

Vzpěr, vlastní tvar

Ing. Petr Lehner

Co se dozvíme a naučíme?

Spojení geometrie a modelu dvou úloh a využití výsledků.

Tvorba geometrie dle vnitřní knihovny.

Analýza kritické síly vybočení.

Vyhodnocení vlastních tvarů vybočení prutové konstrukce.

Porovnání ručního a MKP modelu.

Vzpěr

Tlačené štíhlé pruty jsou ohroženy ztrátou stability

- Prut zůstává přímý až do kritické hodnoty zatížení
- Po překročení kritické hodnoty může prut vybočit

Vzpěr = namáhání prutu tlakovou osovou silou

Vybočení ideálně přímého prutu:

Eulerova geometrická metoda

Předpoklady řešení:

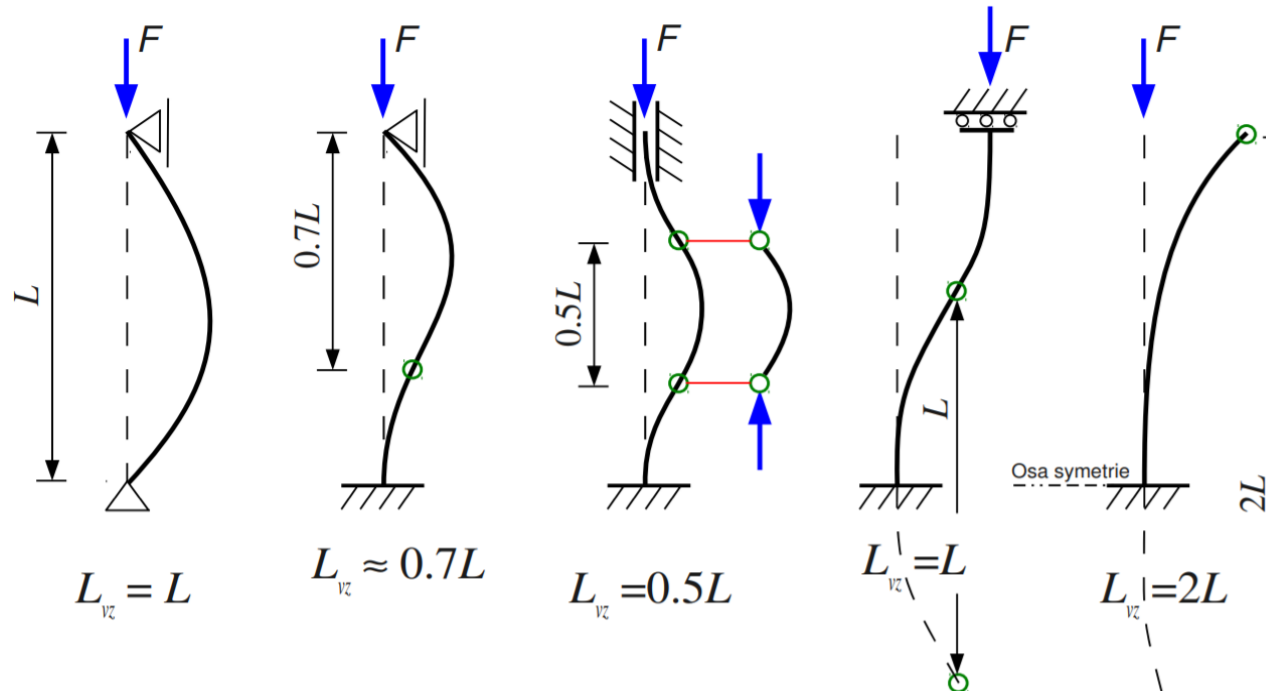
- ideálně pružný materiál
- prut je přímý
- tlaková síla působí v ose prutu
- deformace jsou řádově menší než délka prutu (teorie malých deformací)
- statické účinky se vyšetřují na zdeformovaném prutu (teorie II.řádu)

$$F_{cr} = \pi^2 \cdot \frac{E \cdot I}{L_{cr}^2}$$

$$L_{cr} = \beta \cdot l$$

L_{cr} ...vzpěrná délka

β ...součinitel vzpěrné délky



MKP model vs. ruční výpočet

Materiál: Ocel

E = 200 Gpa

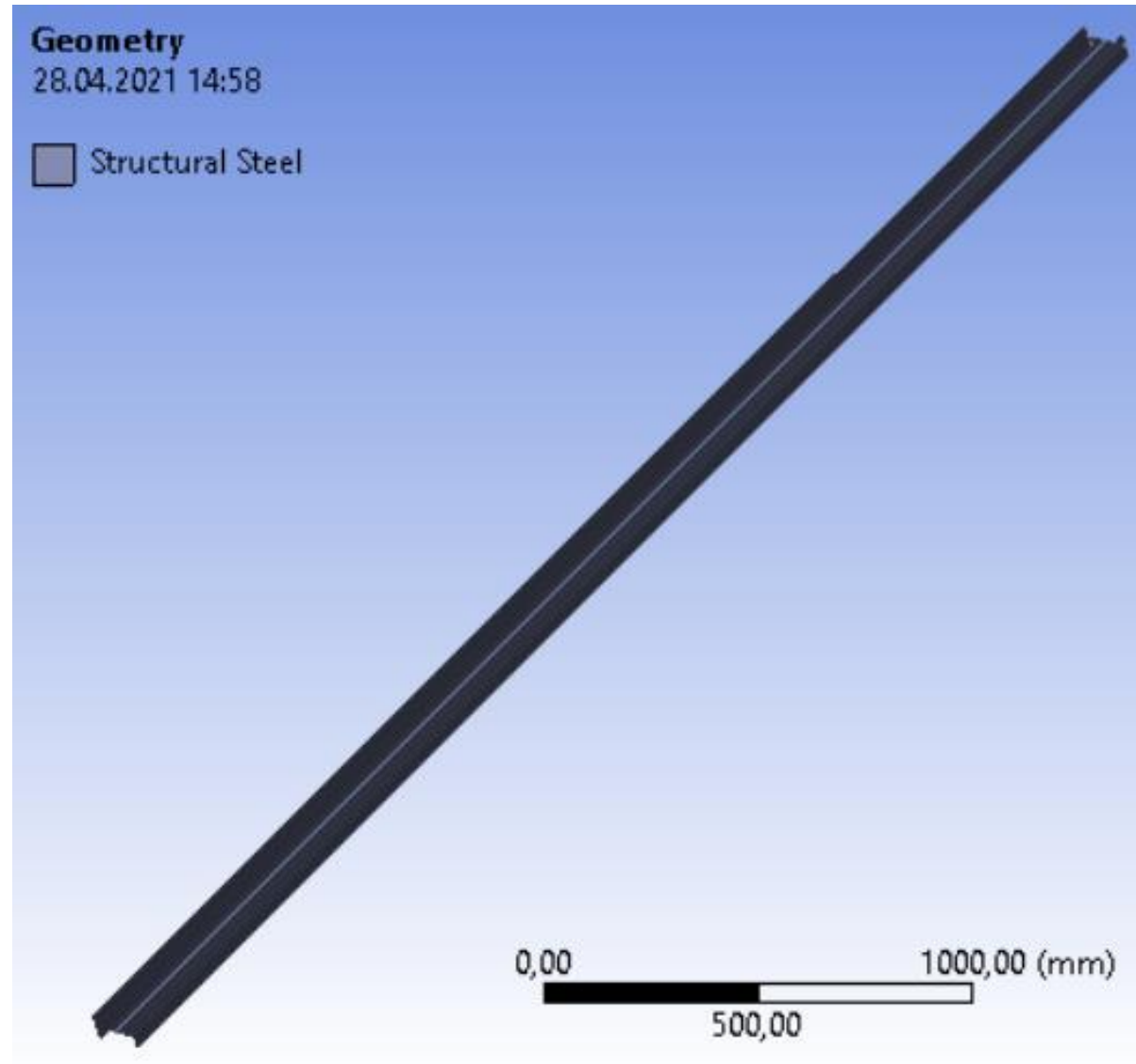
IPE 120

I_y = 3.18E-06 m⁴

I_z = 2.76E-07 m⁴

Délka prutu 4 m

$$F_{cr} = \pi^2 \cdot \frac{E \cdot I}{L_{cr}^2}$$



Ansys Workbench

The screenshot displays the Ansys Workbench interface for an "Unsaved Project - Workbench". The main workspace shows a "Project Schematic" with two analysis systems, A and B, connected by dependency lines. System A is "Static Structural" and System B is "Eigenvalue Buckling". Both systems share the same input components: Engineering Data, Geometry, Model, Setup, and Solution. The "Properties of Project Schematic" panel on the right shows a table with the following data:

	A	B
1	Property	Value
2	Notes	
3	Notes	
4	Solution Process	
5	Update Option	Run in Foreground

The left sidebar, titled "Toolbox", lists various analysis systems. Two items are circled in red: "Eigenvalue Buckling" and "Static Structural".

Ready | Job Monitor... | No DPS Connection | Show Progress | Show 0 Messages

Engineering data – vlastnosti materiálu

The screenshot displays the ANSYS Workbench interface with the following components:

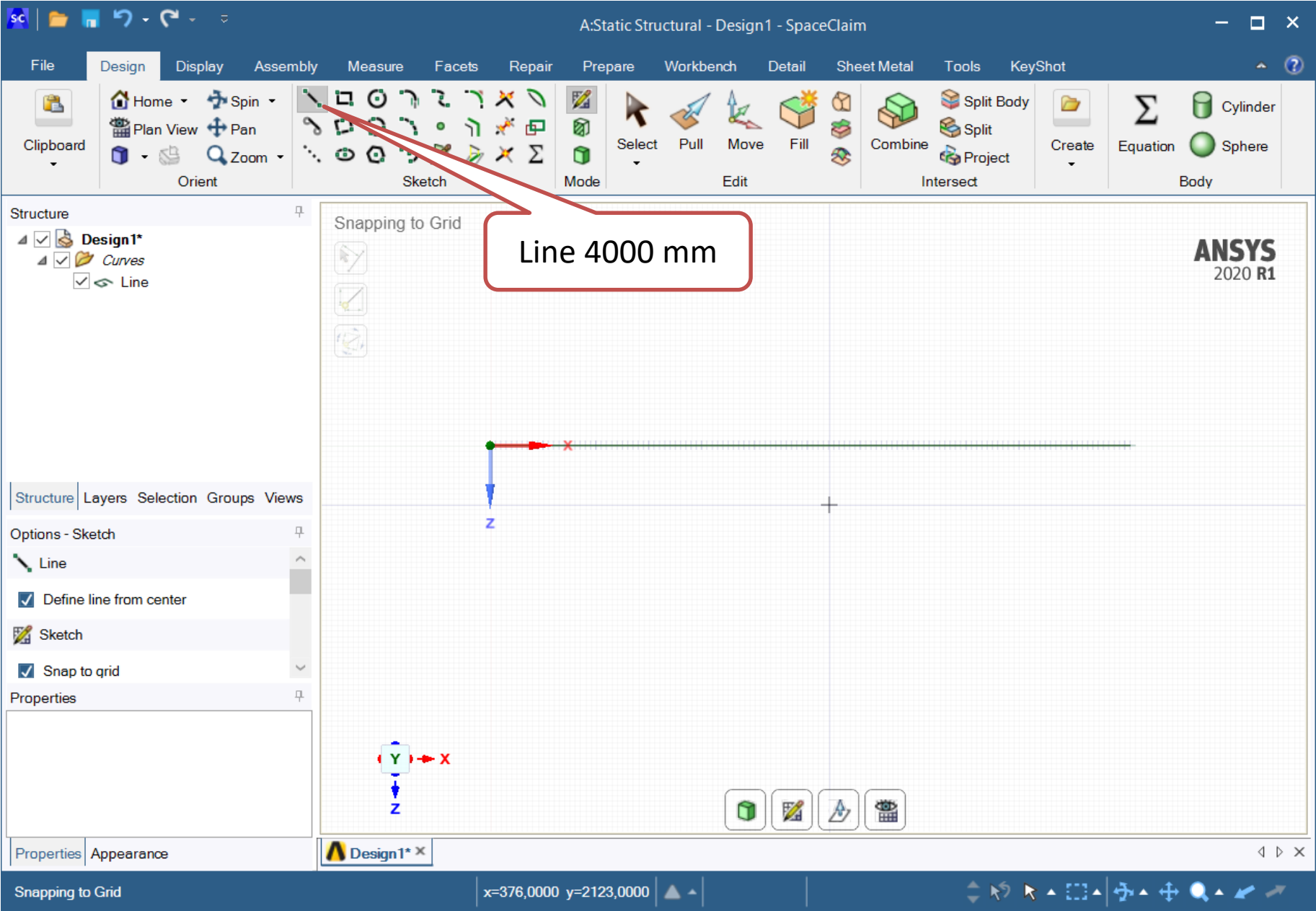
- Outline of Schematic A2, B2: Engineering Data:**

	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Structural Steel				Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				
- Table of Properties Row 8: Isotropic Elasticity:**

	A	B
1	Temperature (C)	Young's Modulus (Pa)
2		2E+11
*		
- Properties of Outline Row 3: Structural Steel:**

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m ⁻³		
4	Isotropic Secant Coefficient of Thermal Expansion				
5	Coefficient of Thermal Expansion	1,2E-05	C ⁻¹		
6	Isotropic Elasticity				
7	Derive from	Young's M...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	Strain-Life Parameters				
13	Display Curve Type	Strain-Life			
14	Strength Coefficient	9,2E+08	Pa		
15	Strength Exponent	-0,106			
16	Ductility Coefficient	0,213			
- Chart of Properties Row 8: Isotropic Elasticity:** A graph showing Young's Modulus (Pa) on the y-axis (ranging from 1 to 2.5) versus Temperature (C) on the x-axis (ranging from -1 to 1). A horizontal dashed line is drawn at 200 GPa (2E+11 Pa), with a red callout box pointing to it.

Geometrie – Space Claim



Geometrie – průřez z knihovny – IPE120

The screenshot displays the ANSYS 2020 R1 software interface. The main window title is "A:Static Structural - Design1 - SpaceClaim". The top ribbon includes tabs for File, Design, Display, Assembly, Measure, Facets, Repair, Prepare, Workbench, Detail, Sheet Metal, Tools, and KeyShot. The "Prepare" tab is active, showing various tools like Interference, Remove, and Detect.

The "Eurocode Library Manager" dialog box is open, showing a list of IPE profiles. The "Family" is set to "IPE". The "Selected Cross Sections" list contains "IPE 120". The "Import" button is highlighted with a red circle.

The "Profiles" dropdown menu is open, showing options like "New Profile Library", "More Profiles...", and "Standard Library". The "Standard Library" option is highlighted with a red circle.

The bottom status bar shows "Snapping to Grid" and coordinates "x=2160,000 y=2187,000".

Geometrie – aplikace průřezu na těžiště

vytvoření

Výběr linie

Výběr průřezu

ANSYS 2020 R1

Click to set a secondary selection to be used within other tools

Structure

- SYS*
- Beams
 - Beam (Eurocode_IPE 120)
- Beam Profiles
 - Eurocode_IPE 120

Options - Selection

Properties

Color	143; 175; 143
Layout	False
Beam	
Type	Beam
Orientation	0°
Profile Name	Eurocode_IPE 120

Length = 4 000 mm

1 Curve

Model – zobrazení objemu a kontrola geometrie

Multiple Systems - Mechanical [ANSYS Academic Research Mechanical and CFD]

File Home Model Display Selection Automation

Isometric Previous Rotate +Sx Rotate -Sx Pan Up Pan Down Look At Next Rotate +Sy Rotate -Sy Pan Left Pan Right Views Angle 10 Rotate +Sz Rotate -Sz Zoom In Zoom Out Orient Random Rescale Preferences Annotation Display Show Mesh Thick Shells and Beams Style Cross Section Display Style Vertex Edge Explode Viewports Show Display

Outline: Project* Model (A4, B4) Geometry SYS\Beam (Eurocode_IPE 120) Materials Structural Steel Cross Sections Eurocode_IPE 120 Coordinate Systems Mesh Static Structural (A5) Analysis Settings Solution (A6) Solution Information Eigenvalue Buckling (B5) Pre-Stress (Static Structural) Analysis Settings Solution (B6) Solution Information

Details of "Model (A4, B4)"

Lighting	
Ambient	0,1
Diffuse	0,6
Specular	1
Color	
Filter Options	
Control	Enabled

Model 28.04.2021 13:30

ANSYS 2020 R1 ACADEMIC

Zobrazit průřez

kontrola

0,00 500,00 1000,00 (mm)

Messages: Text Associat

Messages pane No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Mesh – default size – 40 mm

Multiple Systems - Mechanical [ANSYS Academic Research Mechanical and CFD]

File Home Mesh Display Selection Automation

Quick Launch

Isometric Previous Rotate +Sx Rotate -Sx Pan Up Pan Down Random Rescale Preferences Annotation Display Show Mesh Thick Shells and Beams Style Cross Section Display Style Vertex Edge Explode Viewports Show Display

Outline

Name Search Outline

Project*

- Model (A4, B4)
 - Geometry
 - SYS\Beam (Eurocode_IPE 120)
 - Materials
 - Structural Steel
 - Cross Sections
 - Eurocode_IPE 120
 - Coordinate Systems
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Solution (A6)
 - Solution Information
 - Eigenvalue Buckling (B5)
 - Pre-Stress (Static Structural)
 - Analysis Settings
 - Solution (B6)
 - Solution Information

Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	40, mm
Sizing	
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechani...
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None
Inflation	
Batch Connections	
Advanced	
Statistics	

40 mm

ANSYS 2020 R1 ACADEMIC

0,00 500,00 1000,00 (mm)

Messages

Text Associat

Ready No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Static Structural - Fixed Support

The screenshot displays the ANSYS Academic Research Mechanical and CFD software interface. The main window shows a 3D model of a beam with a fixed support at one end. The software title is "Multiple Systems - Mechanical [ANSYS Academic Research Mechanical and CFD]". The interface includes a top menu bar (File, Home, Environment, Display, Selection, Automation), a toolbar with various manipulation tools, and a central workspace showing the beam model. A scale bar indicates 0,00 to 1000,00 (mm) with a 500,00 mm mark. A coordinate system (X, Y, Z) is visible in the bottom right of the workspace.

The **Outline** panel on the left shows the project hierarchy:

- Project*
- Model (A4, B4)
 - Geometry
 - SYS\Beam (Eurocode_IPE 120)
 - Materials
 - Structural Steel
 - Cross Sections
 - Eurocode_IPE 120
 - Coordinate Systems
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Solution (A6)
 - Solution Information
- Eigenvalue Buckling (B5)
 - Pre-Stress (Static Structural)
 - Analysis Settings
 - Solution (B6)
 - Solution Information

The **Details of "Fixed Support"** panel shows the following information:

Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Fixed Support
Suppressed	No

A red callout box labeled "Fixed" points to the "Fixed Support" entry in the Outline panel.

The **Graph** and **Tabular Data** panels are visible at the bottom of the interface.

ANSYS 2020 R1 ACADEMIC

Static Structural – Force – $x = -1 \text{ kN}$

ANSYS 2020 R1 ACADEMIC

A: Static Structural
 Force
 Time: 1, s
 28.04.2021 13:37

Force: 1000, N
 Components: -1000,;0,;0, N

Force Components
 $x = -1 \text{ kN}$

0,00 500,00 1000,00 (mm)

Steps	Time [s]	X [N]	Y [N]	Z [N]
1	1	= 0,	= 0,	= 0,
2	1	-1000,	0,	0,
*				

Nastavení Total deformation

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a beam structure with a total deformation plot overlaid. The plot shows a linear displacement of approximately 1000 mm. The interface includes a menu bar (File, Home, Solution, Display, Selection, Automation), a toolbar with various analysis tools, and a central workspace. The 'Details of "Total Deformation"' panel is open, showing the following settings:

Details of "Total Deformation"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
Mode	1,
Identifier	
Suppressed	No
Results	
<input type="checkbox"/> Load Multiplier	
<input type="checkbox"/> Minimum	
<input type="checkbox"/> Maximum	
<input type="checkbox"/> Average	
Minimum Occurs On	
Maximum Occurs On	

The 'Outline' panel on the left shows the project hierarchy. Two 'Total Deformation' entries are circled in red:

- Under 'Solution (A6)', 'Total Deformation' is circled.
- Under 'Solution (B6)', 'Total Deformation' is circled.

The 'Graph' panel at the bottom shows a plot of the total deformation. The plot title is 'B: Eigenvalue Buckling' and it includes the text 'Total Deformation' and '28.04.2021 13:39'. The plot shows a linear displacement of approximately 1000 mm. A scale bar indicates 0,00 to 1000,00 (mm) with a midpoint at 500,00. The ANSYS 2020 R1 Academic logo is visible in the top right corner of the plot area.

Nastavení počtu vlastních tvarů vybočení = 6

The screenshot displays the ANSYS Mechanical software interface. The main window shows a 3D model of a beam structure. The software title is "Multiple Systems - Mechanical [ANSYS Academic Research Mechanical and CFD]". The interface includes a ribbon menu with tabs for File, Home, Environment, Display, Selection, and Automation. The "Automation" tab is active, showing various toolbars and a "Quick Launch" search bar.

The "Outline" pane on the left shows the project hierarchy:

- Project*
- Model (A4, B4)
 - Geometry
 - SYS\Beam (Eurocode_IPE 120)
 - Materials
 - Structural Steel
 - Cross Sections
 - Eurocode_IPE 120
 - Coordinate Systems
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Force
 - Solution (A6)
 - Solution Information
 - Total Deformation
 - Eigenvalue Buckling (B5)
 - Pre-Stress (Static Structural)
 - Analysis Settings
 - Solution (B6)

The "Details of 'Analysis Settings'" pane is open, showing the following settings:

Options	
Max Modes to Find	6
Solver Controls	
Solver Type	Progra...
Include Negative Load Multiplier	Progra...
Output Controls	
Analysis Data Management	

The "Max Modes to Find" value is circled in red. The main view shows a 3D model of a beam with a coordinate system (X, Y, Z) and a scale bar indicating 0,00 to 1000,00 (mm).

The "Graph" pane at the bottom left shows a single data point at 1, and the "Tabular Data" pane is empty.

The status bar at the bottom indicates "2 Messages", "No Selection", and units: "Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius".

Zobrazení všech pěti vlastních tvarů

B: Eigenvalue Buckling
Total Deformation
Type: Total Deformation
Load Multiplier (Linear): 8,5327
Unit: mm
28.04.2021 14:45

1 Max
0,88889
0,77778
0,66667
0,55556
0,44444
0,33333
0,22222
0,11111
0 Min

0,00 1000,00 (mm)
500,00

Graph
Animation | 20 Frames

Mode	Load Multiplier
1	8,5327
2	76,732
3	97,926
4	212,89
5	416,43

Create Mode Shape Results

Zobrazení všech šesti vlastních tvarů

