

Šroubovaný spoj konzole na sloup

Ing. Petr Lehner

Co se dozvíme a naučíme?

Analýza tuhosti šroubového spoje na sloup.

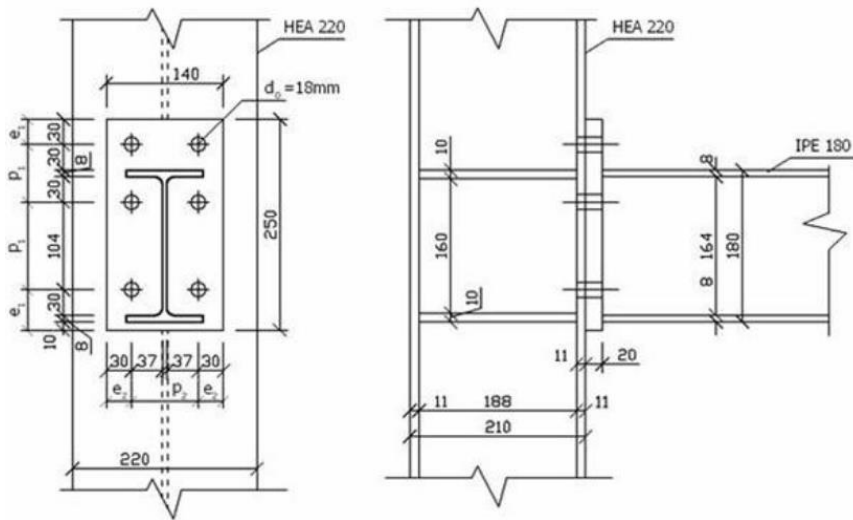
Nastavení sítě konečných prvků na válcovaném profilu.

Nepřímé kontakty pomocí Joints.

Jaké jsou možnosti jonts z hlediska reálného chování.

Rozdíl mezi ideálním svařovaným a šroubovaným spojem.

Šroubovaný spoj konzole na sloup



Důvod

- chování detailů u složitějších modelů
- stanovení tuhosti
- první fáze přípravy ukotvení

Materiál

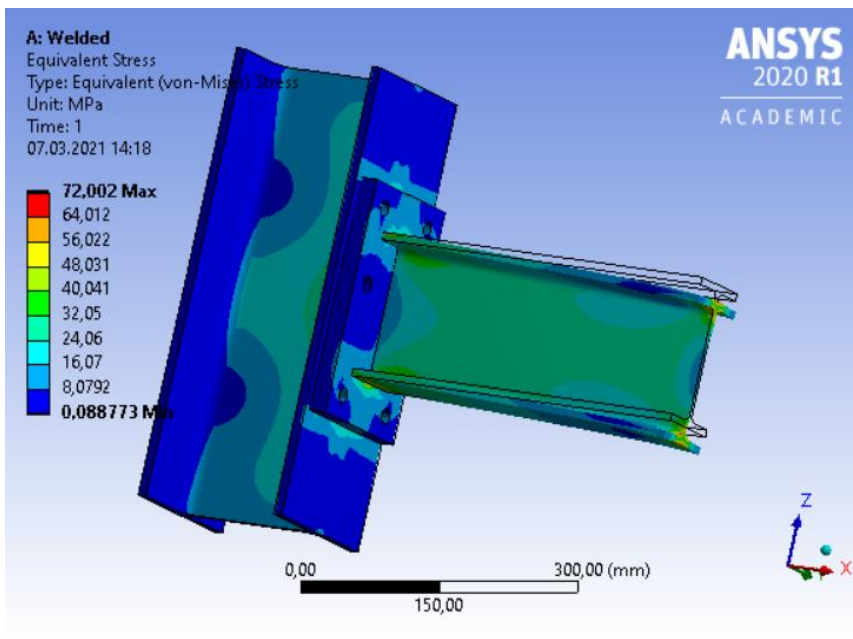
- ocel (default)

Konstrukce

- import z Autocad

2. Příklady

- svařený
- šroubovaný



Ansys Workbench - Static Structural

The screenshot displays the Ansys Workbench interface. The main window is titled "Unsaved Project - Workbench". The menu bar includes File, View, Tools, Units, Extensions, Jobs, and Help. The toolbar contains icons for Import..., Reconnect, Refresh Project, Update Project, and ACT Start Page. The Project Schematic area shows a tree view with the following items:

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

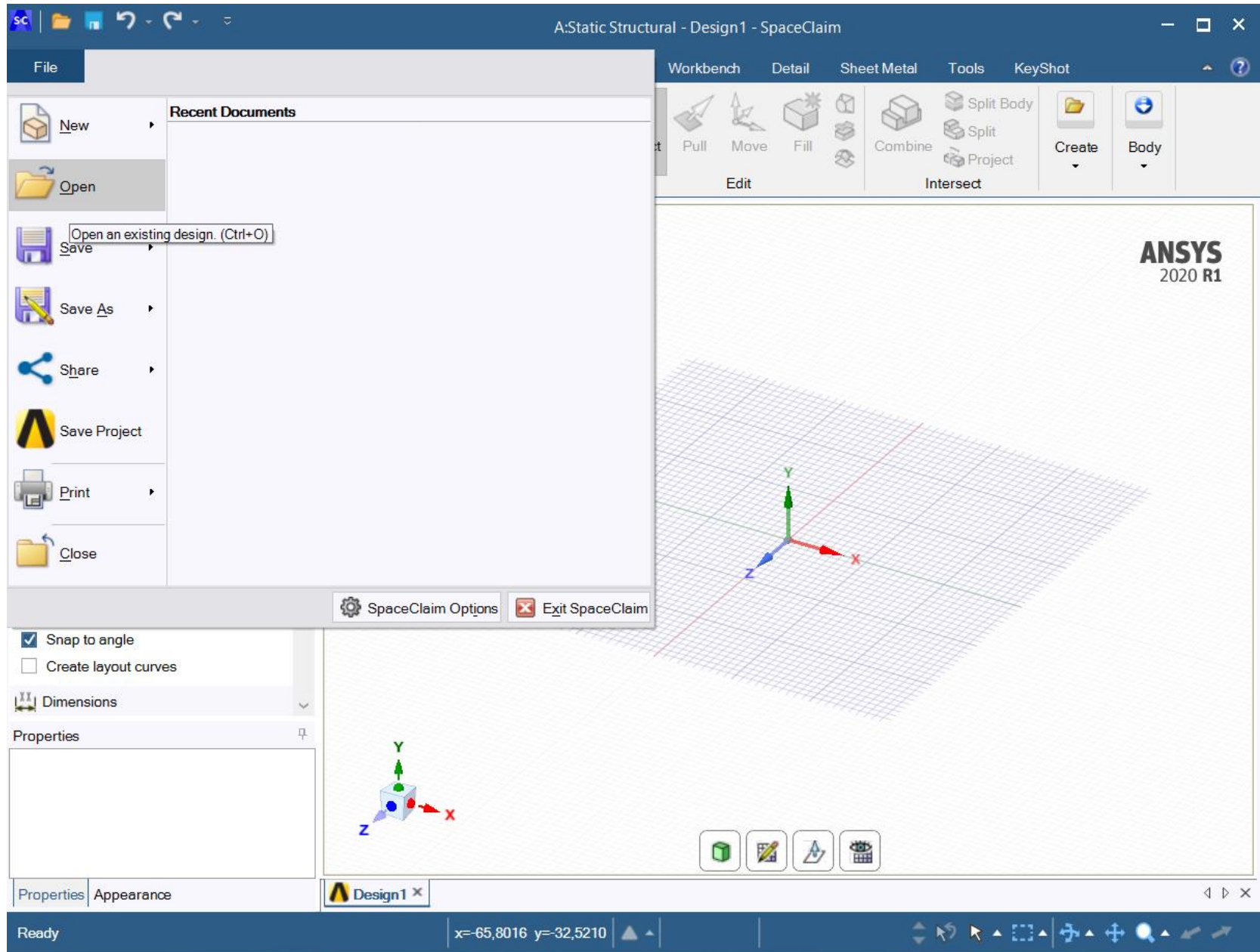
The "Geometry" item is circled in red. A callout box points to it with the text "dvojklik".

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	R...

The status bar at the bottom indicates "Ready" and includes buttons for Job Monitor..., Show Progress, and Show 0 Messages.

Space Claim – import geometrie z Autocad



Space Claim – kontrola – smazat plochu

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

ANSYS
2020 R1

Structure

- Design2*
 - Solid1
 - Solid2
 - Solid3
 - geometrie_06

Properties

Appearance

Clip face	<input type="checkbox"/>
Fill	Automatic
Include Curves	True
Outline	Automatic

Misc

1 Plane

Model – Mechanical – Geometry and Materials

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a mechanical assembly, likely a bracket or support structure, rendered in a perspective view. The model is composed of several parts, including a base plate, a vertical support, and a horizontal beam. The interface includes a top menu bar with options like File, Home, Model, Display, Selection, and Automation. Below the menu bar is a toolbar with various icons for modeling and analysis. The left side of the interface features an Outline pane showing the project hierarchy, including Model (A4), Static Structural (A5), and Solution (A6). The center of the interface displays the 3D model, with a coordinate system (X, Y, Z) and a scale bar indicating dimensions in millimeters (0,00 to 300,00 mm). The bottom of the interface shows a Messages pane and a status bar with information such as "Messages pane", "No Selection", and "Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius".

Model
07.03.2021 12:17

ANSYS
2020 R1
ACADEMIC

0,00 150,00 300,00 (mm)

Messages
Text Association

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

Connections – Contacts – Bonded ≈ svařeno

Context A : Static Structural - Mechanical [ANSYS Academic Research Mechanical and CFD]

File Home **Connections** Display Selection Automation Quick Launch

Duplicate Solve Analysis Named Selection Commands Images Coordinate System Comment Section Plane Connection Group Spring Beam Spot Weld Contact Contact Tool Contact Solution Information Joint Views

Remote Point Chart Annotation Connect Bearing Body Interaction

Outline Details of "Contact Region" Contact Body View

Name Project* Model (A4) Geometry SYS\Solid1 SYS\Solid2 SYS\Solid3 Materials Structural Steel Coordinate Systems Global Coordinate System Connections Contacts Contact Region Contact Region Mesh Static Structural (A5) Analysis Settings Solution (A6) Solution Inform

Details of "Contact Region"	
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	SYS\Solid1
Target Bodies	SYS\Solid3
Protected	No
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	2,0037 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Small Sliding	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled
Geometric Modification	
Contact Geometry Co...	None
Target Geometry Corr...	None

Contact Region
07.03.2021 13:11

- Contact Region (Contact Bodies)
- Contact Region (Target Bodies)

ANSYS 2020 R1 ACADEMIC

0,00 100,00 Z X Y

0,00 200,00 (mm) 100,00 Z X Y

0,00 70,00 Z X Y

Messages Text Associat

Make a copy of the selected object(s) and automatically insert it into the current Outline location. Messages pane No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Connections – Contacts – Bonded ≈ svařeno

The screenshot displays the ANSYS Workbench interface for a static structural analysis. The 'Connections' tab is active, and the 'Details of "Contact Region 2"' panel is open. The contact is defined as 'Bonded' between 'SYS\Solid2' (Contact Bodies) and 'SYS\Solid3' (Target Bodies). The 'Definition' section shows 'Type: Bonded', 'Scope Mode: Automatic', 'Behavior: Program Controlled', and 'Trim Contact: Program Controlled'. The 'Advanced' section shows 'Formulation: Program Controlled', 'Small Sliding: Program Controlled', and 'Detection Method: Program Controlled'. The 'Geometric Modification' section shows 'Contact Geometry Correction: None' and 'Target Geometry Correction: None'. The main view shows a 3D model of a mechanical assembly with a contact region highlighted in blue. A 'Target Body View' inset shows a close-up of the contact area. The 'Messages' pane at the bottom is empty.

Details of "Contact Region 2"	
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	SYS\Solid2
Target Bodies	SYS\Solid3
Protected	No
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	2,0037 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Small Sliding	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled
Geometric Modification	
Contact Geometry Correction	None
Target Geometry Correction	None

Mesh – Element Quality – Size = 10 mm

The screenshot displays the ANSYS 2020 R1 Academic interface for a Static Structural analysis. The main window shows a 3D model of a mechanical part with a mesh applied. The mesh is color-coded by element quality, with a scale from 0.095141 (red) to 0.99776 (blue). The mesh size is set to 10 mm. The software title bar indicates the analysis is a Static Structural - Mechanical simulation.

Details of "Mesh"

Category	Property	Value
Display	Display Style	Element Quality
	Element Size	10, mm
Defaults	Physics Preference	Mechanical
	Element Order	Program Controlled
Sizing	Element Size	10, mm
Quality	Element Quality	0.99776 Max 0.095141 Min
Statistics	Nodes	34676
	Elements	10004

Mesh Statistics

Quality	Value
Max	0.99776
Min	0.095141

ANSYS 2020 R1 ACADEMIC

07.03.2021 13:33

0,00 100,00 200,00 (mm)

Messages

Text

Assoc

Ready

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

Mesh – Insert – Method - Multizone

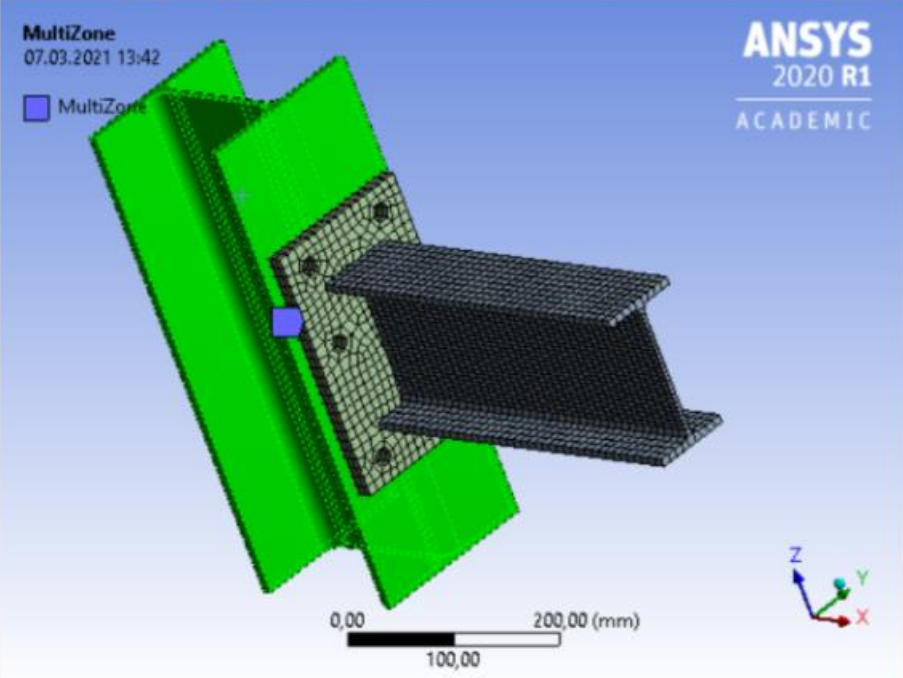
The screenshot displays the ANSYS Academic Research Mechanical and CFD software interface. The main window title is "A : Static Structural - Mechanical [ANSYS Academic Research Mechanical and CFD]". The ribbon at the top includes "File", "Home", "Mesh", "Display", "Selection", and "Automation". The "Mesh" tab is active, showing various meshing tools like "Isometric", "Look At", "Previous", "Next", "Rotate", "Pan", "Rescale", "Random", "Preferences Annotation", "Display", "Show Mesh", "Thick Shells and Beams Style", "Cross Section", "Display Style", "Vertex", "Edge", "Explode", "Viewports", and "Show Display".

The Outline tree on the left shows the project structure:

- Project*
- Model (A4)
 - Geometry
 - SYS\Solid1
 - SYS\Solid2
 - SYS\Solid3
 - Materials
 - Structural Steel
 - Coordinate Systems
 - Global Coordinate S
 - Connections
 - Contacts
 - Contact Region
 - Contact Region
 - Mesh
 - MultiZone
- Static Structural (A5)
 - Analysis Settings
 - Solution (A6)
 - Solution Inform

Details of "MultiZone" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Not Allowed
Element Order	Use Global Setting
Src/Trg Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
<input type="checkbox"/> Sweep Element Size	Default
Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	10, mm
Write ICEM CFD Files	No



Messages

Text

Associat

Insert Supports

Vypnout „Show Mesh“

The screenshot displays the ANSYS Mechanical interface for a static structural analysis. The software title is "A : Static Structural - Mechanical [ANSYS Academic Research Mechanical and CFD]". The toolbar shows various navigation and display options, including "Show Mesh" and "Thick Shells and Beams Style". The Outline tree on the left shows the project structure, including "Model (A4)", "Static Structural (A5)", and "Solution (A6)". The Details panel for the "Fixed Support" shows the following information:

Details of "Fixed Support"	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Fixed Support
Suppressed	No

The 3D model view shows a mechanical part with a fixed support applied to the top and bottom flanges. A coordinate system (X, Y, Z) and a scale bar (0,00 to 300,00 mm) are visible. The status bar at the bottom indicates "Ready" and "No Messages".

Fixed Support aplikovat na horní a dolní průřez

Insert Remote Force, Z = -10 000 N

The image shows the ANSYS Academic Research Mechanical and CFD interface. The main window displays a 3D model of a mechanical assembly with a red square indicating the location of a remote force. The force is defined as 10000 N with components (0, 0, -10000) N. The location is specified as (420, -1,2588e-016, 250) mm. The software version is ANSYS 2020 R1 Academic.

Details of "Remote Force"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Coordinate	420, mm
<input type="checkbox"/> Y Coordinate	-1,2588e-016 mm
<input type="checkbox"/> Z Coordinate	250, mm
Location	Click to Change

Definition	
Type	Remote Force
Define By	Components
<input type="checkbox"/> X Component	0, N (ramped)
<input type="checkbox"/> Y Component	0, N (ramped)
<input checked="" type="checkbox"/> Z Component	-10000 N (ramped)
Suppressed	No
Behavior	Deforma

Advanced	
----------	--

Graph

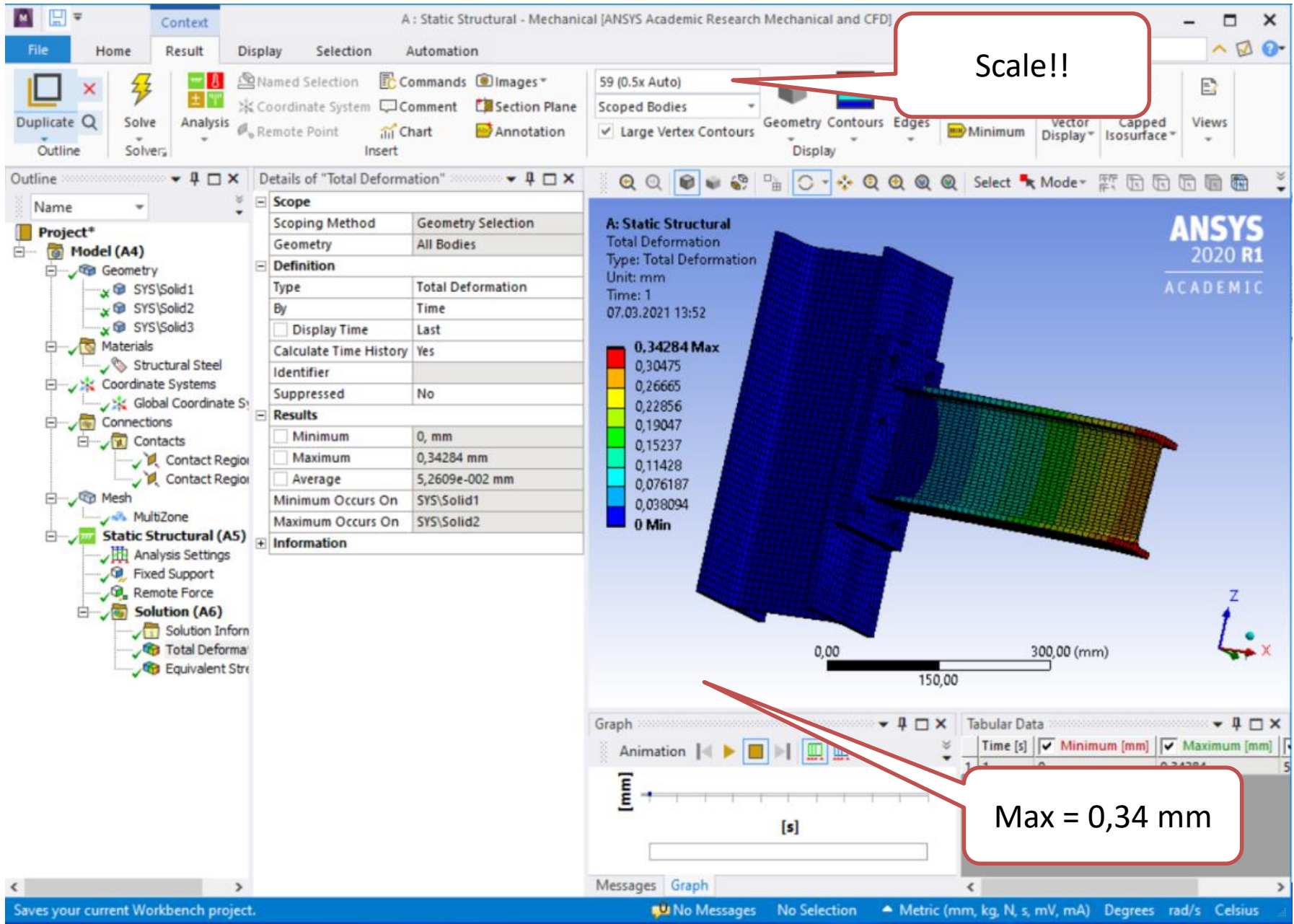
Steps	Time [s]	X [N]	Y [N]	Z [N]
1	0	0	0	0
2	1	0	0	-10000

Tabular Data

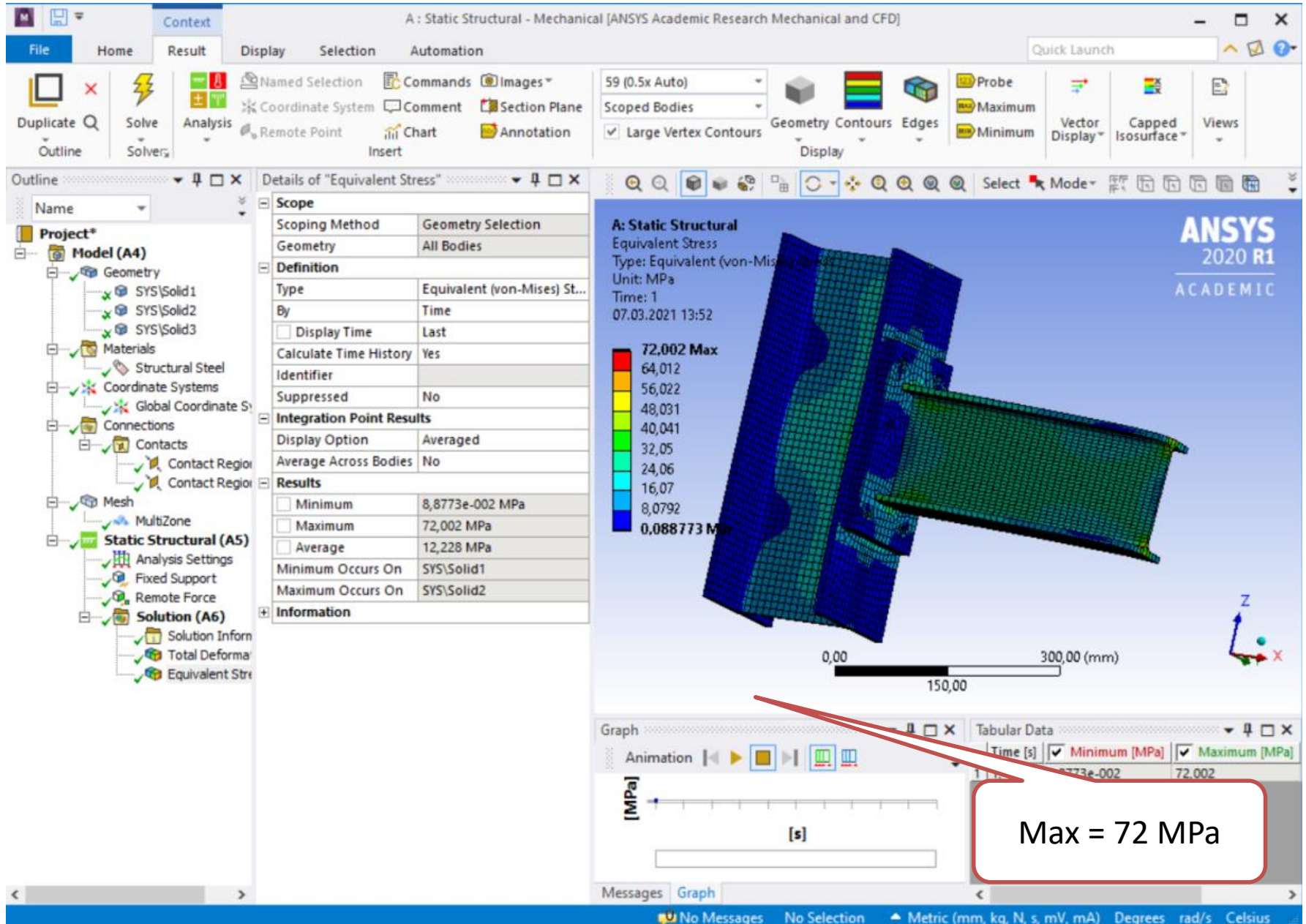
Steps	Time [s]	X [N]	Y [N]	Z [N]
1	0	0	0	0
2	1	0	0	-10000

Remote Force Define by: Components Z = -10000 N

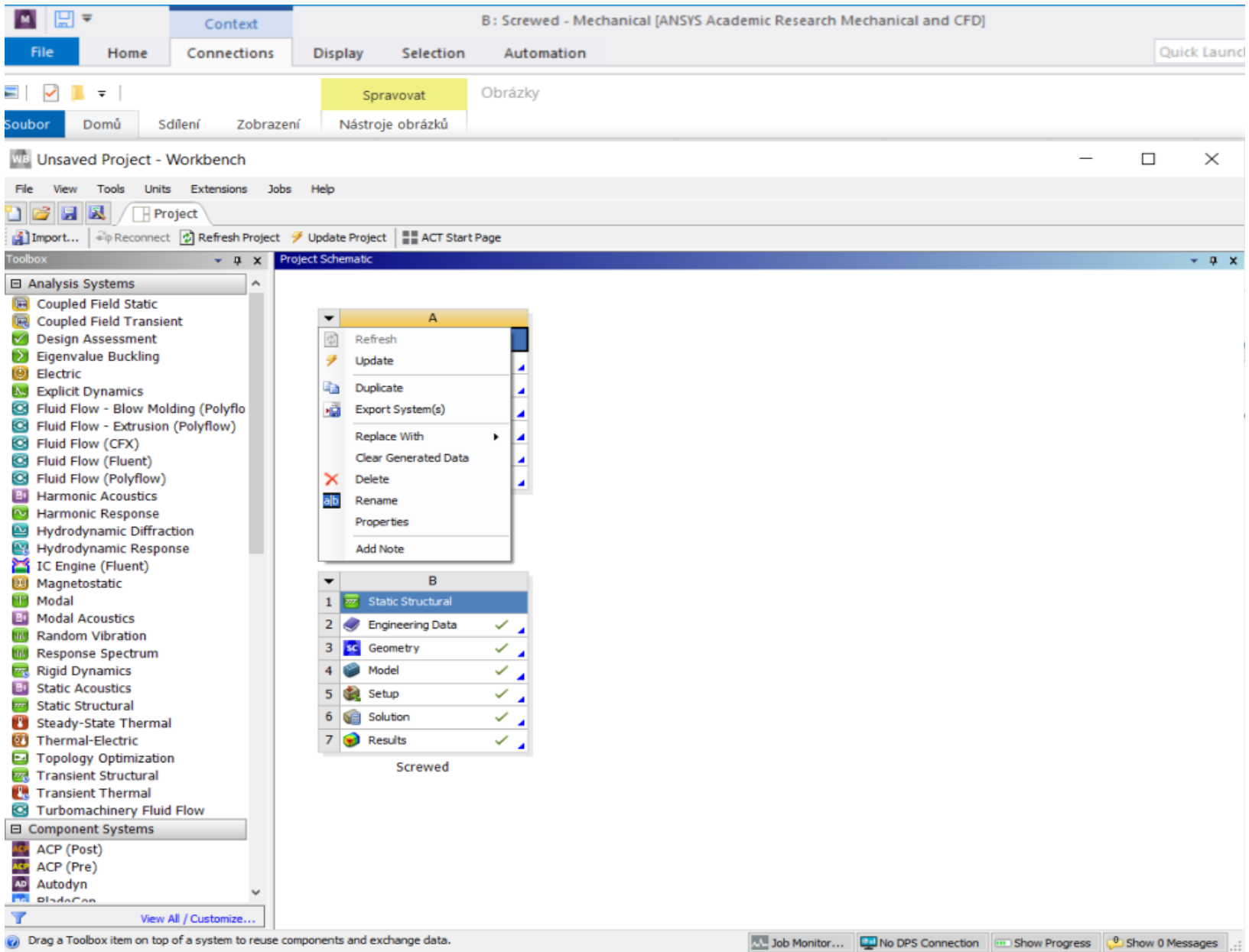
Výsledky – Total Deformation



Výsledky – Equivalent Stress



2. příklad: Duplicate project



Connections – Contacts – Frictional

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a mechanical assembly with a frictional contact defined between two solids. The details panel on the left provides the following information:

Details of "Frictional - SYS\Solid1 To SYS\Solid3"	
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	SYS\Solid1
Target Bodies	SYS\Solid3
Protected	No
Definition	
Type	Frictional
Friction Coefficient	0,11
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	2,0037 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Small Sliding	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Stabilization Damping Factor	0,
Pinball Region	Program Controlled
Time Step Controls	None
Geometric Modification	
Interface Treatment	Add Offset, No R...
Offset	0, mm
Contact Geometry Correction	None
Target Geometry Correction	None

The 3D model view shows the contact between two solids, with a red callout box highlighting the friction coefficient value of 0,11. The software version is ANSYS 2020 R1 Academic. The status bar at the bottom indicates "Ready" and "No Messages".

Connections – Joint – Body-Body -Revolute

The screenshot displays the ANSYS Mechanical software interface. The top ribbon is set to the 'Connections' tab. The 'Joint' button is highlighted, and a context menu is open, showing the 'Revolute' option selected. The 'Revolute' option is further detailed in a tooltip: 'Permits only rotation about the Z axis.' The main 3D view shows two cylindrical parts, one green and one blue, with a revolute joint being applied to their inner surfaces. A red callout box points to the selected surfaces with the text: 'Vybrat dvě plochy – cylindrical = vnitřní otvor (ctrl+klik)'. Another red callout box points to the 'Revolute' option in the context menu with the text: 'Joint – Body-Body – Revolute'. The bottom status bar shows '2 Cylinders Selected: Area = 1694,3 mm²'.

Permits only rotation about the Z axis.

No Messages 2 Cylinders Selected: Area = 1694,3 mm² Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Connections – 6 spojovacích prvků

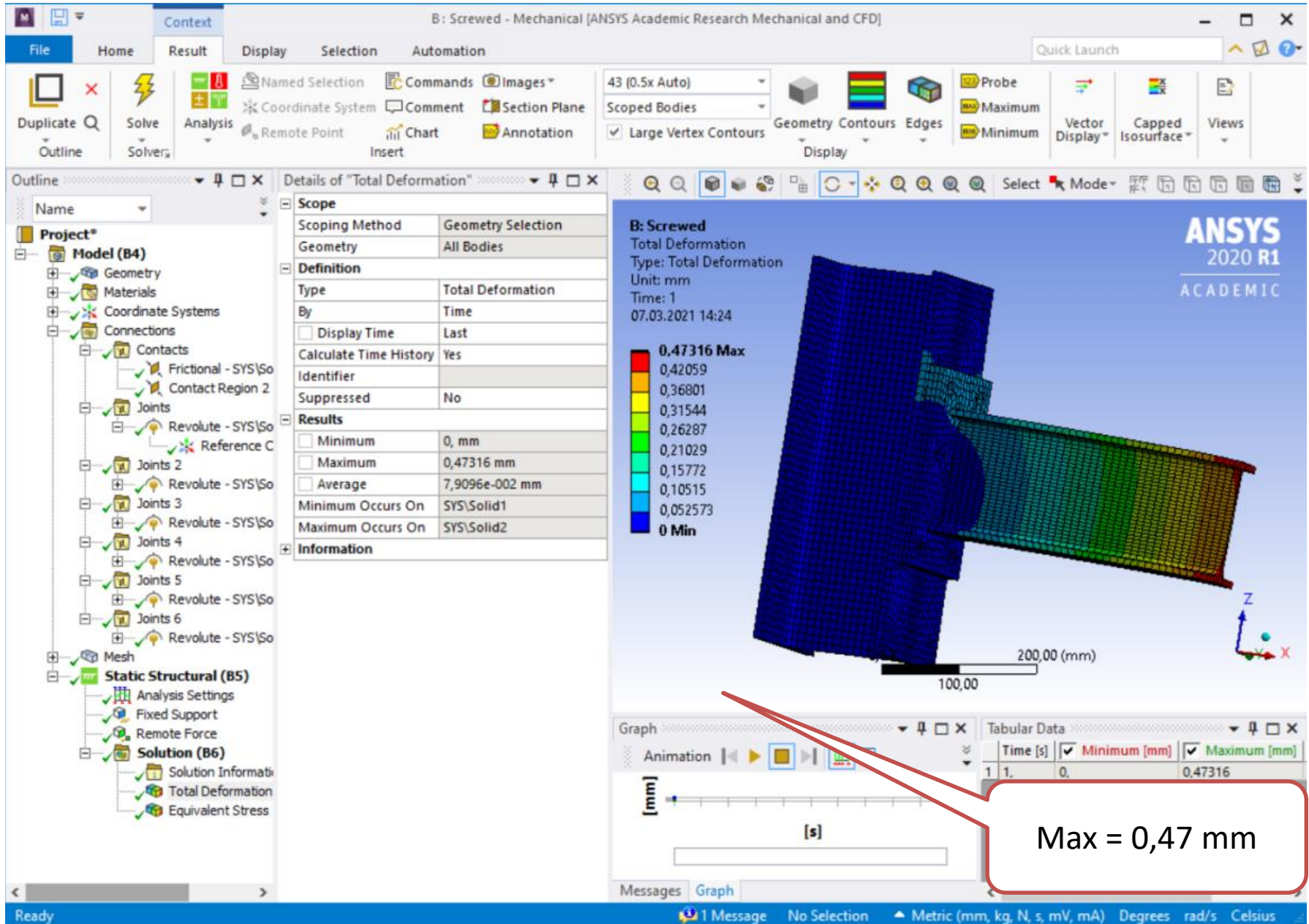
The screenshot displays the ANSYS Mechanical software interface for a model named "Screwed - Mechanical". The "Connections" tab is active, and a "Revolute" joint is defined between two bodies.

Details of "Revolute - SYS\Solid3 To SYS\Solid1"

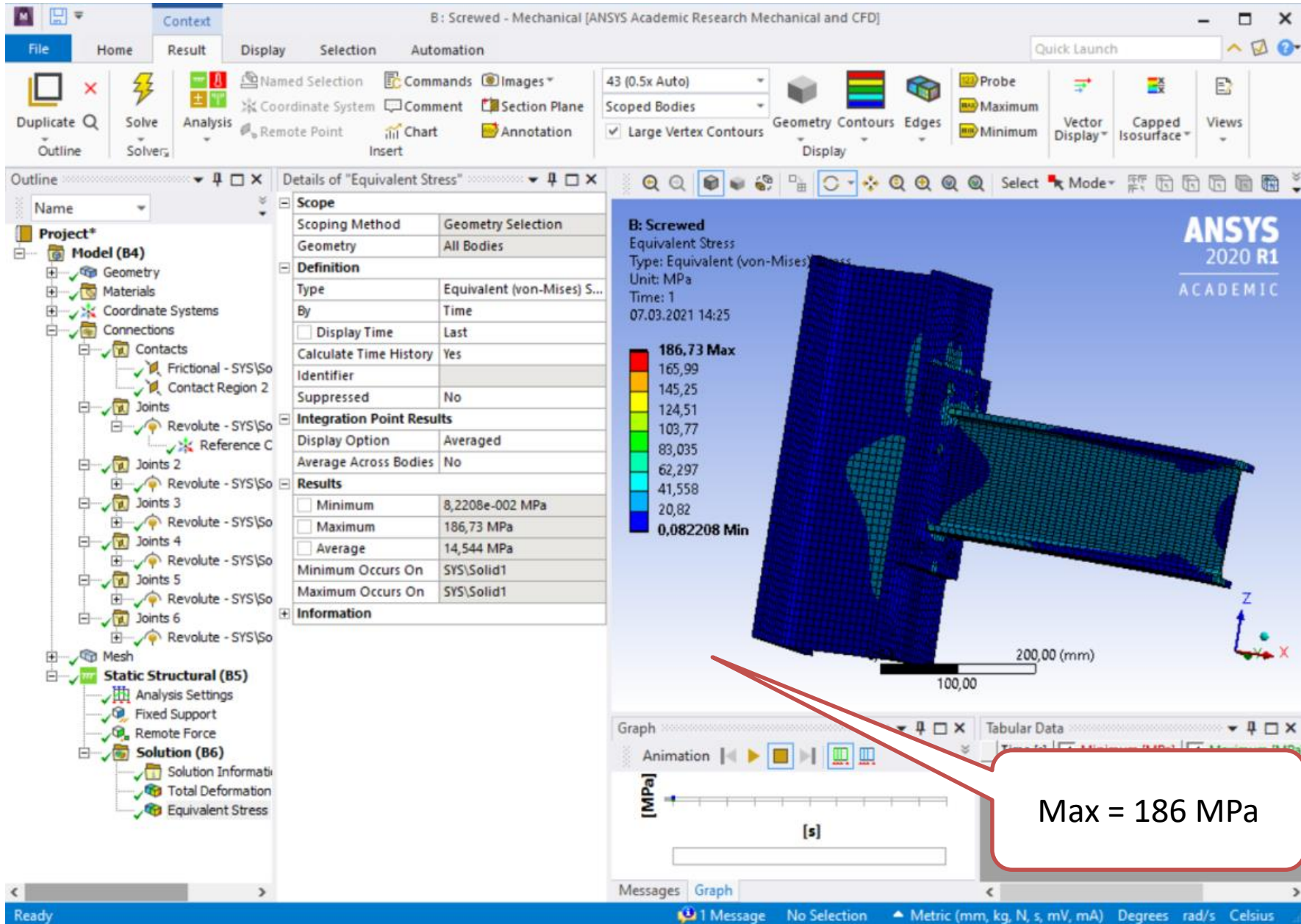
Definition	
Connection Type	Body-Body
Type	Revolute
Torsional Stiffness	0, N-mm/*
Torsional Damping	0, N-mm-s/*
Suppressed	No
Element APDL Name	
Reference	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	SYS\Solid3
Coordinate System	Reference Coordinate Syst...
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	SYS\Solid1
Initial Position	Unchanged
Behavior	Rigid
Pinball Region	All
Stops	
RZ Min Type	None
RZ Max Type	None

The interface also shows a 3D model of a green plate with a hole and a blue plate with a hole, connected by a revolute joint. The coordinate system (X, Y, Z) is visible. The software version is ANSYS 2020 R1 ACADEMIC. The status bar at the bottom indicates "No Messages", "No Selection", and units: Metric (mm, kg, N, s, mV, mA), Degrees, rad/s, Celsius.

Výsledky – Total Deformation



Výsledky – Equivalent Stress



Výsledky – změna pohledu bez sítě - Edges

Context B: Screwed - Mechanical [ANSYS Academic Research Mechanical and CFD]

File Home Result Display Selection Automation Quick Launch

Duplicate Outline Solve Solvers Analysis Coordinate System Remote Point Commands Comment Chart Annotation Section Plane Image

43 (0.5x Auto) Scoped Bodies Large Vertex Contours

Geometry Contours Edges Probe Maximum Minimum Vector Display Capped IsoSurface Views

Display No WireFrame Show Undeformed WireFrame Show Undeformed Model Show E

Show Undeformed WireFrame
Display the result with a wireframe overlay of the undeformed model.
Press F1 for help.

B: Screwed
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
07.03.2021 14:25

186.73 Max
165.99
145.25
124.51
103.77
83.035
62.297
41.558
20.82
0.082208 Min

200,00 (mm)
100,00

Graph
Animation [MPa] [s]

Messages Graph

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

Outline

Name

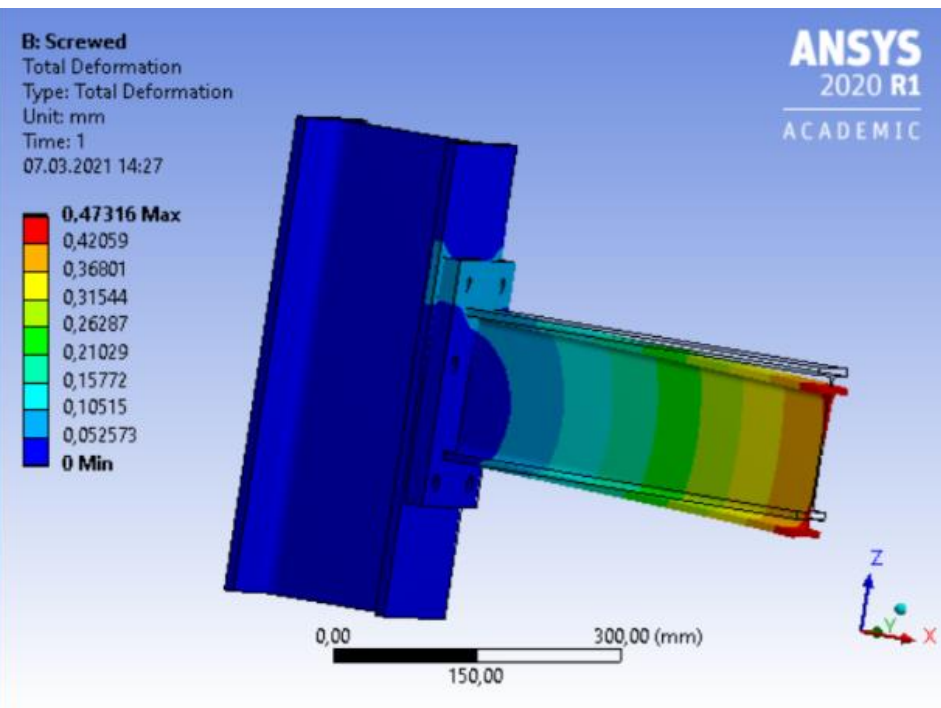
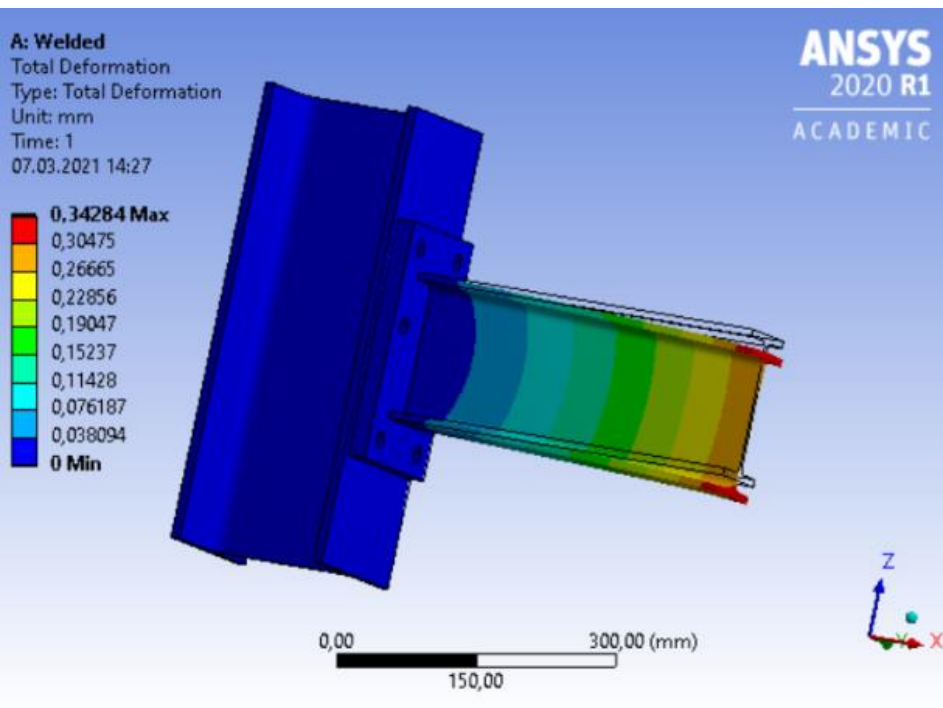
Project*

- Model (B4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Connections
 - Contacts
 - Frictional - SYS\S0
 - Contact Region 2
 - Joints
 - Revolute - SYS\S0
 - Reference C
 - Joints 2
 - Revolute - SYS\S0
 - Joints 3
 - Revolute - SYS\S0
 - Joints 4
 - Revolute - SYS\S0
 - Joints 5
 - Revolute - SYS\S0
 - Joints 6
 - Revolute - SYS\S0
 - Mesh
 - Static Structural (B5)
 - Analysis Settings
 - Fixed Support
 - Remote Force
 - Solution (B6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress

Porovnání výsledků – Total Deformation

„Svařované“

„Šroubované“



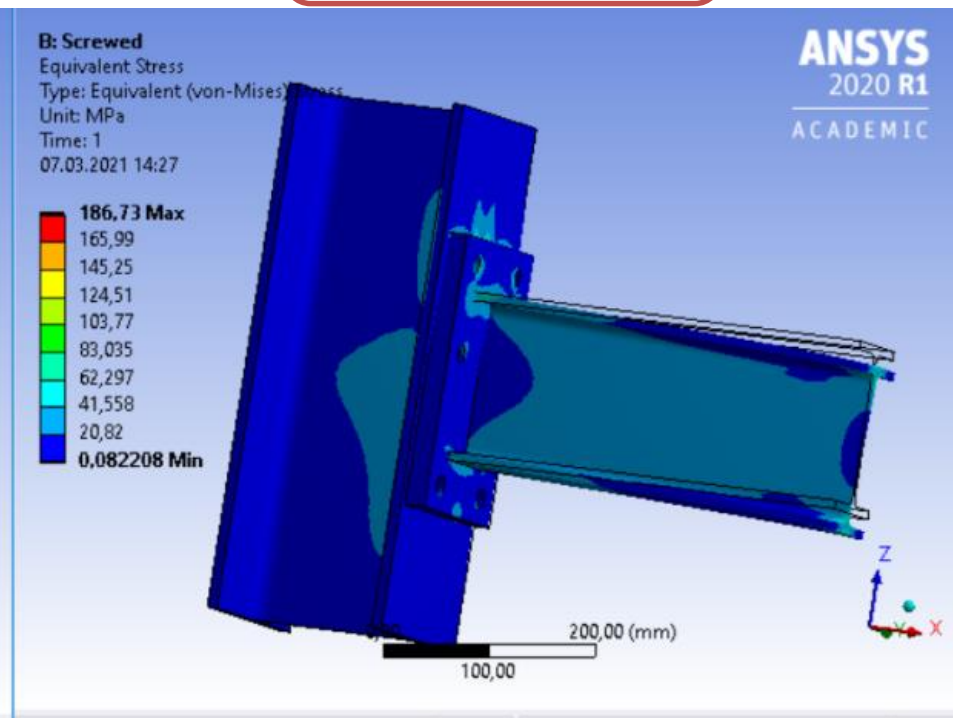
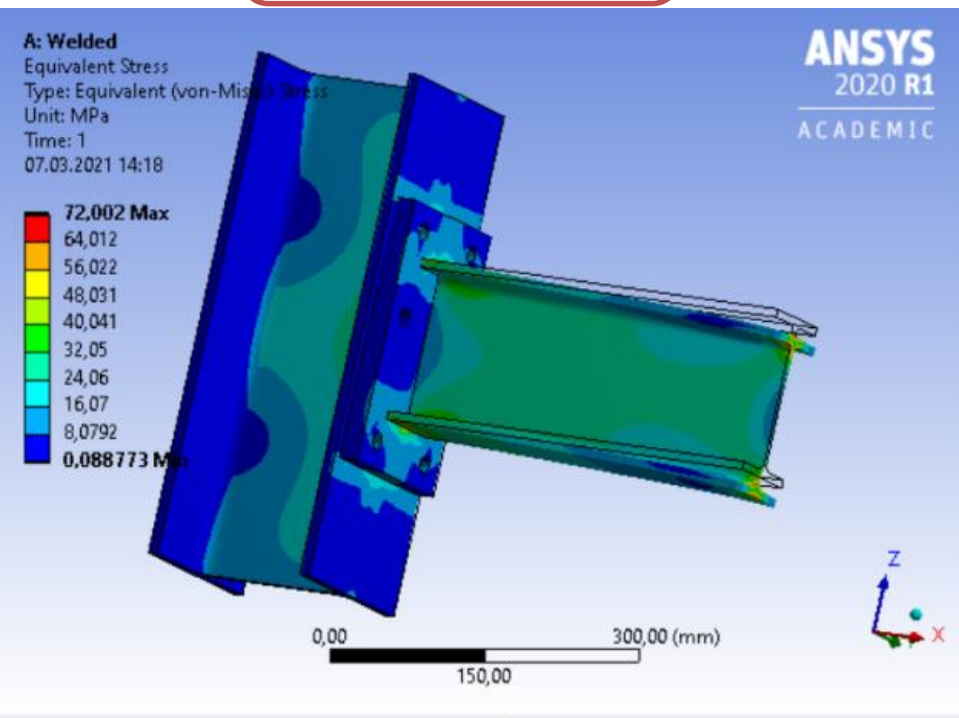
Max = 0,34 mm

Max = 0,47 mm

Porovnání výsledků – Equivalent Stress

„Svařované“

„Šroubované“



Max = 72 MPa

Max = 186 MPa