



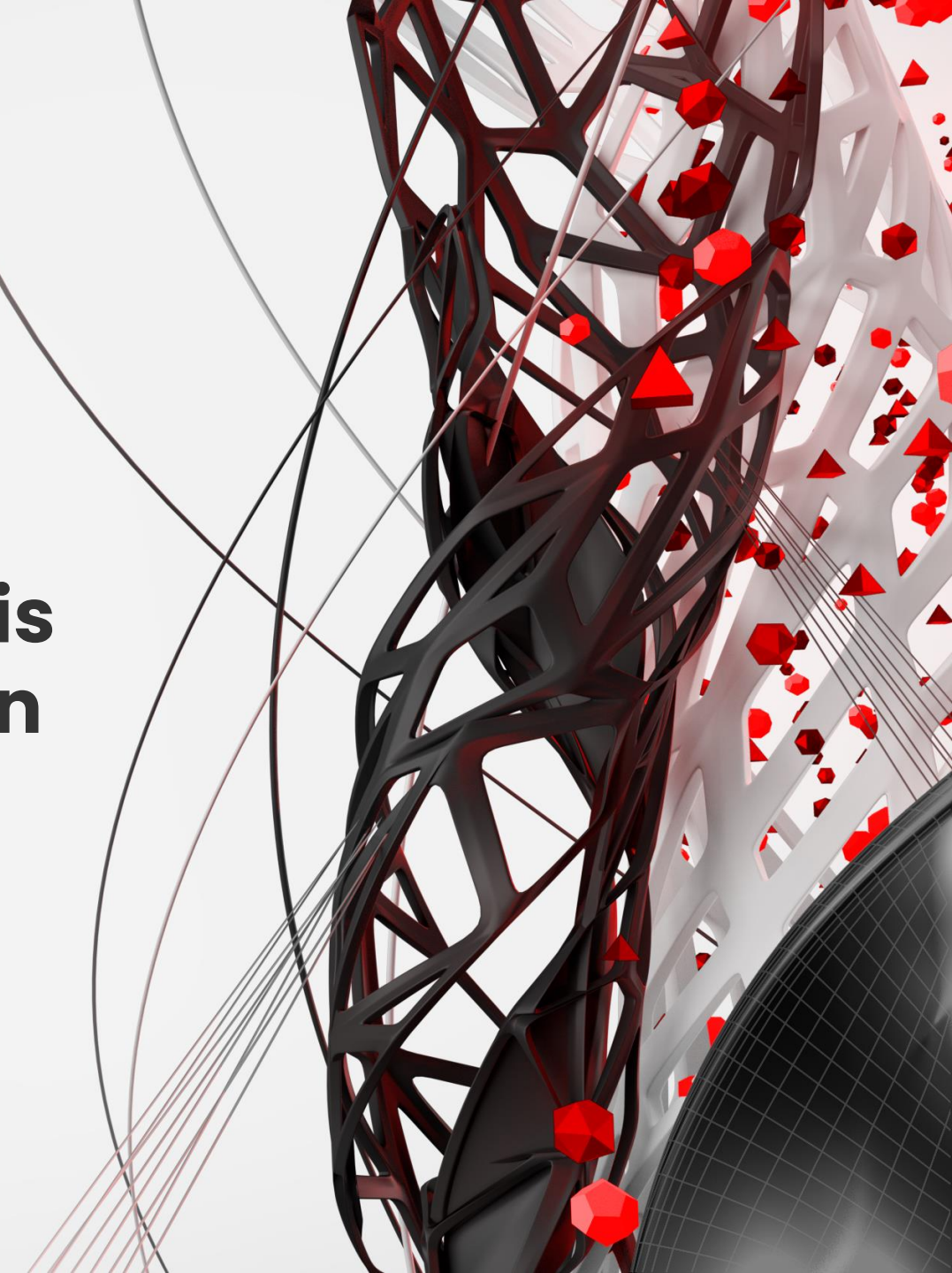
# Workshop Structural Analysis Stepped shaft in axial tension

**JOSÉ CARLOS ZART**

CAE Applications Intern Academic Area. Latin America/Spain


[jzart@esss.co](mailto:jzart@esss.co) / [www.esss.co](http://www.esss.co)

**SIMULATING THE FUTURE**



# Table of contents

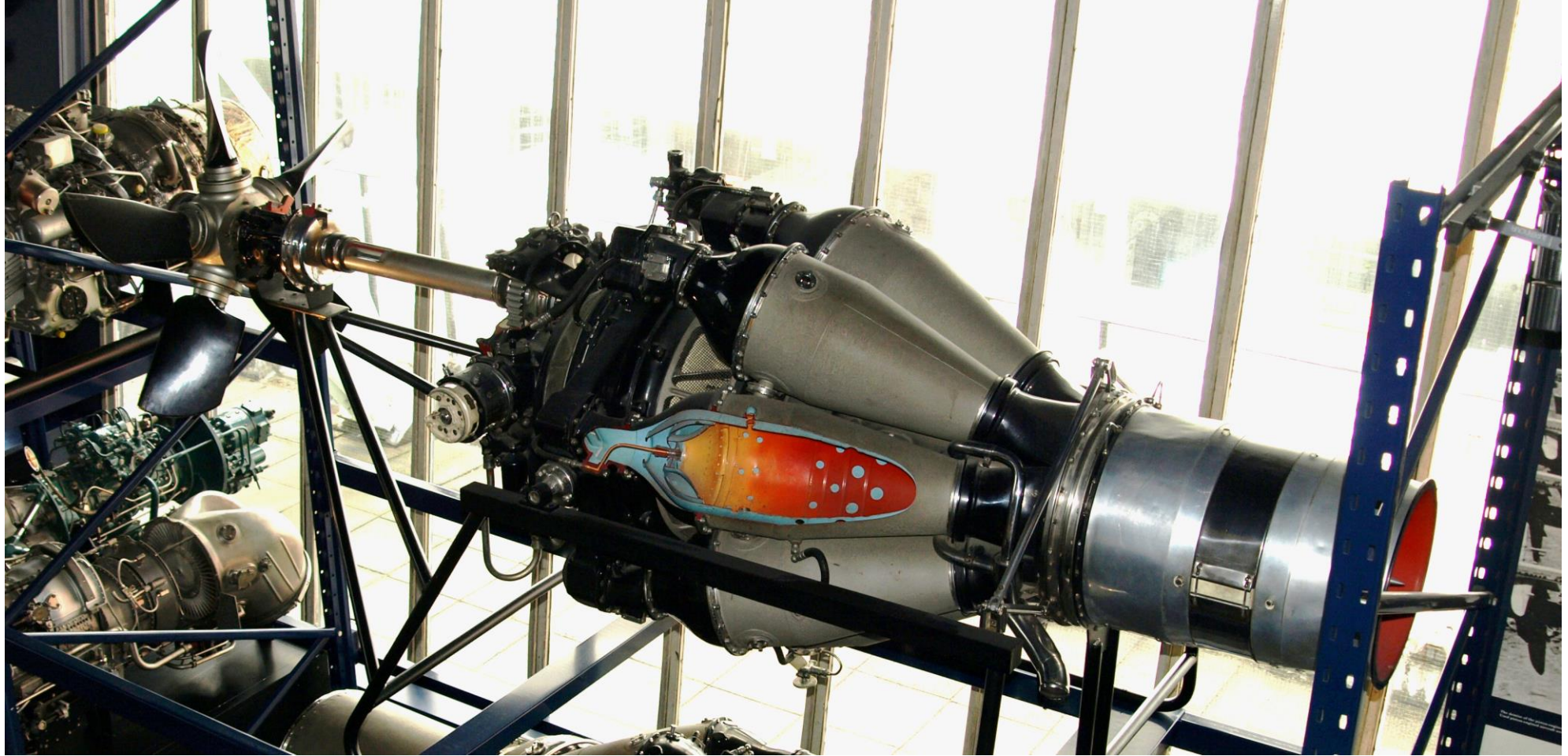
- 1 Introduction
- 2 Problem Specification
- 3 Pre-Analysis
- 4 Geometry
- 5 Mesh

- 6 Physics Setup
- 7 Numerical Solution
- 8 Numerical Results
- 9 Verification
-  Exercises

1

# Introduction

# 1. INTRODUCTION

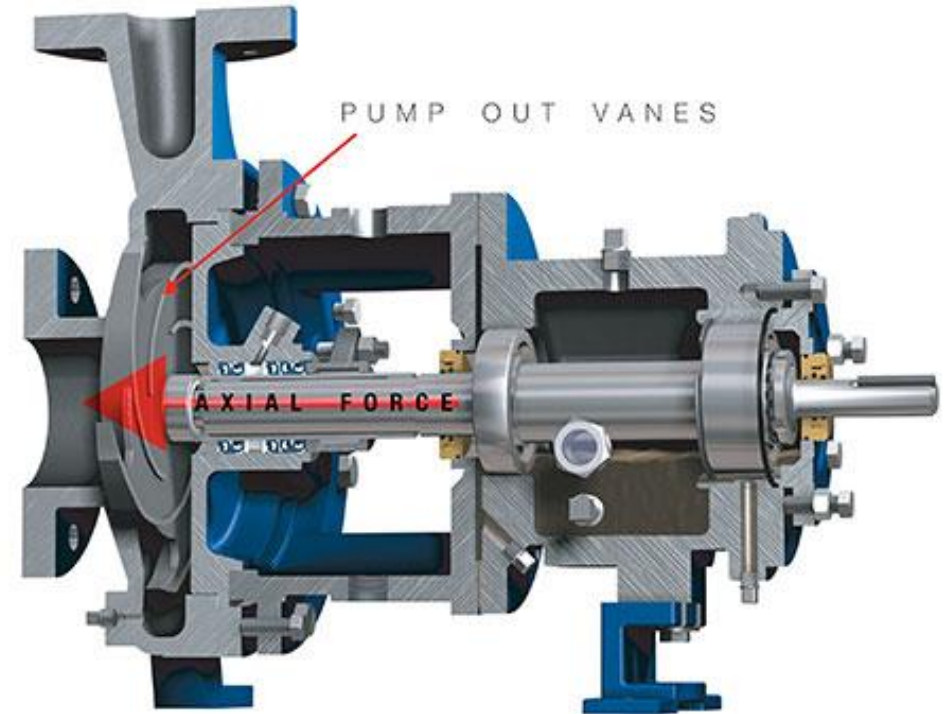


[https://en.wikipedia.org/wiki/Rolls-Royce\\_RB.50\\_Trent](https://en.wikipedia.org/wiki/Rolls-Royce_RB.50_Trent)

# 1. INTRODUCTION



Árbol de levas VW Tiguan



<https://www.pumpsandsystems.com/why-are-there-holes-my-new-impeller-part-1>

# 1. INTRODUCTION



[https://hayn.com/cables/systems/architectural\\_adjuster\\_fork\\_system.html](https://hayn.com/cables/systems/architectural_adjuster_fork_system.html)

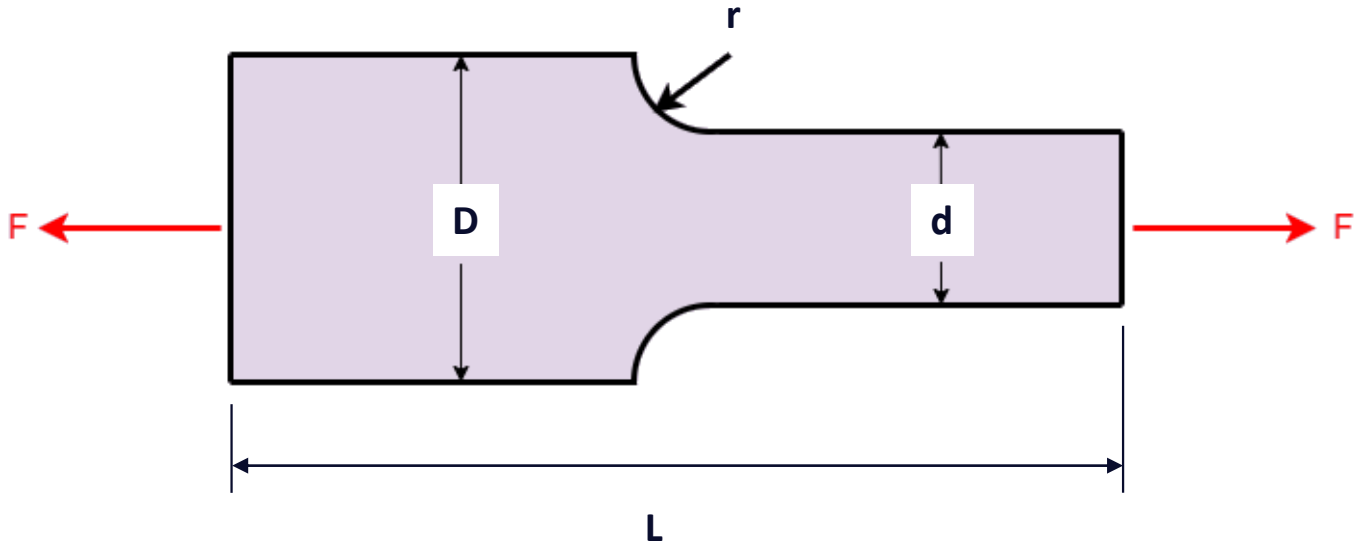


<https://blog.maxprocorp.com/the-difference-between-tension-shear-and-bending-joints>

2

# Problem Specification

# 2. PROBLEM SPECIFICATION



Value	Unit
$D = 100$	mm
$d = 75$	mm
$r = 10$	mm
$L = 400$	mm
$F = 3,0E5$	N



# 2. PROBLEM SPECIFICATION

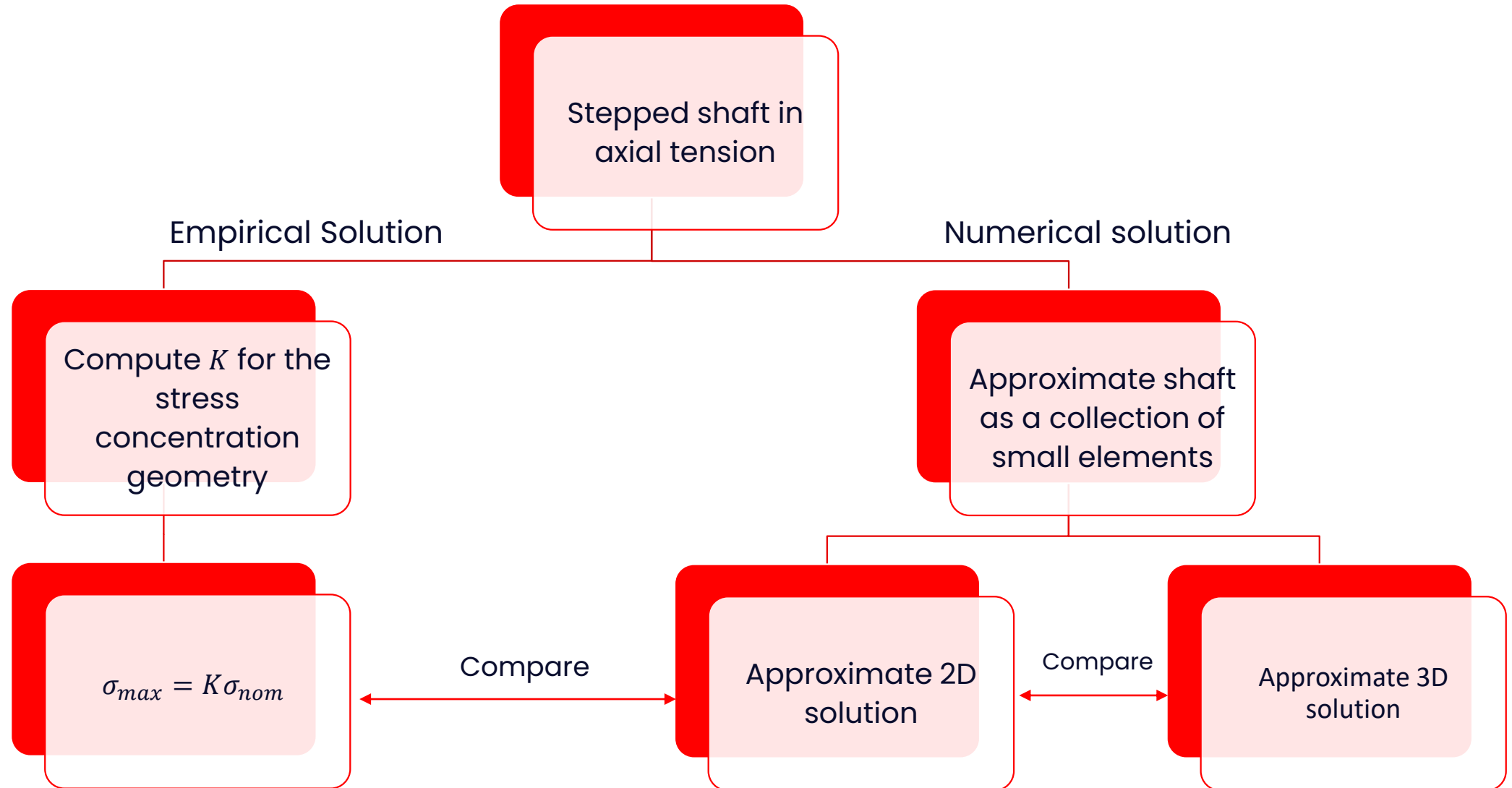
## AISI 4320 Normalized

Property	Value	Unit
Density	7,9e3	kg/m <sup>3</sup>
Young's Modulus	212	GPa
Poisson's ratio	0,295	-
Yield Strength	515	MPa
Tensile Strength	875	MPa
Compressive Strength	515	MPa

3

# Pre-Analysis

# 3. PRE-ANALYSIS



# 3. PRE-ANALYSIS

$$\sigma_{max} = K\sigma_{nom}$$

$$\sigma_{nom} = \frac{F}{A_{min}} = \frac{4F}{\pi d^2}$$

$$K = C_1 + C_2 \frac{2h}{D} + C_3 \left(\frac{2h}{D}\right)^2 + C_4 \left(\frac{2h}{D}\right)^3$$

$$h = \frac{D - d}{2}$$

Roark's Formulas for Stress and Strain, Warren C. Young and Richard G. Budynas, 2002

$$C_1 = 1,225 + 0,831 \sqrt{\frac{h}{r}} - 0,01 \left(\frac{h}{r}\right)$$

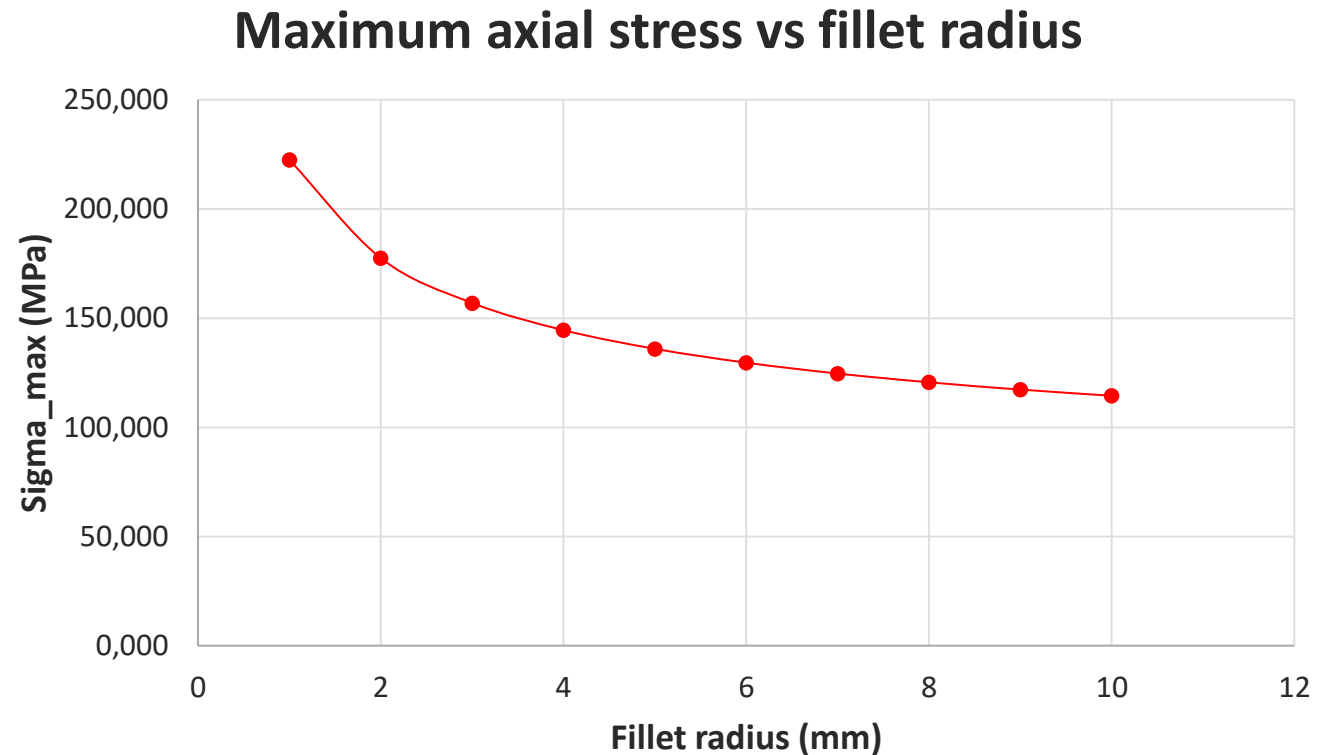
$$C_2 = -1,831 - 0,318 \sqrt{\frac{h}{r}} - 0,049 \left(\frac{h}{r}\right)$$

$$C_3 = 2,236 - 0,522 \sqrt{\frac{h}{r}} + 0,176 \left(\frac{h}{r}\right)$$

$$C_4 = -0,63 + 0,009 \sqrt{\frac{h}{r}} - 0,117 \left(\frac{h}{r}\right)$$

# 3. PRE-ANALYSIS

r (mm)	K	$\sigma_{max}$ (MPa)
1	3,276	222,446
2	2,613	177,436
3	2,310	156,887
4	2,127	144,459
5	2,001	135,902
6	1,908	129,546
7	1,835	124,582
8	1,775	120,567
9	1,726	117,230
10	1,685	114,401

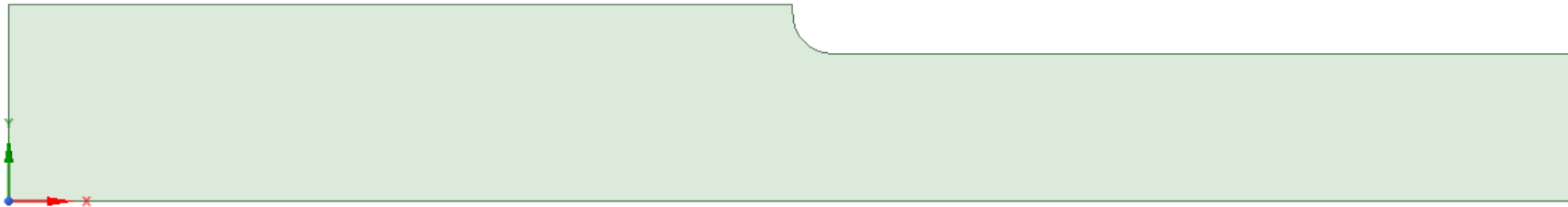


4

# Geometry

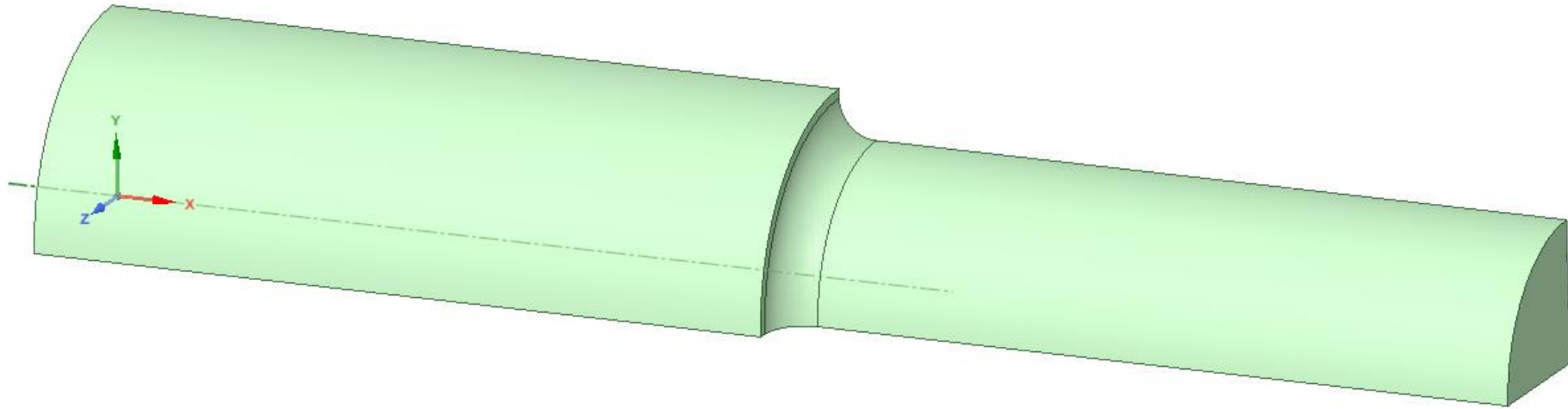
# 4. GEOMETRY

2D - Symmetric



# 4. GEOMETRY

3D - Symmetric





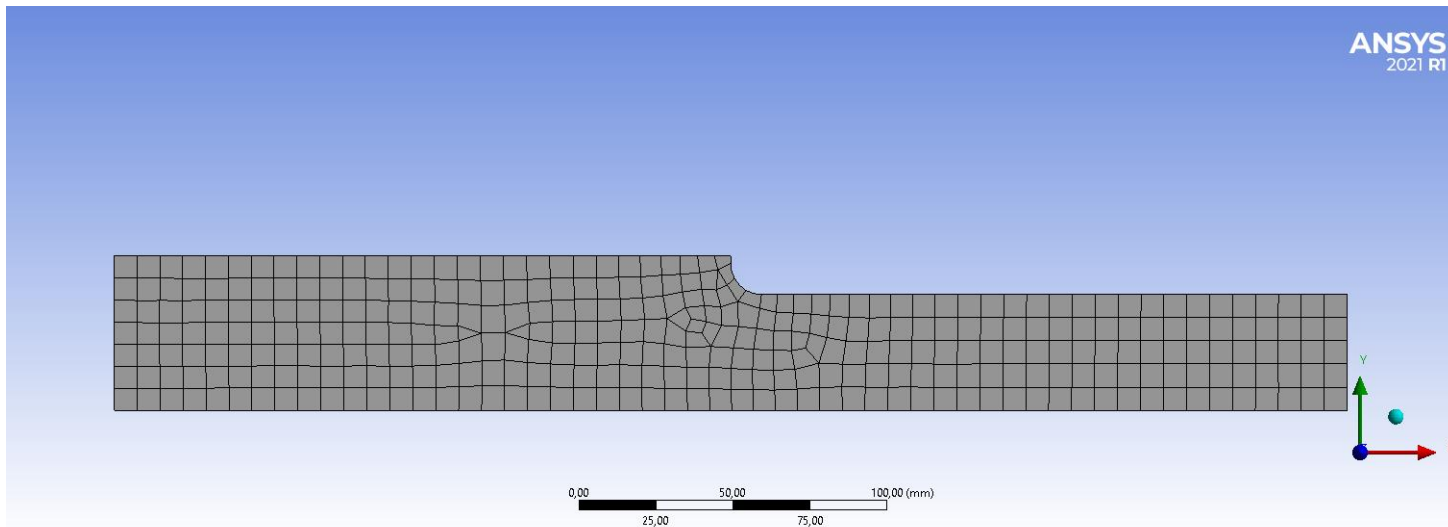
5

Mesh

# 5. MESH

## 2D-Mesh – nonlinear elements

- Tri-6
- Quad-8

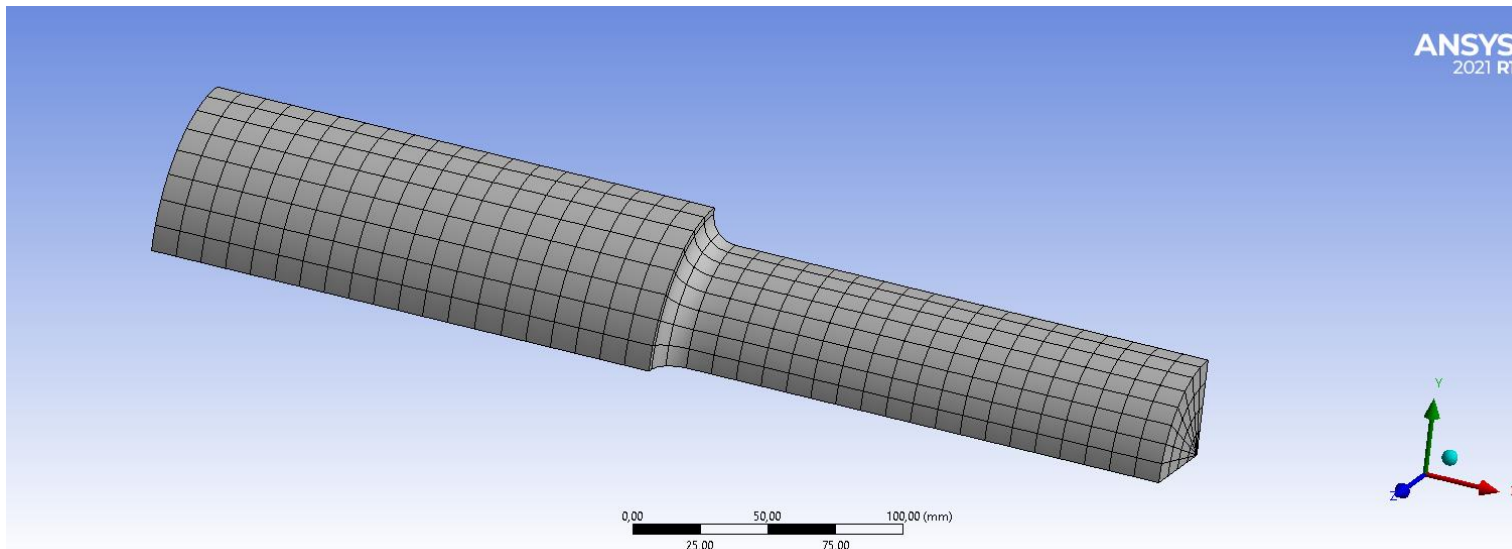


Face Sizing (mm)
7,5
5
3,33
2,22
1,48
0,99
0,66
0,44

# 5. MESH

## 3D-Mesh – nonlinear elements

- Hex-20
- Wed-15



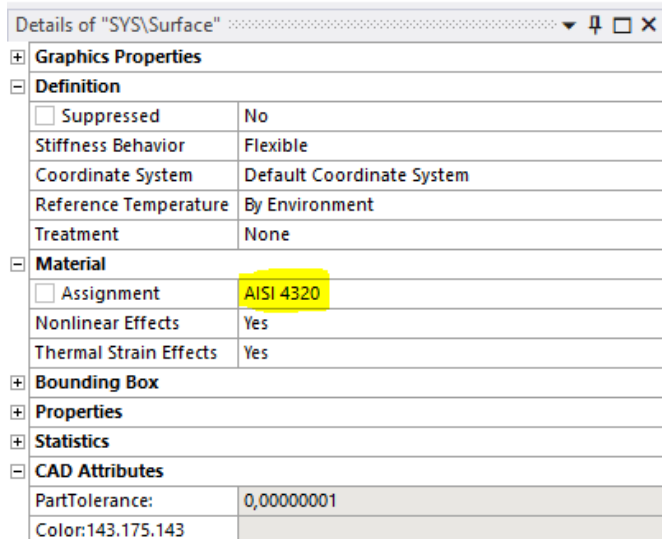
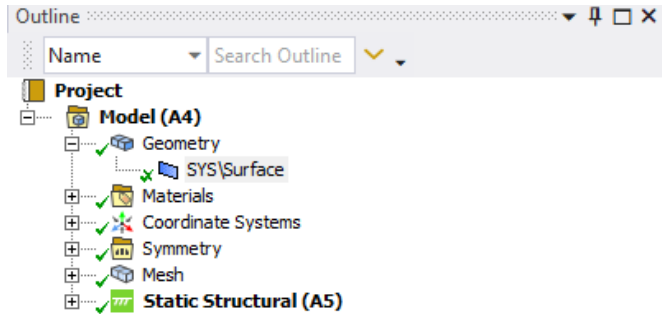
Body Sizing (mm)
10
8
6
4
2,5
2
1,5
1

6

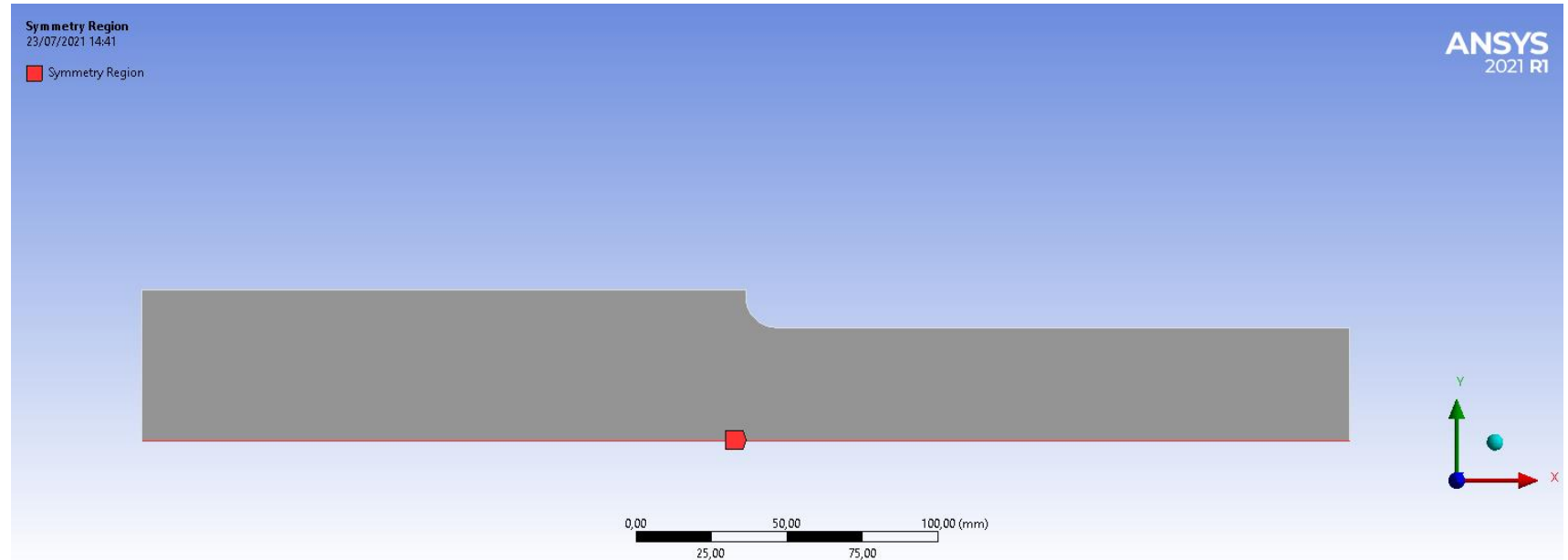
# Physics Setup

# 6. PHYSICS SETUP

## Material Assignment

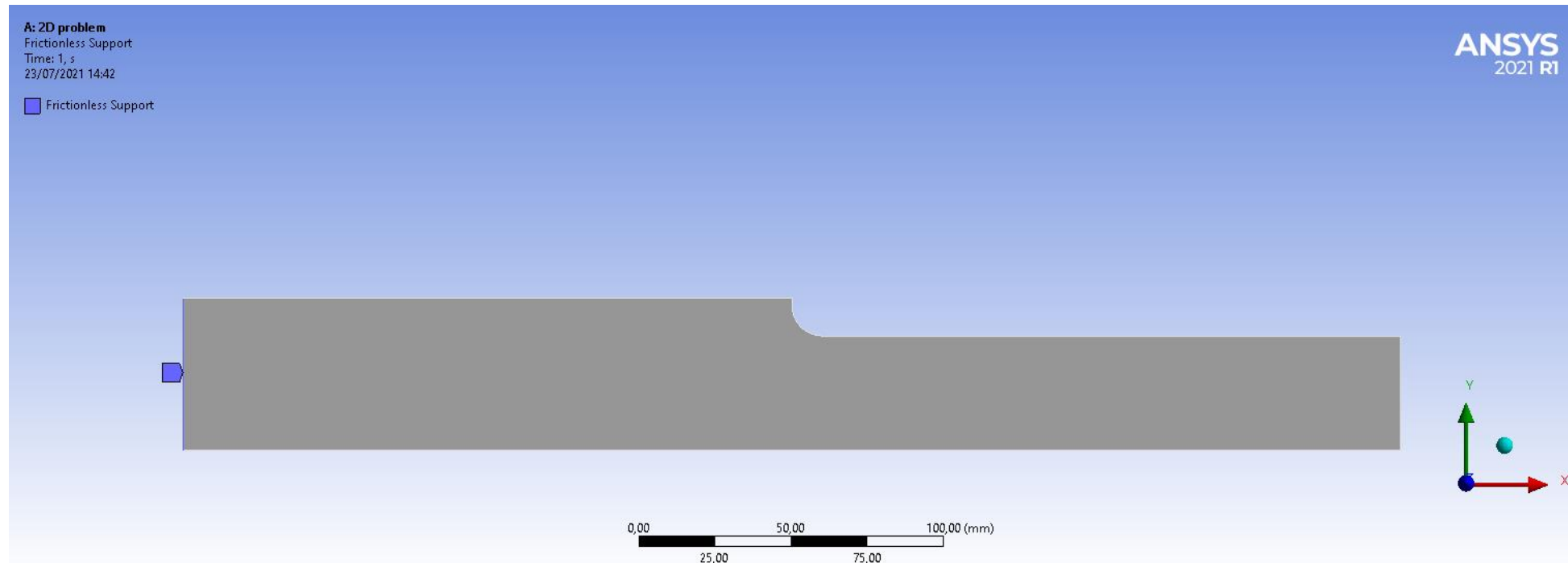


## Symmetry Region – 2D problem



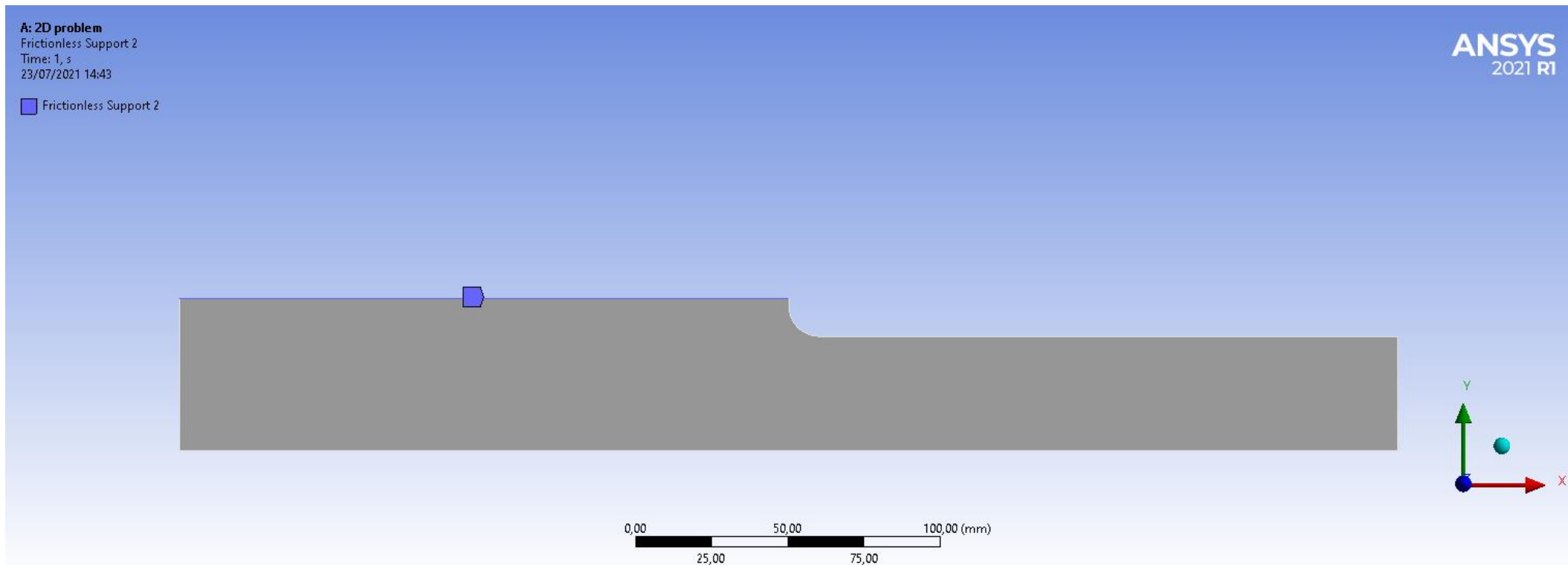
# 6. PHYSICS SETUP

## Frictionless Support – 2D problem



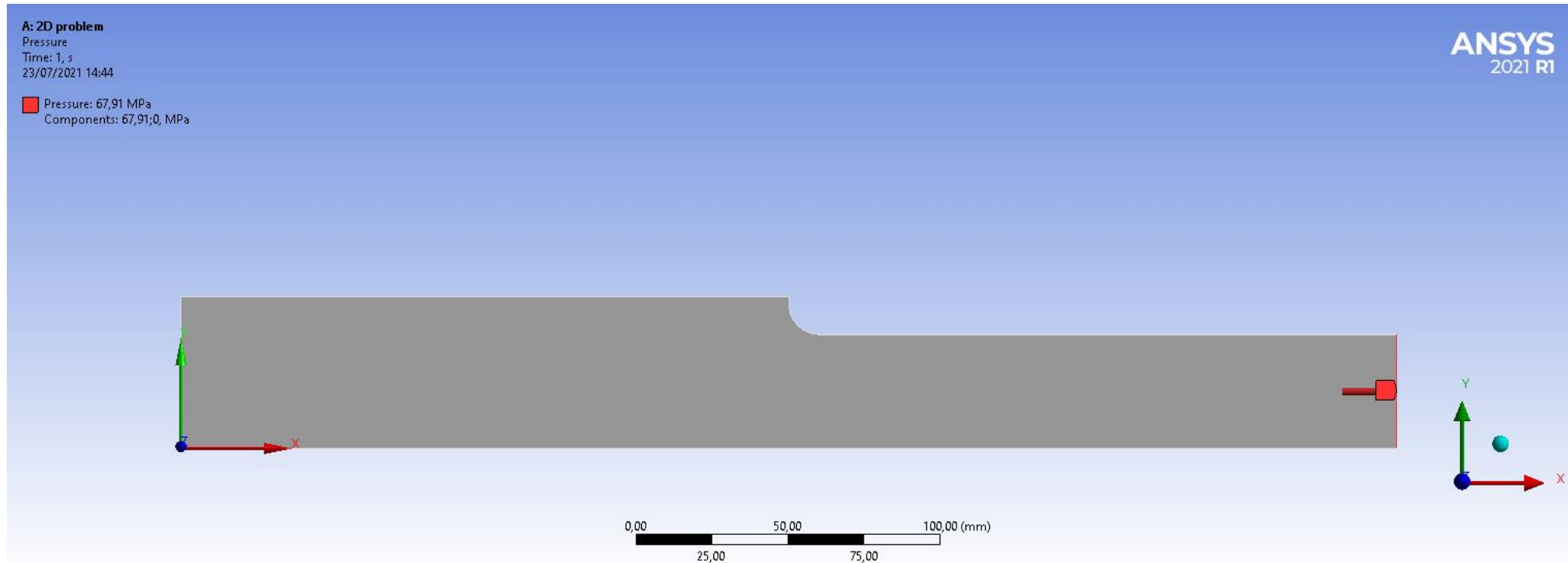
# 6. PHYSICS SETUP

## Frictionless Support 2 – 2D problem



# 6. PHYSICS SETUP

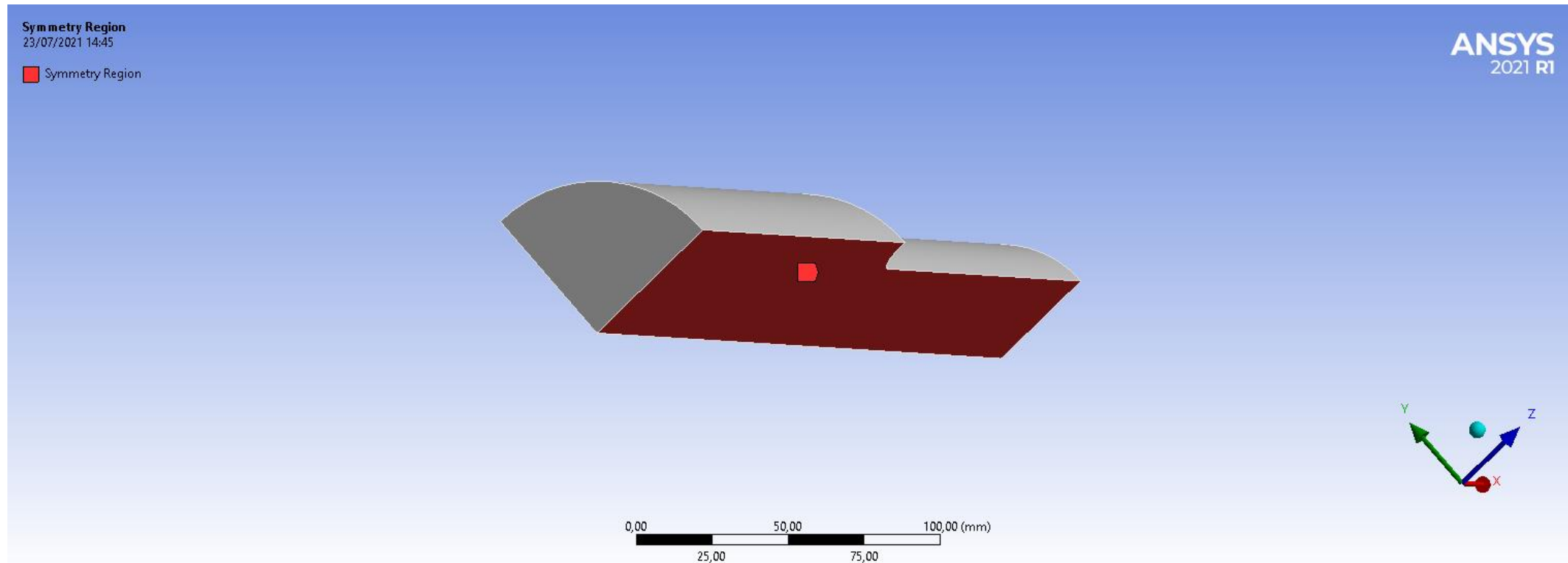
**Pressure = 67,91 MPa (2D Problem)**





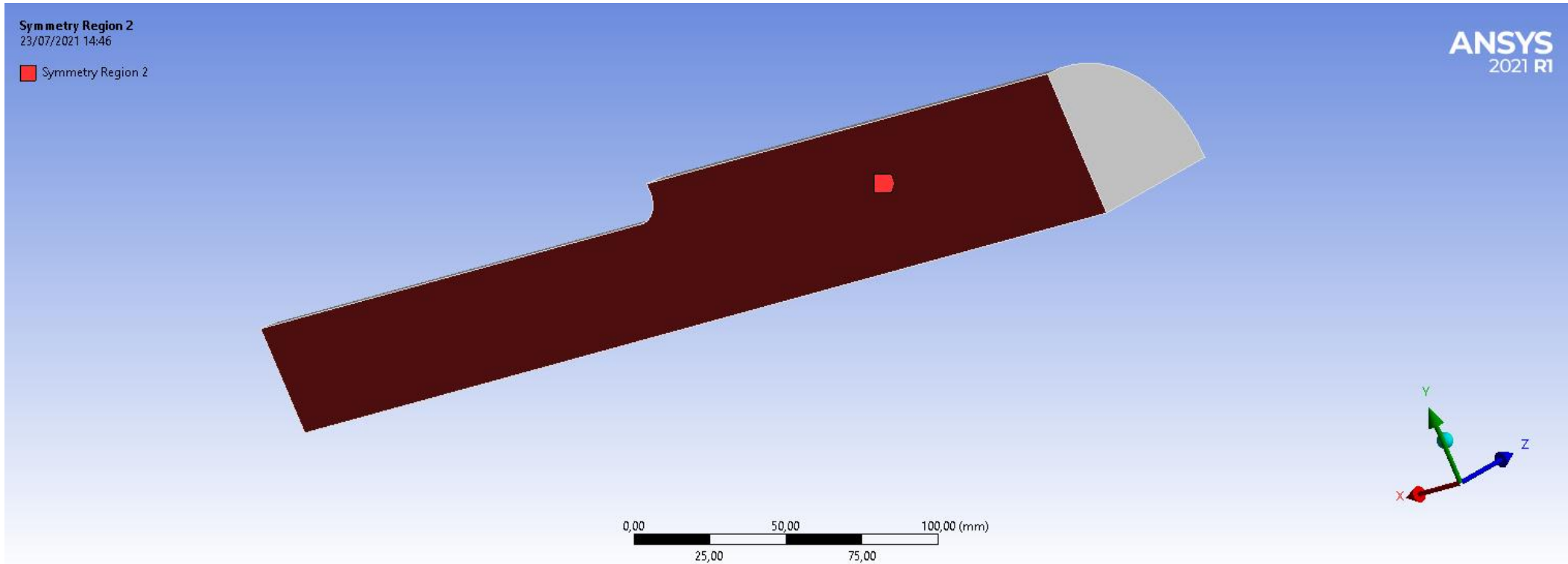
# 6. PHYSICS SETUP

## Symmetry 1 – 3D problem



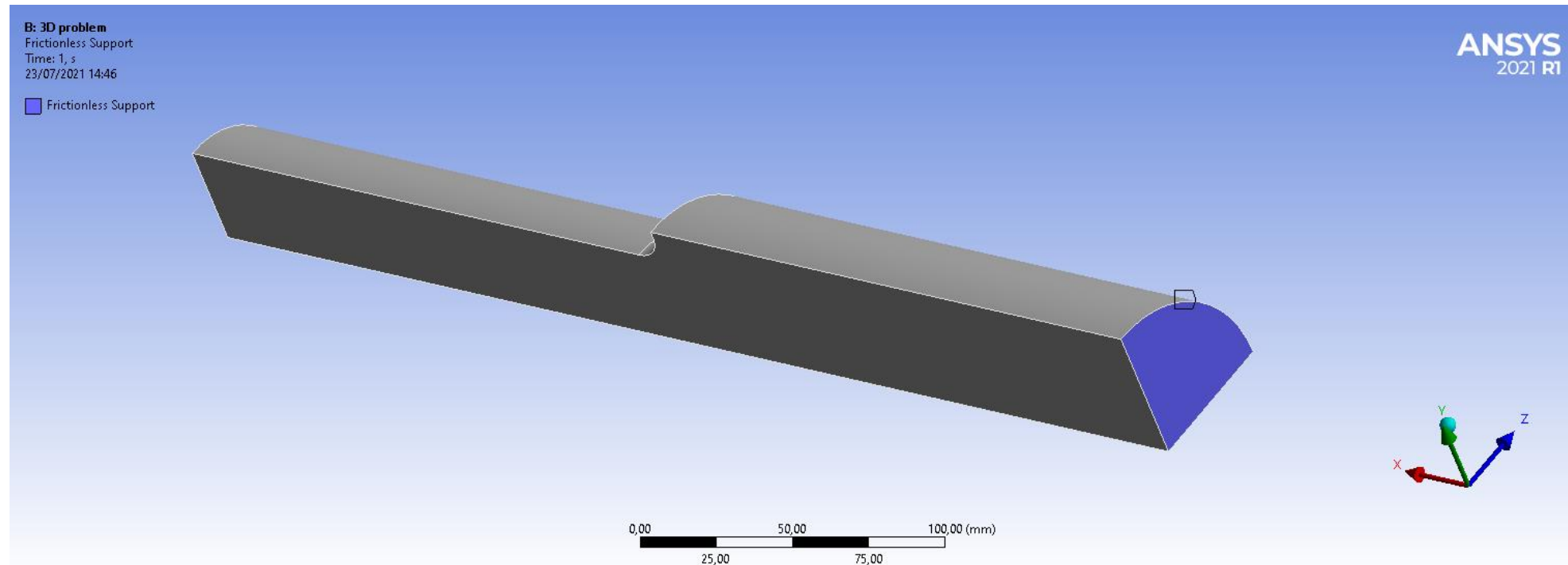
# 6. PHYSICS SETUP

## Symmetry 2 – 3D problem



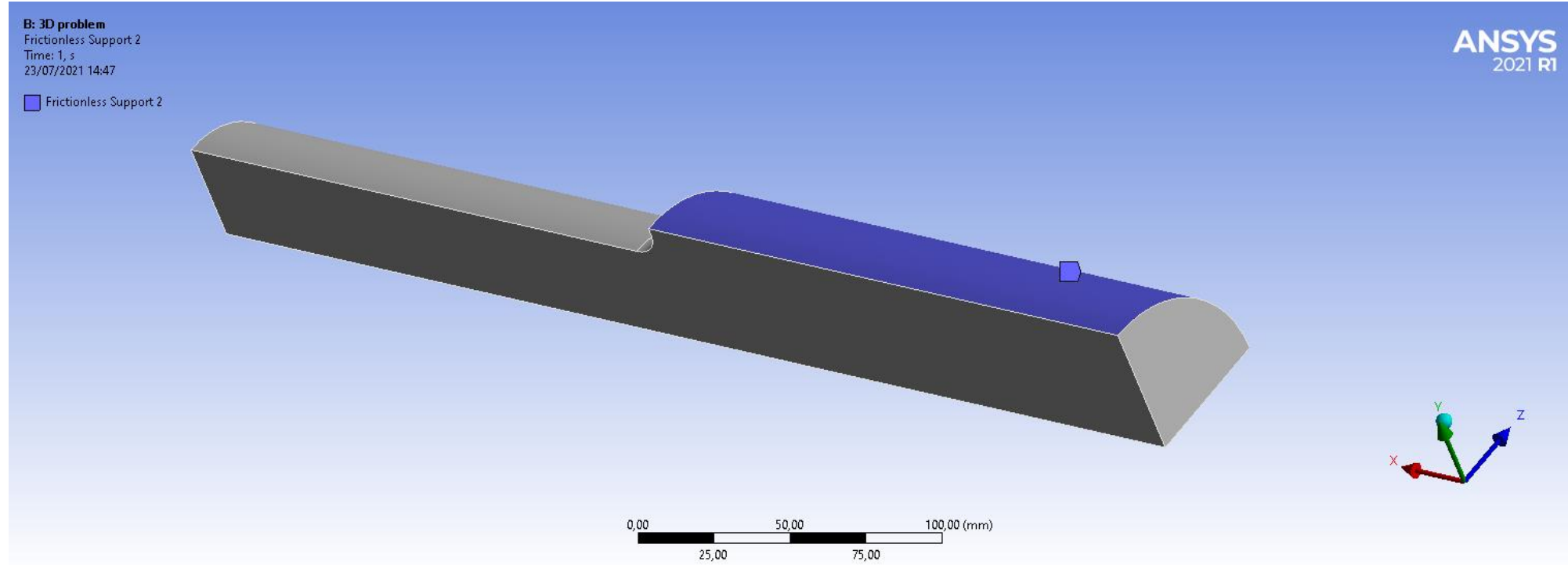
# 6. PHYSICS SETUP

## Frictionless support – 3D problem



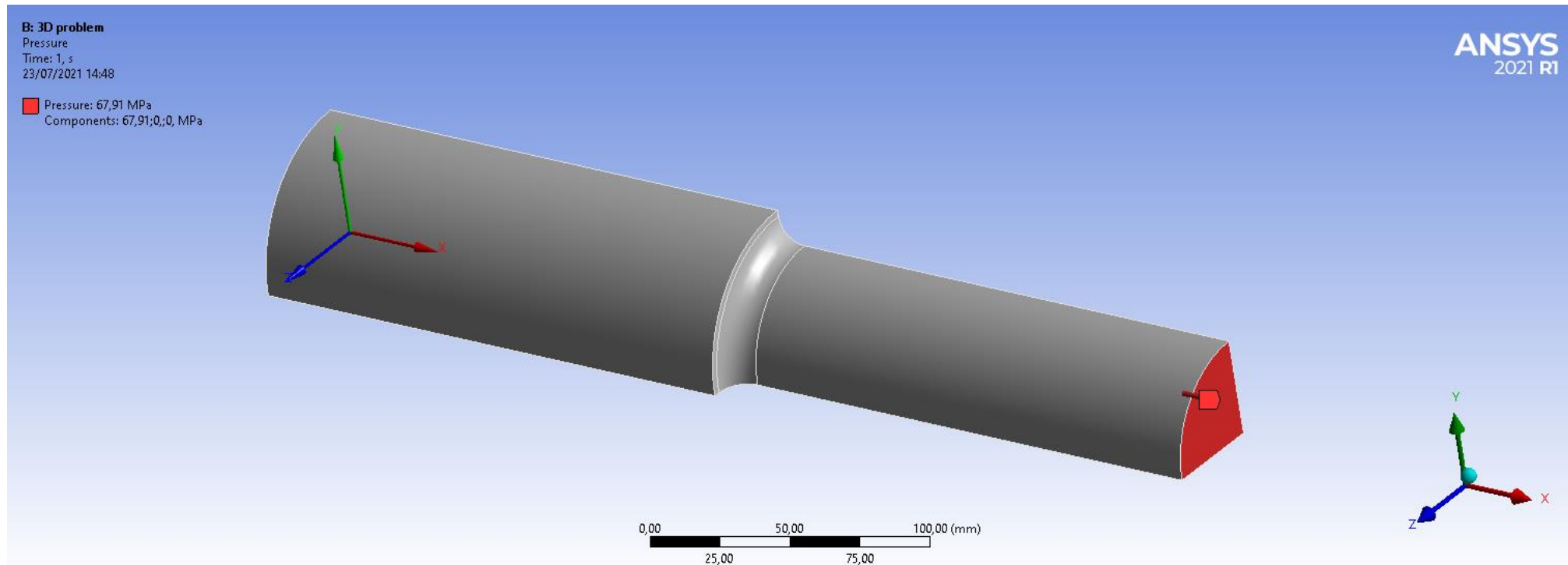
# 6. PHYSICS SETUP

## Frictionless support 2 – 3D problem



# 6. PHYSICS SETUP

**Pressure = 67,91 MPa (3D problem)**



7

# Numerical Solution

# 7. NUMERICAL SOLUTION

## Displacement

The screenshot shows the ANSYS Workbench interface for a Static Structural analysis. The tree view on the left includes 'Static Structural (B5)', 'Analysis Settings', 'Pressure', 'Frictionless Support', 'Frictionless Support 2', and 'Solution (B6)'. The 'Solution (B6)' context menu is open, showing options like 'Insert', 'Clear Generated Data', 'Rename', 'Group All Similar Children', 'Open Solver Files Directory', and 'Worksheet: Result Summary'. The 'Deformation' sub-menu is expanded, showing 'Total' and 'Directional'. The 'Total' option is highlighted with a red box. Below the tree view, the 'Details of "Solution (B6)"' panel is visible, showing settings for 'Solution', 'Adaptive Mesh Refinement', and 'Information'.

Details of "Solution (B6)"	
<b>Solution</b>	
Number Of Cores to Use (Beta)	Solve Process Settings
<b>Adaptive Mesh Refinement</b>	
Max Refinement Loops	1,
Refinement Depth	2,
<b>Information</b>	
Status	Done
<input type="checkbox"/> MAPDL Elapsed Time	3, s

## Normal Stress

The screenshot shows the ANSYS Workbench interface for a Static Structural analysis. The tree view on the left includes 'Static Structural (B5)', 'Analysis Settings', 'Pressure', 'Frictionless Support', 'Frictionless Support 2', and 'Solution (B6)'. The 'Solution (B6)' context menu is open, showing options like 'Insert', 'Clear Generated Data', 'Rename', 'Group All Similar Children', 'Open Solver Files Directory', and 'Worksheet: Result Summary'. The 'Stress' sub-menu is expanded, showing options like 'Equivalent (von-Mises)', 'Maximum Principal', 'Middle Principal', 'Minimum Principal', 'Maximum Shear', 'Intensity', 'Normal', 'Shear', 'Vector Principal', 'Error', 'Membrane Stress', 'Bending Stress', 'Bottom Peak Stress (Beta)', and 'Top Peak Stress (Beta)'. The 'Normal' option is highlighted with a red box. Below the tree view, the 'Details of "Solution (B6)"' panel is visible, showing settings for 'Solution', 'Adaptive Mesh Refinement', and 'Information'.

Details of "Solution (B6)"	
<b>Solution</b>	
Number Of Cores to Use (Beta)	Solve Process Settings
<b>Adaptive Mesh Refinement</b>	
Max Refinement Loops	1,
Refinement Depth	2,
<b>Information</b>	
Status	Done
<input type="checkbox"/> MAPDL Elapsed Time	3, s
MAPDL Memory Used	112, MB
MAPDL Result File Size	2,0625 MB

# 7. NUMERICAL SOLUTION

## Safety Factor

The screenshot displays the ANSYS Workbench interface. On the left, the 'Static Structural (B5)' tree is visible, with 'Solution (B6)' selected. A context menu is open over 'Solution (B6)', showing options like 'Insert', 'Clear Generated Data', 'Rename', 'Group All Similar Children', 'Open Solver Files Directory', and 'Worksheet: Result Summary'. The 'Stress Tool' option is highlighted, and its sub-menu is open, showing 'Max Equivalent Stress' (highlighted with a red box), 'Max Shear Stress', 'Mohr-Coulomb Stress', and 'Max Tensile Stress'. Below the tree, the 'Details of Solution (B6)' panel is shown, containing sections for 'Solution', 'Adaptive Mesh Refinement', and 'Information'.

Details of "Solution (B6)"	
<b>Solution</b>	
Number Of Cores to Use (Beta)	Solve Process Settings
<b>Adaptive Mesh Refinement</b>	
Max Refinement Loops	1,
Refinement Depth	2,
<b>Information</b>	
Status	Done
<input type="checkbox"/> MAPDL Elapsed Time	3, s
MAPDL Memory Used	112, MB
MAPDL Result File Size	2,0625 MB

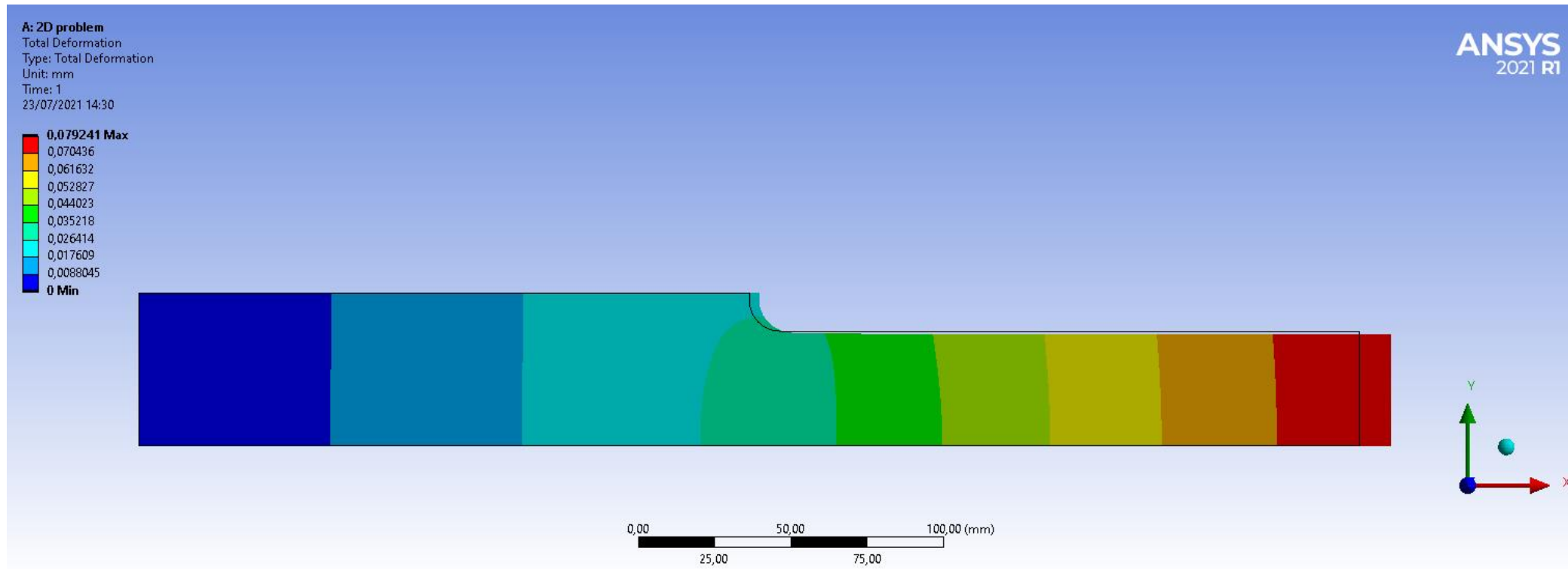


8

# Numerical Results

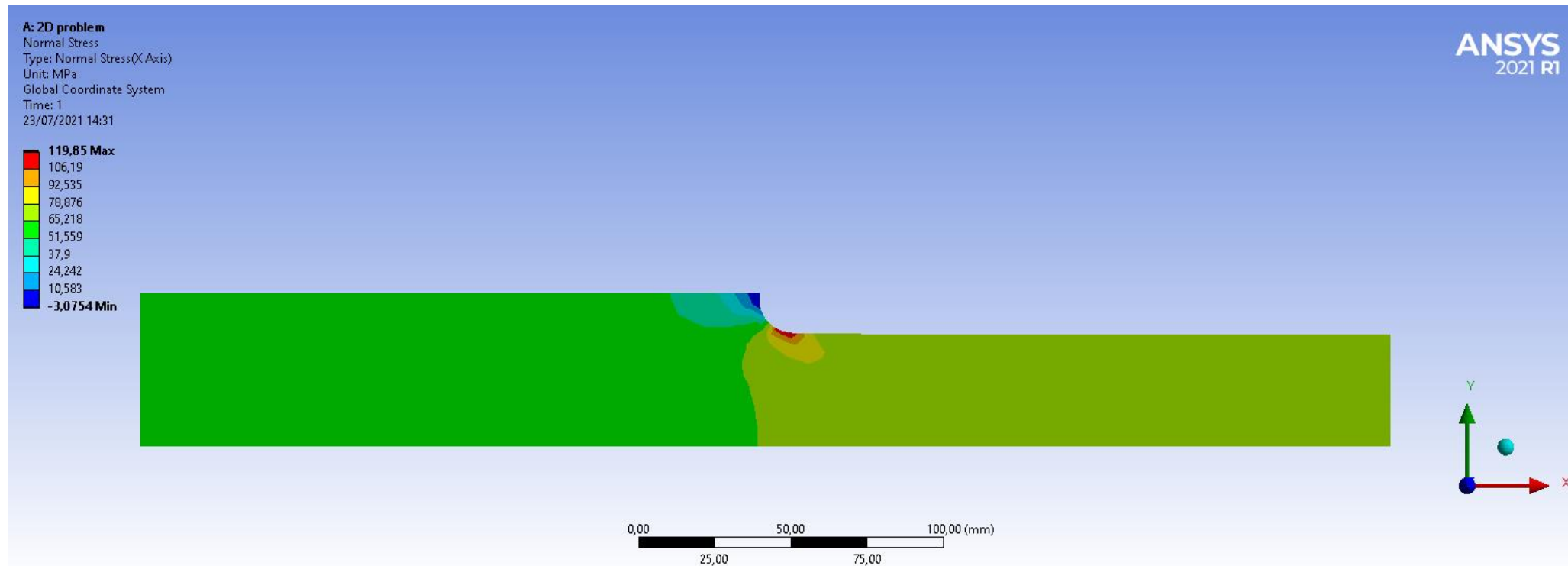
# 8. NUMERICAL RESULTS

## Displacement 2D – Coarse Mesh



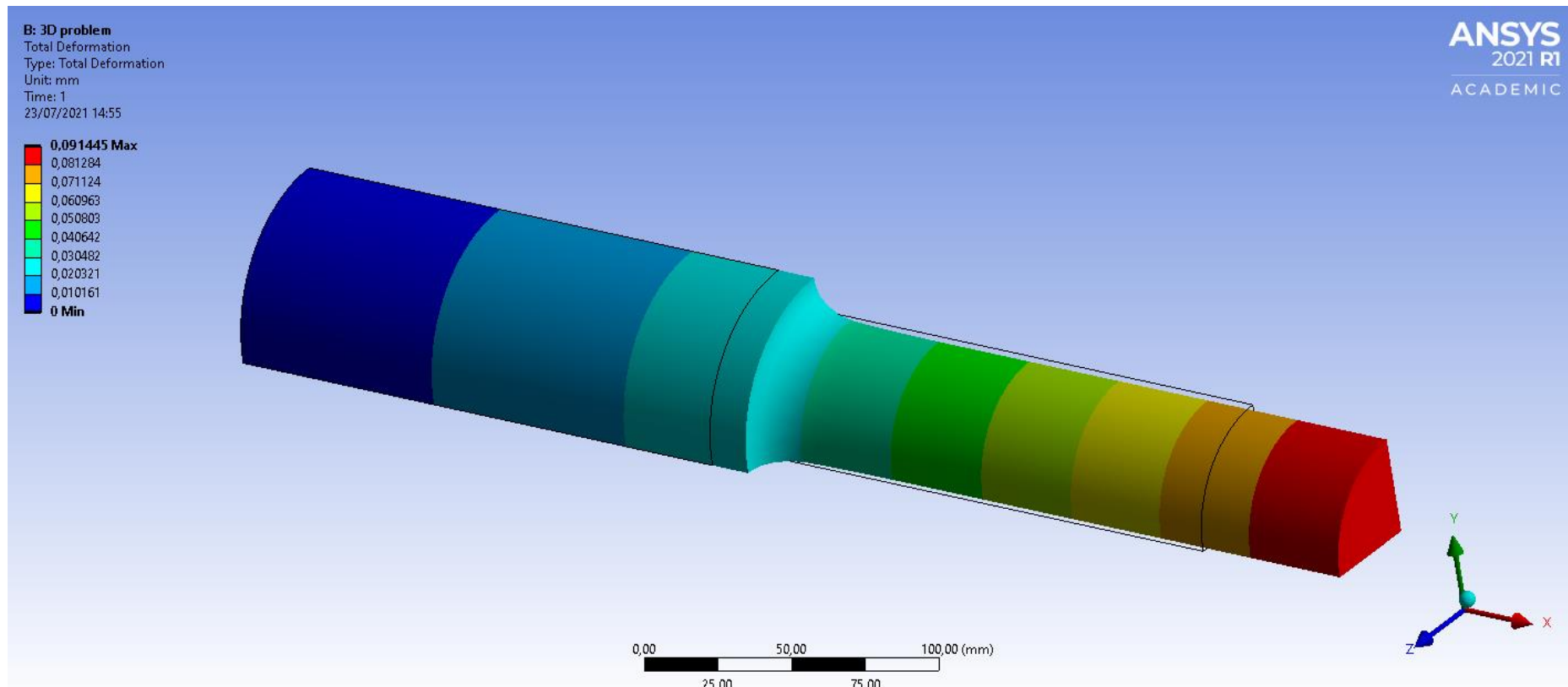
# 8. NUMERICAL RESULTS

## Normal Stress 2D – Coarse Mesh



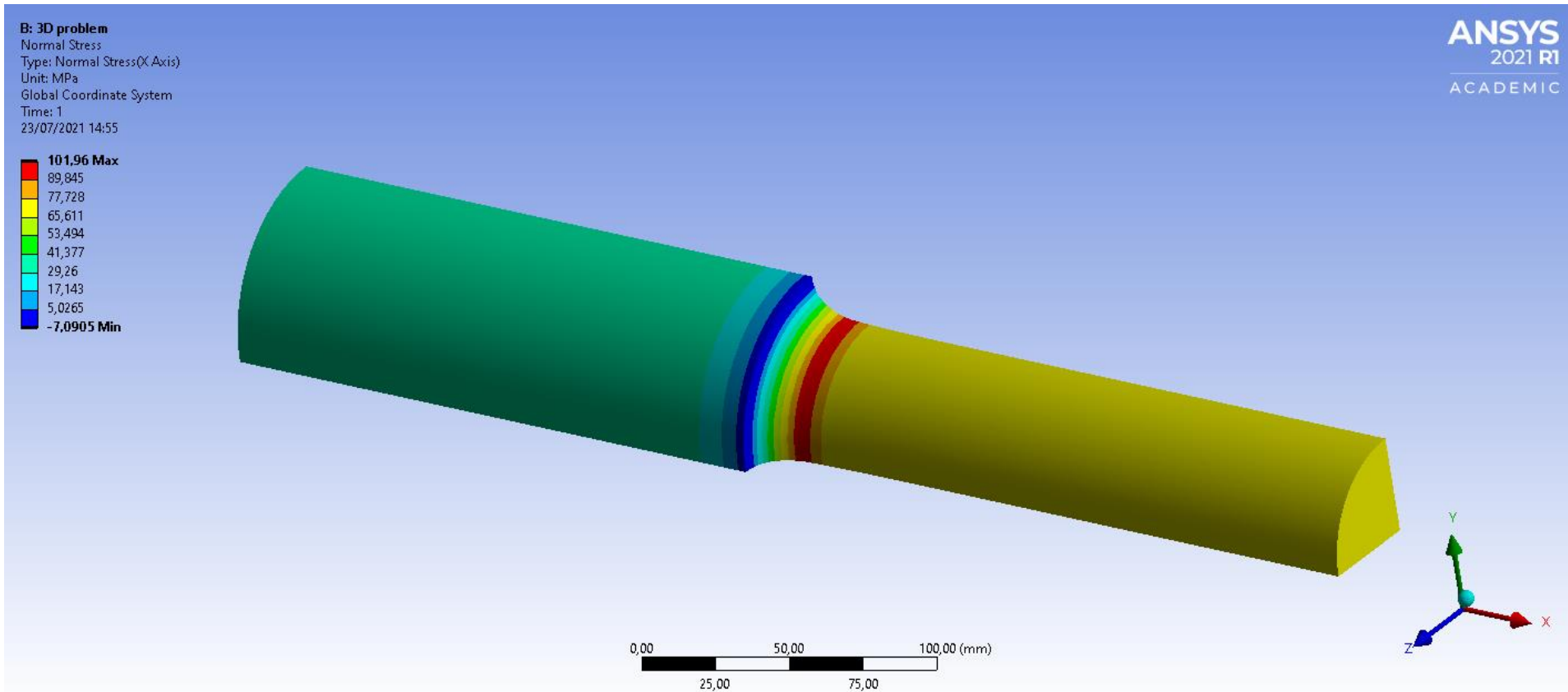
# 8. NUMERICAL RESULTS

## Displacement 3D – Coarse Mesh



# 8. NUMERICAL RESULTS

## Normal Stress 3D – Coarse Mesh



9

# Verification

# 9. VERIFICATION

## Results for $r = 10$ mm (Coarser mesh)

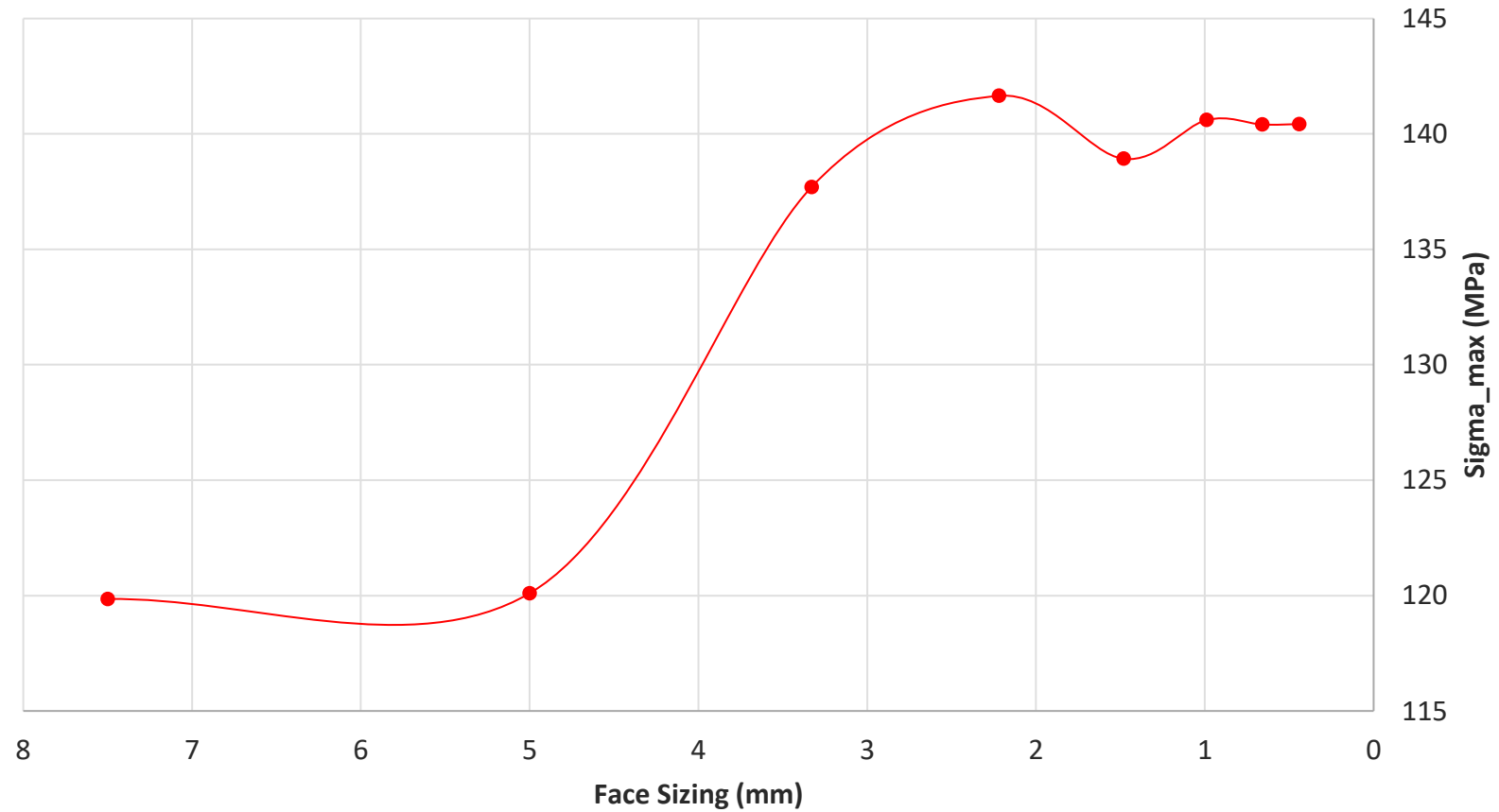
Empirical	2D problem	3D problem
114,401 MPa	119,85 MPa	101,96 MPa
-	4,76 %	10,87 %

## Results for $r = 10$ mm (Finer mesh)

Empirical	2D problem	3D problem
114,401 MPa	140,43 MPa	117,35 MPa
-	22,75 %	2,58 %

# 9. VERIFICATION

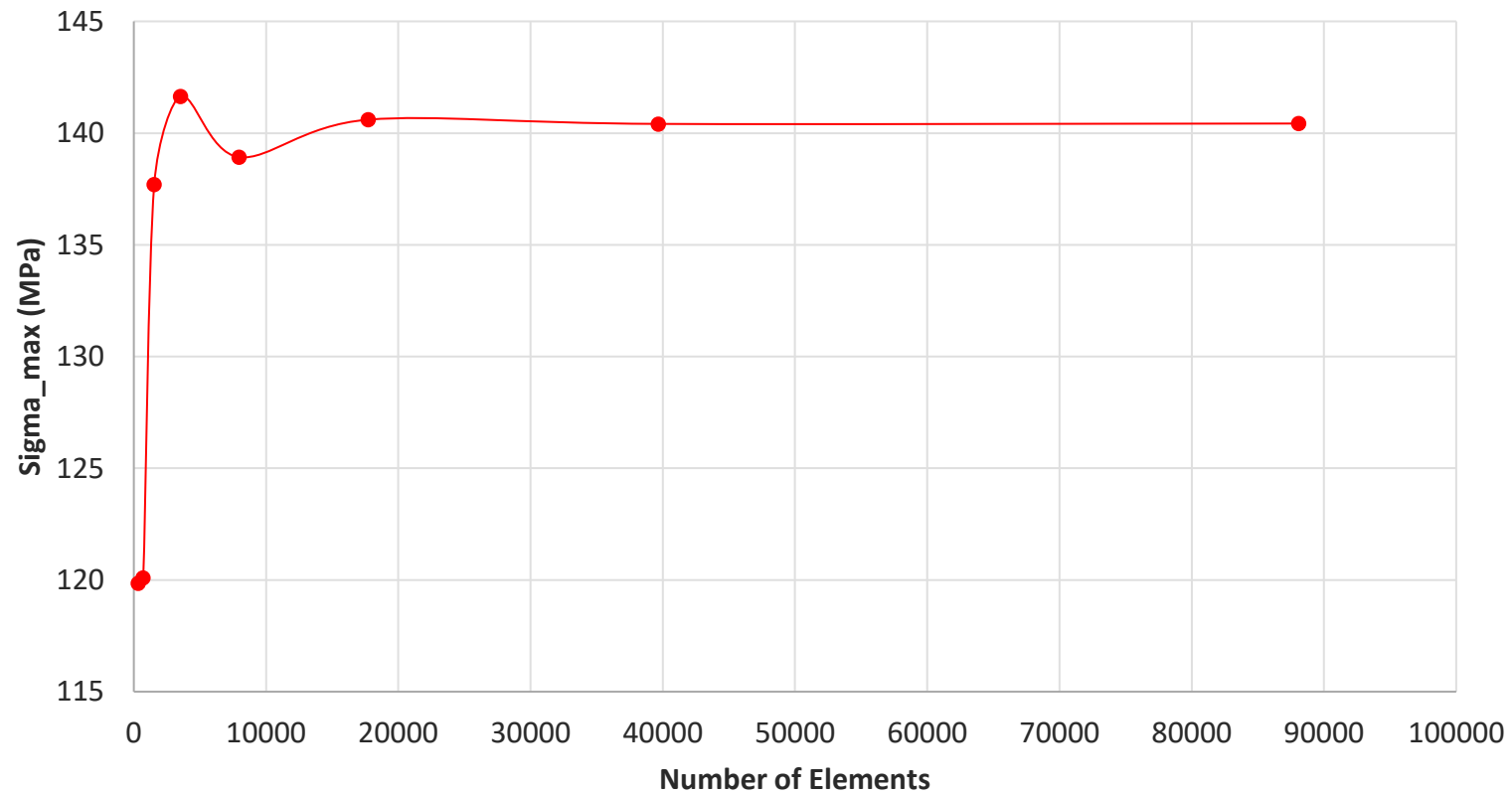
Mesh convergence - 2D problem





# 9. VERIFICATION

Mesh convergence - 2D problem





# Exercises

# EXERCICES

1. Based on the safety factor, find the minimum fillet radius.
2. Do a parametric analysis and plot the maximum normal stress versus the fillet radius.
3. Do a mesh convergence analysis using parametric analysis.
4. What happens if we change the material?
5. How can we reduce the maximum stress without changing either diameters and fillet radius?