Time

Results	
Maximum Value Over Time	
<input type="checkbox"/> X Axis	-12000 N
<input type="checkbox"/> Y Axis	-3,4191e-039 N
<input type="checkbox"/> Z Axis	12000 N
<input type="checkbox"/> Total	16970 N
Minimum Value Over Time	
<input type="checkbox"/> X Axis	-12000 N
<input type="checkbox"/> Y Axis	-3,4191e-039 N
<input type="checkbox"/> Z Axis	12000 N
<input type="checkbox"/> Total	16970 N

Information	
-------------	--

Two red callout boxes provide additional information:

- Bottom-left callout: "Výsledky odpovídají ručnímu výpočtu – hodnota Y téměř 0." (Results correspond to manual calculation – value Y is almost 0.)
- Bottom-right callout: "Reakce dle os. Zobrazená výslednice." (Reaction by axis. Shown resultant.)

# Force Reaction – Displacement

The screenshot shows the ANSYS Academic 2020 R1 interface. The main window displays a truss structure with a force reaction at the bottom right. The 'Details of "Force Reaction 2"' panel is open, showing the following data:

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Displacement
Orientation	Global Coordinate ...
Suppressed	No

Options	
Result Selection	All
<input type="checkbox"/> Display Time	End Time

Results	
Maximum Value Over Time	
<input type="checkbox"/> X Axis	0, N
<input type="checkbox"/> Y Axis	0, N
<input type="checkbox"/> Z Axis	-12000 N
<input type="checkbox"/> Total	12000 N

Minimum Value Over Time	
<input type="checkbox"/> X Axis	0, N
<input type="checkbox"/> Y Axis	0, N
<input type="checkbox"/> Z Axis	-12000 N
<input type="checkbox"/> Total	12000 N

Information	
-------------	--

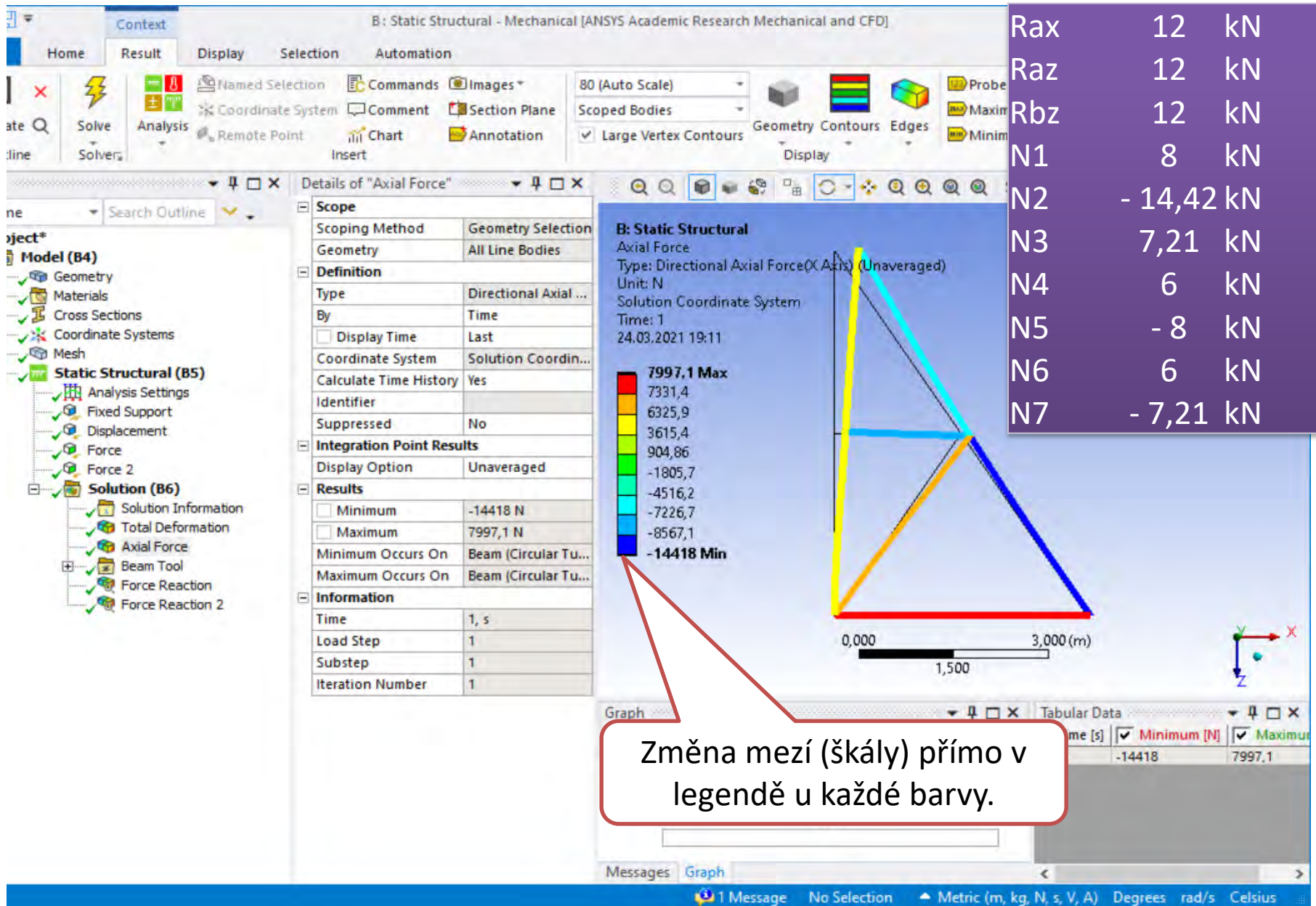
A callout box points to the 'Total' reaction value of 12000 N. The graph below shows the reaction force over time, with a single data point at 1.0 s showing a value of 0.0 N. The 'Tabular Data' panel shows the following data:

Time [s]	Force Reaction 2 (X) [N]
1.0	0.0

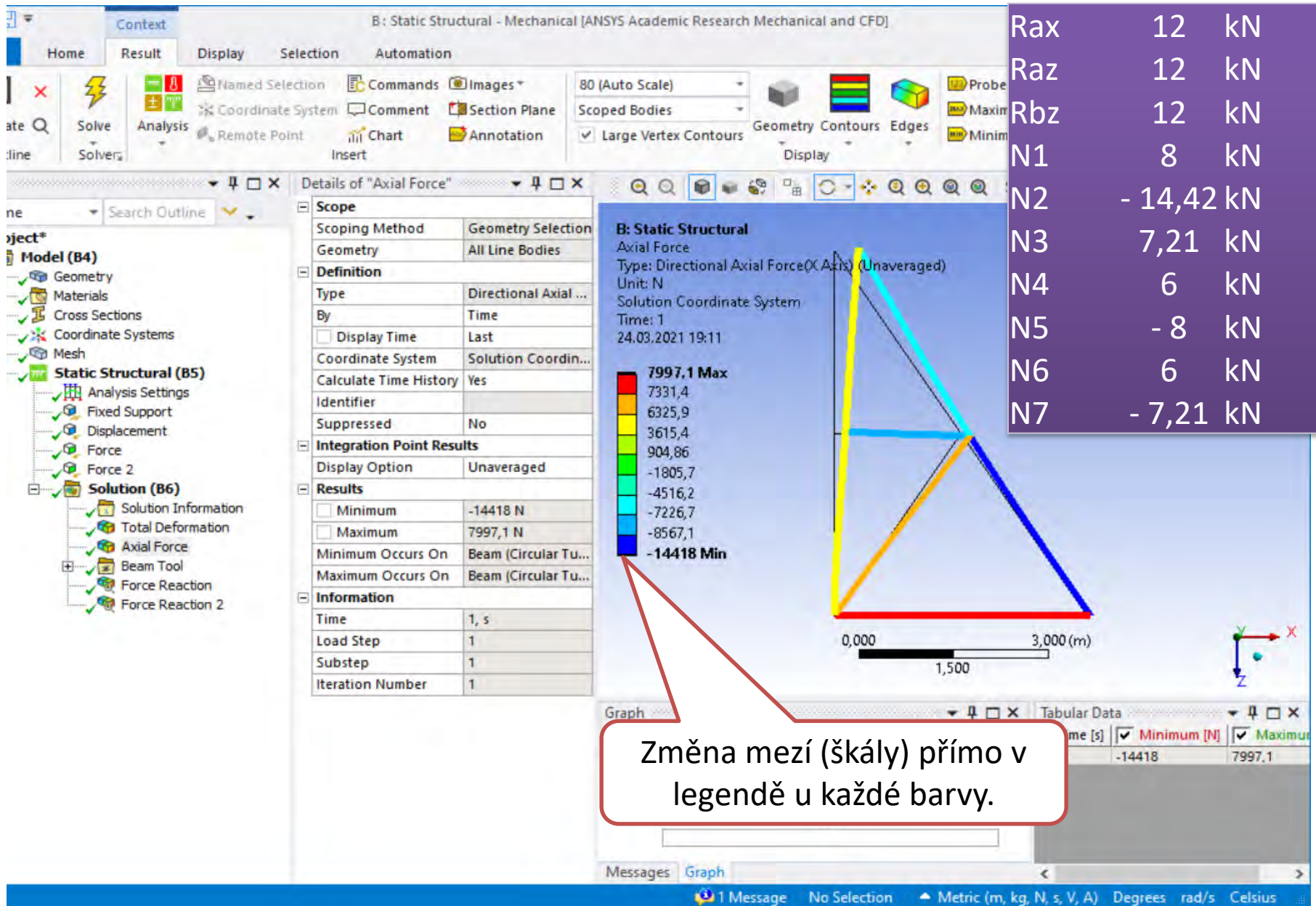
Výsledky odpovídají ručnímu výpočtu.



# Axial Force



# Axial Force





# Total deformation

**B: Static Structural**  
 Total Deformation  
 Type: Total Deformation  
 Unit: m  
 Time: 1  
 27.03.2021 11:59

**0,040608 Max**  
 0,036096  
 0,031584  
 0,027072  
 0,02256  
 0,018048  
 0,013536  
 0,009024  
 0,004512  
**0 Min**

0,000 1,500 3,000 (m)

Time [s]	Minimum [m]	Maximum [m]
1	0	4,0608e-002

$$\delta = \sum_{i=1}^n \frac{N_i \bar{N}_i}{EA_i} l_i$$



# Beam Tool – Combinet Stress

**Details of \*Maximum Combine**

Definition	
Type	Maximum Combi...
By	Time
<input type="checkbox"/> Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Results	
<input type="checkbox"/> Minimum	-510,21 MPa
<input type="checkbox"/> Maximum	283,21 MPa
<input type="checkbox"/> Average	0,71252 MPa
<input checked="" type="checkbox"/> Minimum Occurs On	Beam (Circular Tu...
<input checked="" type="checkbox"/> Maximum Occurs On	Beam (Circular Tu...
Information	

**B: Static Structural**  
Maximum Combined Stress  
Type: Maximum Combined Stress  
Unit: MPa  
Time: 1  
27.03.2021 12:05

**283,21 Max**  
195,05  
106,89  
18,735  
-69,422  
-157,58  
-245,74  
-333,89  
-422,05  
**-510,21 Min**

0 3e+03 (mm)  
1,5e+03

**Tah = 283 Mpa**  
**Tlak = 510 Mpa**

**Změna jednotek na mm > MPa**

Graph [MPa] [s]

# Cross Sections – Circular Tube – změna rozměrů

**Details of "Circular Tube"**

Definition	
Type	CTUBE
Import Type	Manual
Dimensions	
<input type="checkbox"/> Ri	9, e-003 m
<input type="checkbox"/> Ro	1, e-002 m
Physical Properties	
Beam Section	Circular Tube
A	5, 000 m <sup>2</sup>
Iyy	0, 009 m <sup>2</sup> ·m <sup>2</sup>
Izz	0, 009 m <sup>2</sup> ·m <sup>2</sup>

**Circular Tube**  
24.03.2021 19:41

0,000 1,500 3,000 (m)

**ANSYS 2020 R1 ACADEMIC**

**Změna vnějšího a vnitřního průměru**

**Projděte si znovu všechny výsledky. Všimněte si, které se změnily, a které ne.**

1 Message No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsius