

Tvorba geometrie, základní statická úloha

Ing. Petr Lehner

Co se dozvíme a naučíme?

Rozdíly v ručním výpočtu a MKP analýze u staticky určité úlohy.

K čemu jsou programy Ansys Workbench, Space Claim, Ansys Mechanical.

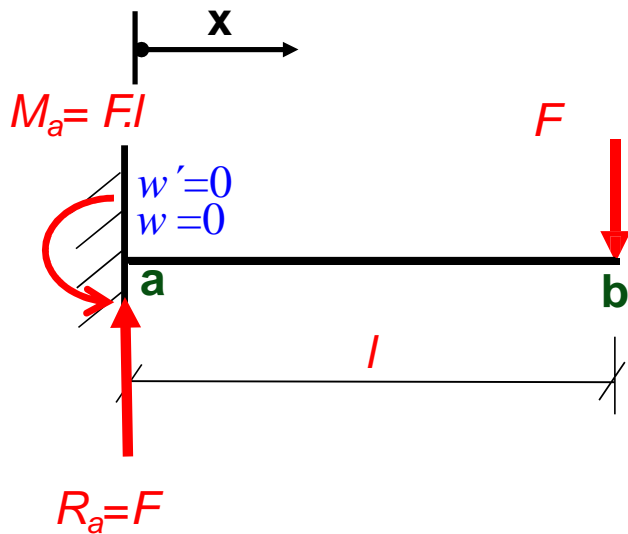
Jak připravit objemový prvek ve Space Claim.

Postup základní statické analýzy v Ansys Mechanical.

Na co si dát pozor při tvorbě sítě a při vyhodnocení výsledků.

Statically determinate structure - cantilever

$$F = 10 \text{ kN}, l = 1 \text{ m}, b = 0,05 \text{ m}, h = 0,1 \text{ m}, E = 200 \text{ GPa}$$



$$M(x) = -M_a + R_a \cdot x$$

$$M(x) = -F \cdot l + F \cdot x$$

$$EI \cdot w'' = -M(x)$$

Základní dif. rovnice 2.řádu

a její dvojí integrace:

$$EI w'' = +F \cdot l - F \cdot x$$

$$EI w' = +F \cdot l \cdot x - F \cdot x^2/2 + C_1$$

$$EI w = F \cdot l \cdot x^2/2 - F \cdot x^3/6 + C_1 x + C_2$$



$$M_a = ?$$

$$\sigma = \frac{M \cdot h}{2 \cdot I} = ?$$

Okrajové podmínky: 1. $w'(x=0)=0: C_1=0$

2. $w(x=0)=0: C_2=0$

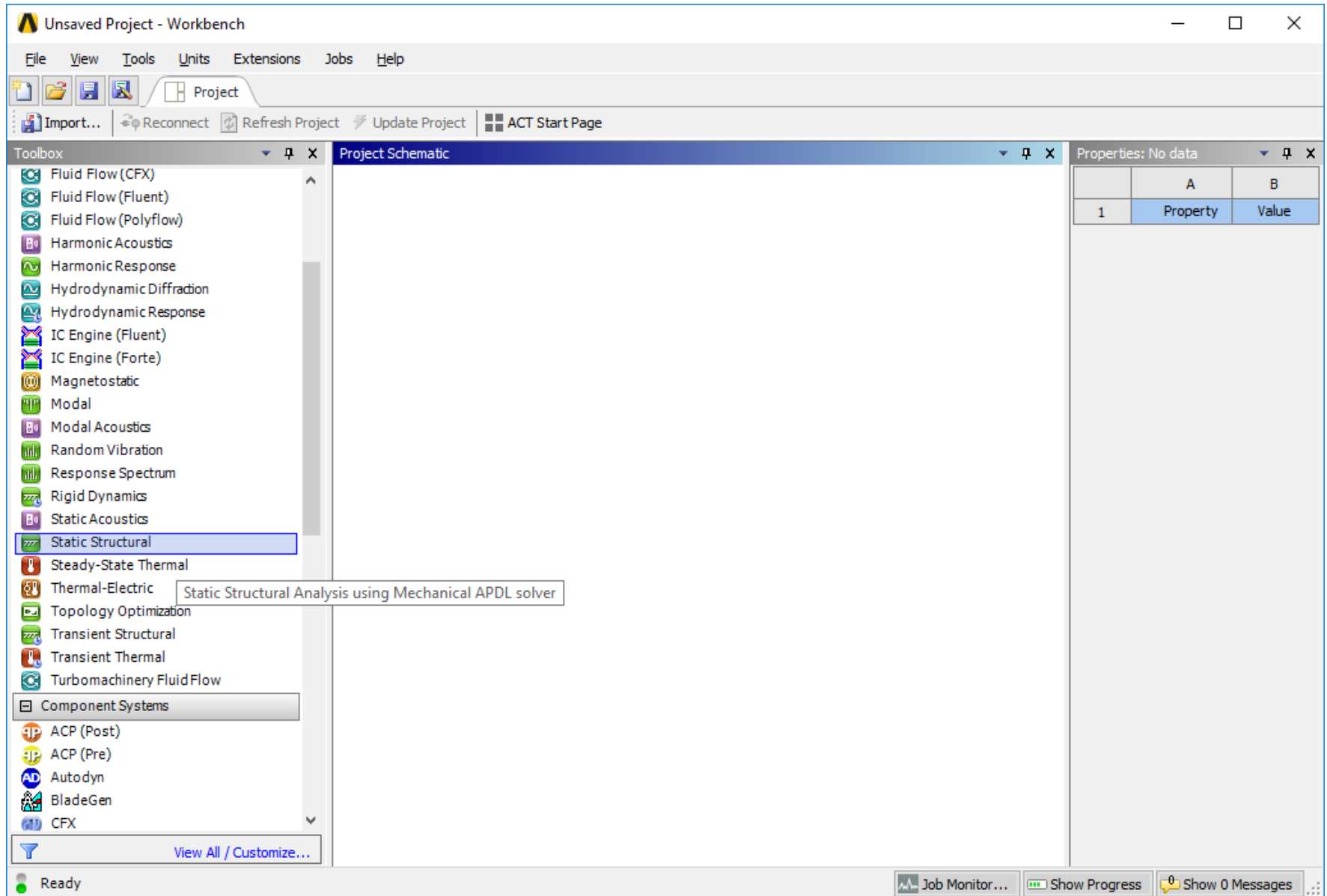
Rovnice pootočení: $w' = 1/EI (F \cdot l \cdot x - F \cdot x^2/2)$

Rovnice ohybové čáry: $w = 1/EI (F \cdot l \cdot x^2/2 - F \cdot x^3/6)$

Úhel pootočení v b: $\varphi_b = w'(x=l) = 1/EI \cdot (F \cdot l \cdot l - F \cdot l^2/2) = Fl^2/(2EI)$

Průhyb v b: $w_b = w(x=l) = 1/EI (F \cdot l \cdot l^2/2 - F \cdot l^3/6) = Fl^3/(3EI) = ?$

Ansys Workbench – Static Structural



Ansys Workbench – Static Structural

The screenshot displays the Ansys Workbench interface for a new project titled "Unsaved Project - Workbench". The main workspace shows the "Project Schematic" for a "Static Structural" analysis. The schematic is organized into a tree view under the name "A", with the following components:

- 1 Static Structural
- 2 Engineering Data ✓
- 3 Geometry ?
- 4 Model ?
- 5 Setup ?
- 6 Solution ?
- 7 Results ?

The text "Example 01" is visible below the schematic tree. On the left, the "Toolbox" lists various analysis types, with "Static Structural" highlighted. On the right, the "Properties of Project Schematic" panel shows a table with the following data:

	A	B
1	Property	Value
2	Notes	
3	Notes	
4	Solution Process	
5	Update Option	R...

The bottom status bar indicates the software is "Ready" and shows "Job Monitor...", "Show Progress", and "Show 0 Messages" buttons.

Engineering Data – materiálové vlastnosti

The screenshot displays the ANSYS Workbench Engineering Data environment. The main window is titled "Unsaved Project - Workbench" and shows the "Outline of Schematic A2: Engineering Data" and "Properties of Outline Row 3: Structural Steel".

Outline of Schematic A2: 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				

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				
6	Isotropic Elasticity				
12	Strain-Life Parameters				
20	S-N Curve	Tabular			
24	Tensile Yield Strength	2,5E+08	Pa		
25	Compressive Yield Strength	2,5E+08	Pa		
26	Tensile Ultimate Strength	4,6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

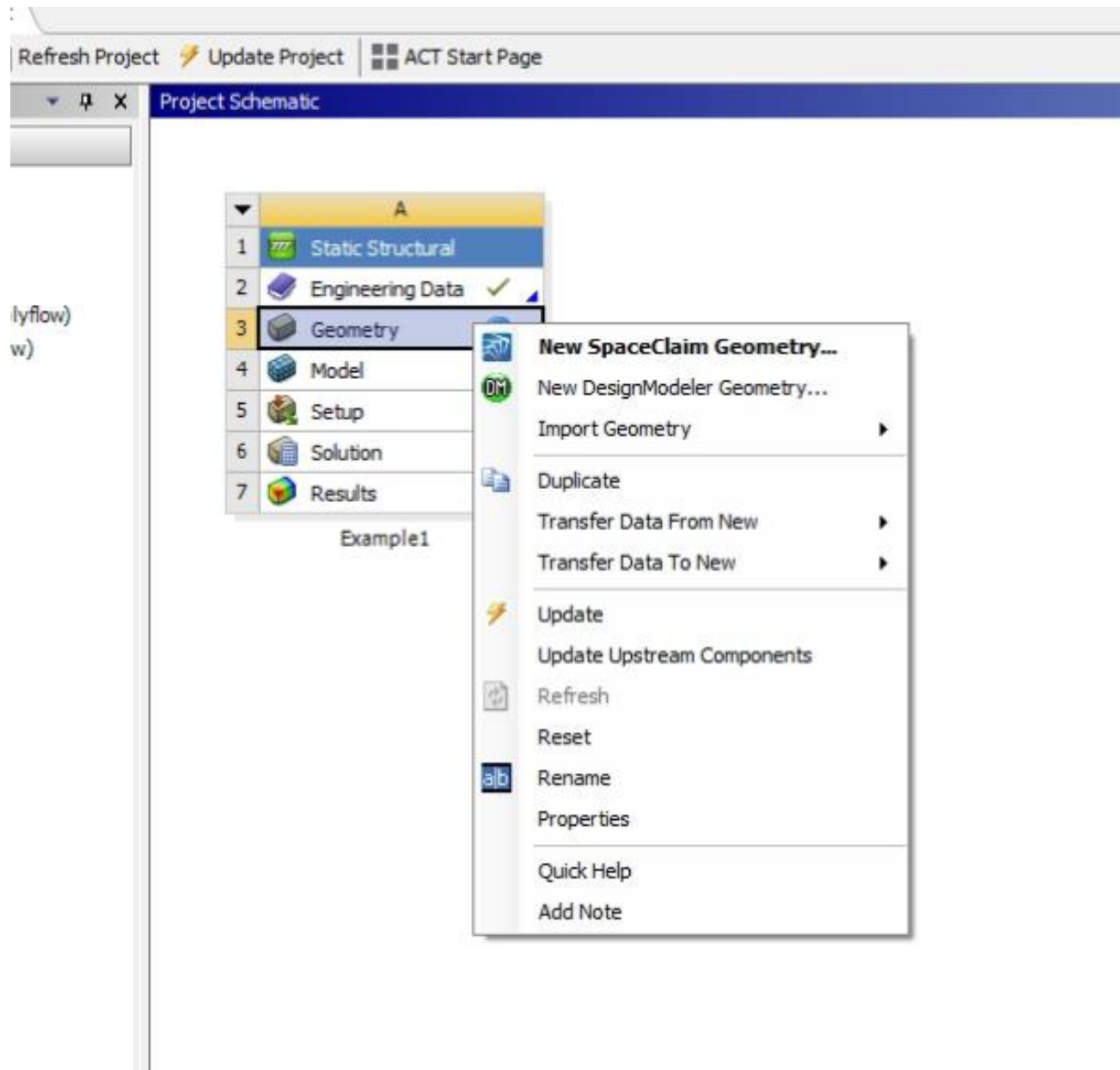
Table of Properties Row 2: Structural Steel Field Variables

	A	B	C	D	E
1	Variable Name	Unit	Default Data	Lower Limit	Upper Limit
2	Temperature	C	22	Program Controlled	Program Controlled
3	Mean Stress	Pa	0	Program Controlled	Program Controlled

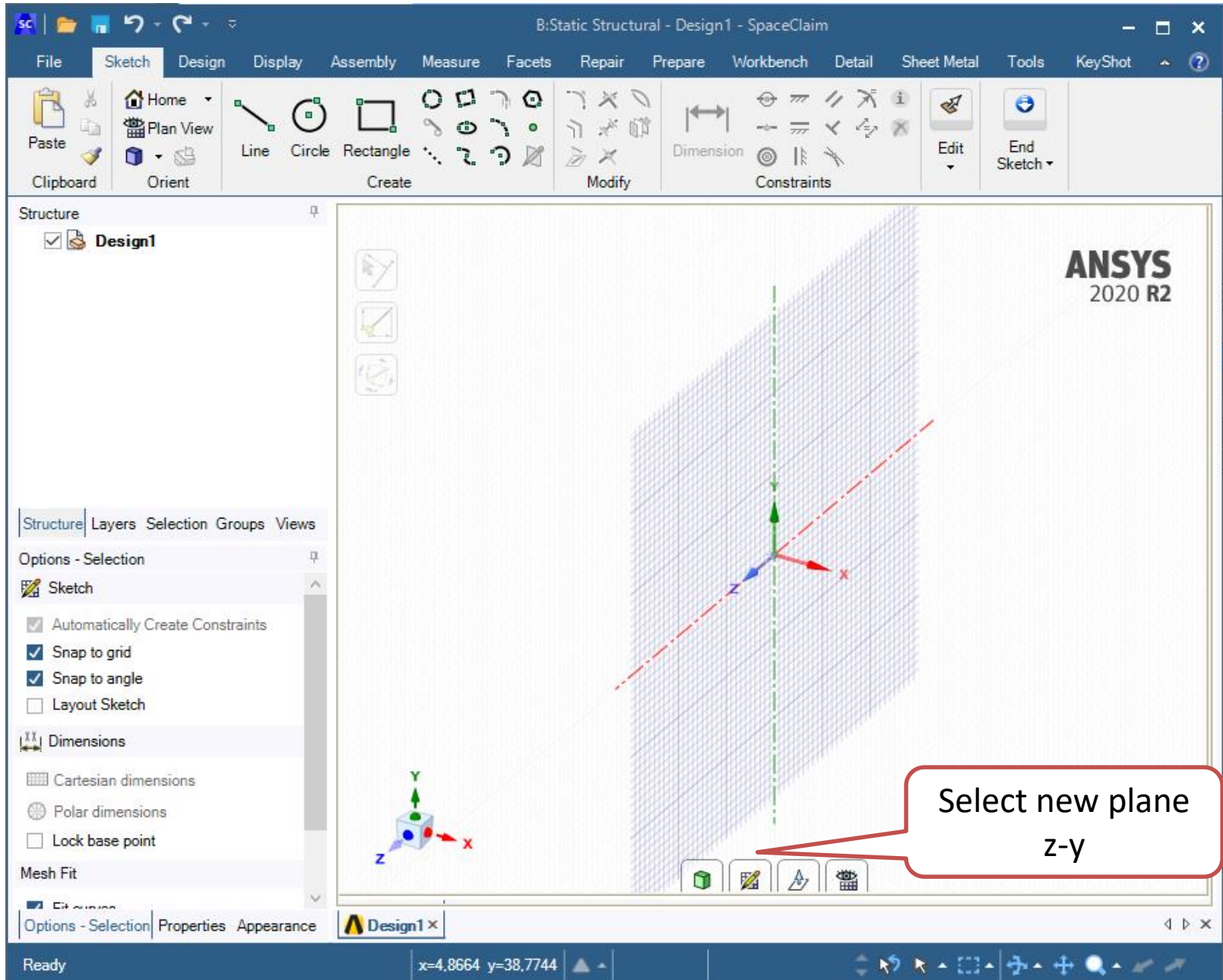
Chart: No data

A red callout box with a white background and a red border contains the text: "Nastavit parametry dle úlohy". A red arrow points from this box to the "Tensile Yield Strength" property in the "Properties of Outline Row 3: Structural Steel" table.

Geometry – import nebo nový model

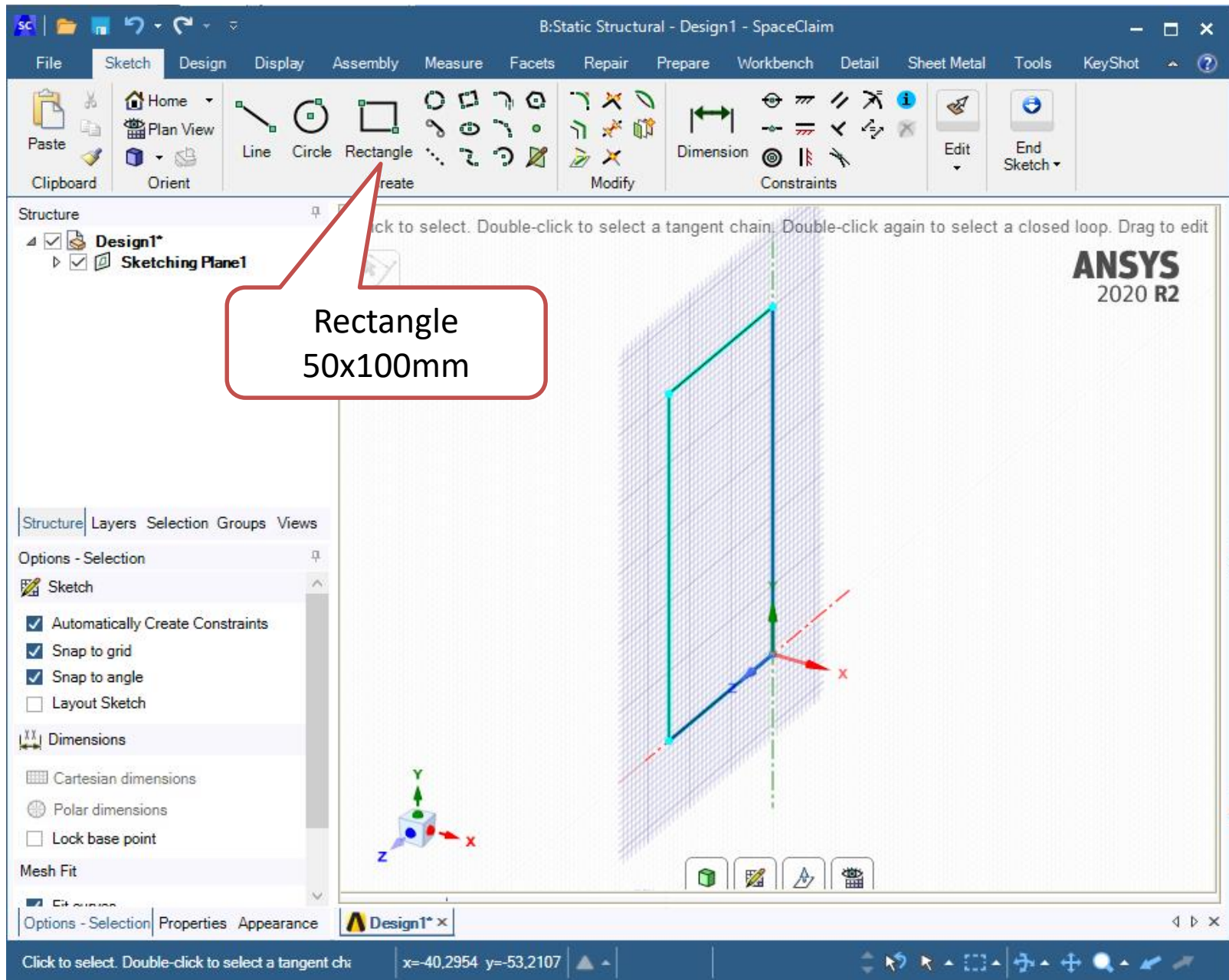


Geometry – SpaceClaim

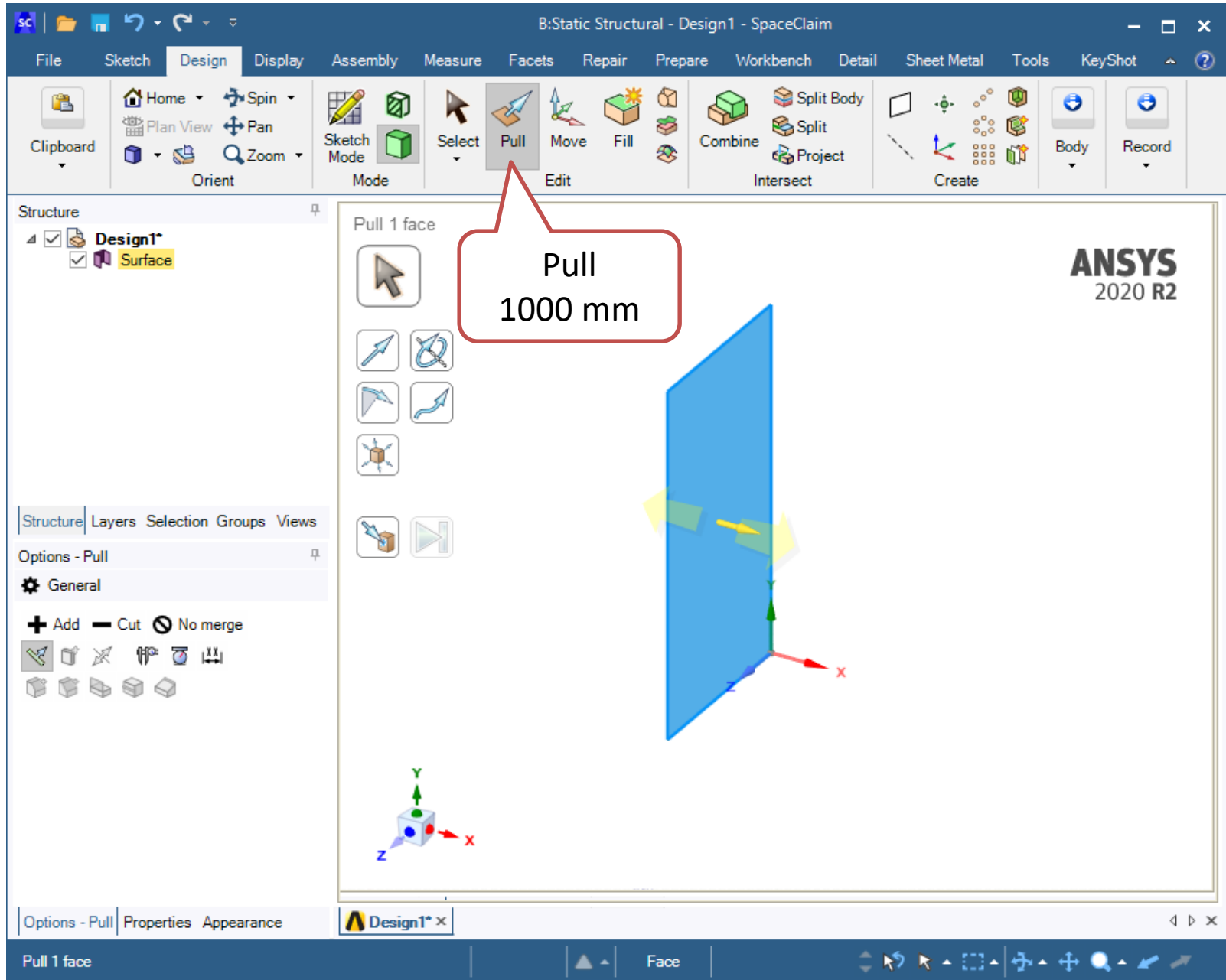


Select new plane
z-y

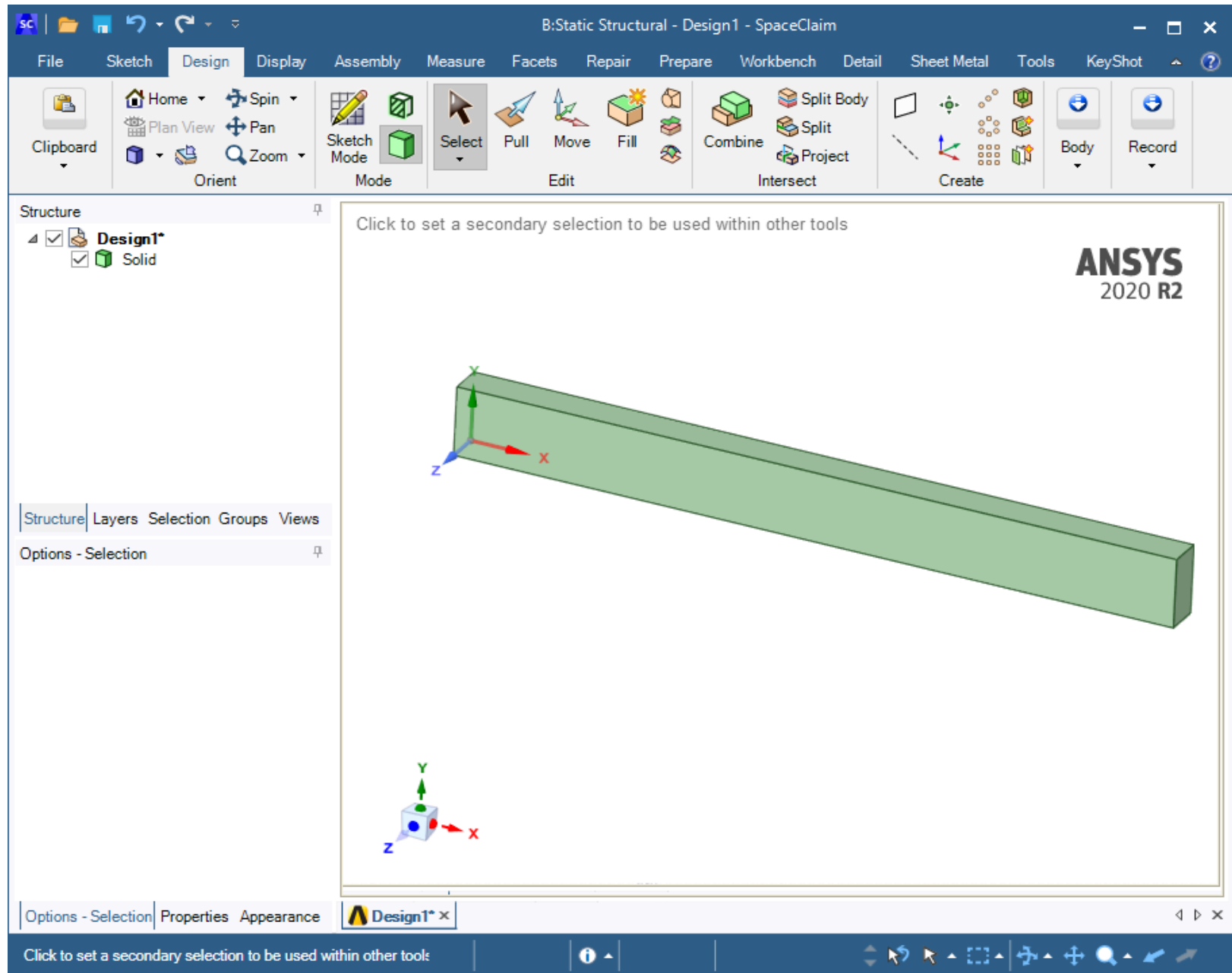
Geometry – SpaceClaim



Geometry – SpaceClaim



Geometry – SpaceClaim



Model

The screenshot displays the ANSYS 2020 R2 Academic software interface. The main window shows a 3D model of a beam, rendered in a dark gray color, positioned diagonally. The beam is supported at one end by a fixed constraint, indicated by a green arrow pointing upwards and a red arrow pointing to the right. A scale bar below the beam indicates a length of 200,00 mm, with a midpoint at 100,00 mm. The coordinate system (X, Y, Z) is visible at the bottom right of the model.

The software interface includes a top menu bar with options like File, Home, Model, Display, Selection, and Automation. The top toolbar contains various tools for navigation and display, such as Rotate, Pan, Zoom, and Show Vertices. The left sidebar shows the Outline tree, which lists the project components: Project*, Model (B4), Geometry, Materials, Coordinate Systems, Mesh, Static Structural (B5), Analysis Settings, Solution (B6), and Solution Information. The Details panel on the right shows the properties for the selected "Model (B4)", including Lighting (Ambient: 0,1, Diffuse: 0,6, Specular: 1, Color) and Filter Options (Control: Enabled).

The bottom status bar shows the current state: Ready, Messages pane, No Selection, Metric (mm, kg, N, s, mV, mA), Degrees, rad/s, Celsius.

Model – Mech – Generate Mesh

The screenshot displays the ANSYS 2020 R2 Academic interface for a static structural analysis. The main window shows a 3D model of a rectangular bar with a mesh applied. The software title is "B: Static Structural - Mechanical [ANSYS Academic Teaching Mechanical and CFD]".

The **Outline** tree on the left shows the project structure:

- Project*
- Model (B4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
 - Static Structural (B5)
 - Analysis Settings
 - Solution (B6)
 - Solution Information

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

Display	
Display Style	Use Geometry...
Defaults	
Physics Preference	Mechanical
Element Order	Program Cont...
<input type="checkbox"/> Element Size	10, mm
Sizing	
Quality	
Inflation	
Advanced	
Statistics	

Two red callout boxes highlight the "Generate mesh" button in the Outline tree and the "10 mm" element size setting in the Details panel.

The 3D view shows a rectangular bar with a mesh. A scale bar at the bottom indicates 0,00 to 200,00 (mm) with a midpoint at 100,00. A coordinate system (X, Y, Z) is visible at the bottom right.

The **Messages** pane at the bottom shows a table with columns for "Text" and "Association".

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

Static Structural – Insert – Fixed Support

The screenshot displays the ANSYS Workbench interface for a Static Structural analysis. The main window shows a 3D model of a beam with a mesh, where a Fixed Support is applied to one end. A red callout box labeled "Fixed" points to the 'Fixed' button in the toolbar. The software title is "B: Static Structural - Mechanical [ANSYS Academic Teaching Mechanical and CFD]".

Outline:

- Project*
- Model (B4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
- Static Structural (B5)
 - Analysis Settings
 - Fixed Support
- Solution (B6)
 - Solution Information

Details of "Fixed Support":

Scope	
Scoping Method	Geometry Selec...
Geometry	1 Face
Definition	
Type	Fixed Support
Suppressed	No

3D Model View:

- Coordinate System: X, Y, Z
- Scale Bar: 0,00 to 200,00 (mm), with 100,00 in the middle.
- Graph Panel: Shows a vertical line with the value "1".

ANSYS 2020 R2 ACADEMIC

Static Structural – Insert – Force

Force

Define by:
Components,
y = -10000 N

B: Static Structural
Force
Time: 1, s
22.03.2021 21:36

Force: 10000 N
Components: 0,;-10000;0, N

Steps	Time [s]	X [N]	Y [N]	Z [N]
1	0,	= 0,	= 0,	= 0,
2	1,	0,	-10000	0,
*				

Solution – Deformation – Total

The screenshot displays the ANSYS 2020 R2 Academic software interface. The main window shows a 3D model of a beam with a total deformation result. The beam is oriented horizontally, and the deformation is shown as a slight downward curve. A scale bar below the beam indicates a length of 200.00 mm, with a midpoint at 100.00 mm. The coordinate system (X, Y, Z) is visible in the bottom right corner.

The software interface includes a top menu bar with options like File, Home, Solution, Display, Selection, and Automation. The left sidebar shows the Outline tree with a hierarchy: Project* > Model (B4) > Static Structural (B5) > Solution (B6). The Solution (B6) folder is expanded, showing Solution Information and Total Deformation. The Details of "Solution (B6)" panel on the right shows the following information:

Adaptive Mesh Refinement	
Max Refinement Loops	1,
Refinement Depth	2,

Information	
Status	Solve ...
<input type="checkbox"/> MAPDL Elapsed Time	
MAPDL Memory Used	
MAPDL Result File Size	

Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

A red callout box on the left side of the image contains the following text:

Solution
>Insert
>Deformation
>Total

The bottom status bar shows "Ready" and "Messages pane No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius".

Solution – Stress – Normal Stress - X

Outline

- Project*
- Model (B4)
 - Geometry
 - SYS-1\Solid
 - Construction Geometry
 - Path
 - Surface
 - Materials
 - Coordinate Systems
 - Global Coordinate System
 - Coordinate System
 - Mesh
 - Static Structural (B5)
 - Analysis Settings
 - Fixed Support
 - Force
 - Solution (B6)
 - Solution Information
 - Total Deformation
 - Normal Stress

Details of "Normal Stress"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Normal Stress
Orientation	X Axis
By	Time
<input type="checkbox"/> Display Time	Last
Coordinate System	Global Coordinate S...
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
<input type="checkbox"/> Minimum	
<input type="checkbox"/> Maximum	
<input type="checkbox"/> Average	
Minimum Occurs On	
Maximum Occurs On	
Information	

B: Static Structural
Normal Stress
23.03.2021 12:42

ANSYS
2020 R2
ACADEMIC

0,00 400,00 (mm)
200,00

Graph Tabular Data

0, 1,

Messages Graph

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

Solve

Solution
>Insert
>Stress
>Normal

Solution – Deformation – Total

B: Static Structural
 Total Deformation
 Type: Total Deformation
 Unit: mm
 Time: 1
 23.03.2021 11:25

4.024 Max
 3,5769
 3,1297
 2,6826
 2,2355
 1,7884
 1,3413
 0,89421
 0,44711
0 Min

cca 4 mm

Total Deformation

Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1	0	4,024	1,5171

Solution – Stress – Normal

B: Static Structural
Normal Stress
Type: Normal Stress(X)
Unit: MPa
Global Coordinate Sys
Time: 1
23.03.2021 12

182 Mpa ???

182,7 Max
142,1
101,5
60,902
20,301
-20,301
-60,902
-101,5
-142,1
-182,7 Min

Property	Value
Scoping Method	Geometry Selection
Geometry	All Bodies
Type	Normal Stress
Orientation	X Axis
By	Time
Display Time	Last
Coordinate System	Global Coordinate S...
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
Minimum	-182,7 MPa
Maximum	182,7 MPa
Average	1,6026e-008 MPa
Minimum Occurs On	SYS-1\Solid
Maximum Occurs On	SYS-1\Solid

Graph: Animation 20 Frames [MPa] [s]

Time [s]	Minimum [MPa]	Maximum [MPa]
1	-182,7	182,7

Model – Construction geometry - Path

The screenshot displays the ANSYS 2020 R2 Academic software interface. The main window shows a 3D model of a rectangular plate with a mesh. A path is defined along the top edge of the plate, starting at point 1 and ending at point 2. The path is highlighted in blue. The software interface includes a menu bar with options like File, Home, Construction Geometry, Display, Selection, and Automation. A toolbar contains various icons for creating and editing geometry. The Outline pane on the left shows the project hierarchy, including Model (B4), Construction Geometry, and Path. The Details pane on the right shows the properties of the selected path, including its definition, start and end coordinates, and location.

Model
>Insert
>Construction geometry
>Path

Start) XYZ
0;100;25 mm
End) XYZ
1000;100;25 mm

Definition	
Path Type	Two Points
Path Coordinate System	Global Co...
Number of Sampling Points	47,
Suppressed	No
Start	
Coordinate System	Global Co...
Start X Coordinate	0, mm
Start Y Coordinate	100, mm
Start Z Coordinate	25, mm
Location	Click to Ch...
End	
Coordinate System	Global Co...
End X Coordinate	1000, mm
End Y Coordinate	100, mm
End Z Coordinate	25, mm
Location	Click to Ch...

Messages

Text Association

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

Solution – Stress – Normal - Path

Scope > Path

Normal Stress

Solve

148 Mpa ???

DŮVODY:
- okrajové podmínky
- aproximace hodnot v síti

Details of "Normal Stress 2"	
Scope	
Scoping Method	Path
Path	Path
Geometry	All Bodies
Definition	
Type	Normal Stress
Orientation	X Axis
By	Time
<input type="checkbox"/> Display Time	Last
Coordinate System	Global Coordinate S...
Calculate Time History	Yes
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
<input type="checkbox"/> Minimum	0,81481 MPa
<input type="checkbox"/> Maximum	148,89 MPa
<input type="checkbox"/> Average	60,568 MPa
Minimum Occurs On	SYS-1\Solid
Maximum Occurs On	SYS-1\Solid
Graph Controls	
X-Axis	S
Information	

B: Static Structural
Normal Stress 2
Type: Normal Stress(X)
Unit: MPa
Global Coordinate S...
Time: 1
23.03.2021 12:48

148,89 Max
132,44
115,98
99,53
83,077
66,625
50,172
33,72
17,267
0,81481 Min

0,00 70,00 (mm)

Graph
Animation
[MPa] 148,89 0 125 250

Solution – Stress – Změna velikosti sítě 25

119 Mpa

119,06 Max
 105,91
 92,759
 79,608
 66,457
 53,306
 40,156
 27,005
 13,854
0,70306 Min

Details of "Normal Stress 2"

Scope	
Scoping Method	Path
Path	Path
Geometry	All Bodies
Definition	
Type	Normal Stress
Orientation	X Axis
By	Time
<input type="checkbox"/> Display Time	Last
Coordinate System	Global Coordinate S...
Calculate Time History	Yes
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
<input type="checkbox"/> Minimum	0,70306 MPa
<input type="checkbox"/> Maximum	119,06 MPa
<input type="checkbox"/> Average	60,071 MPa
Minimum Occurs On	SYS-1\Solid
Maximum Occurs On	SYS-1\Solid
Graph Controls	
X-Axis	S
Information	

Graph

Animation | 20 Frames

Graph: [MPa] vs [mm]

Length [mm]	Value [MPa]
1	0
2	20,833
3	41,667
4	62,5
5	83,333
6	104,17
7	125
8	145,83
9	166,67

ZÁVĚRY

- ruční výpočet je důležitý !
- MKP výpočet je vysoce citlivý na velikost sítě