Internship report

MT15-03M Evgeny Spirin
Scientific supervisor professor Ph.D. in Engineering Science Michail Golovin

fppt.com

CADFEM

CADFEM in D, A, CH

- 60 million euros of revenue
- 2,300 customers
- 12 locations
- 185 employees (worldwide >250)
- Family-run business

CADFEM and ANSYS partnership

- Since company's foundation
- Offering all ANSYS products
- Close technical collaboration
- CADFEM: Competence Center FEM
- ANSYS Germany: Competence Center
- CFD

fppt.com

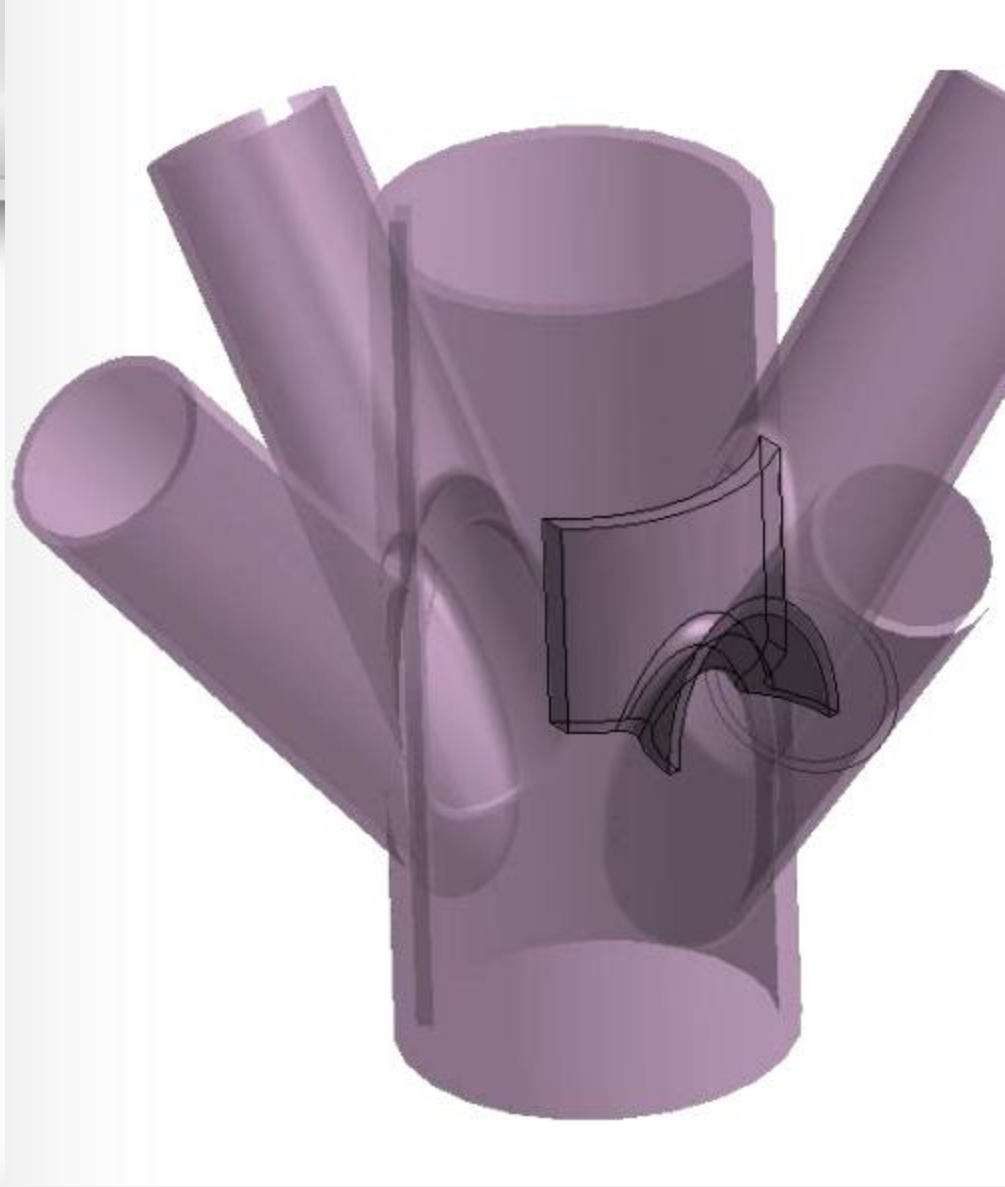
Internship Objectives

- To learn techniques for solution of engineering problems by means of ANSYS environment
- To learn technologies for solution of multiphysics problems
- To calculate a mesh for solving three-dimensional problem of the external flow
- To solve fluid dynamic problem in airfoil cavitations mode

fppt.com

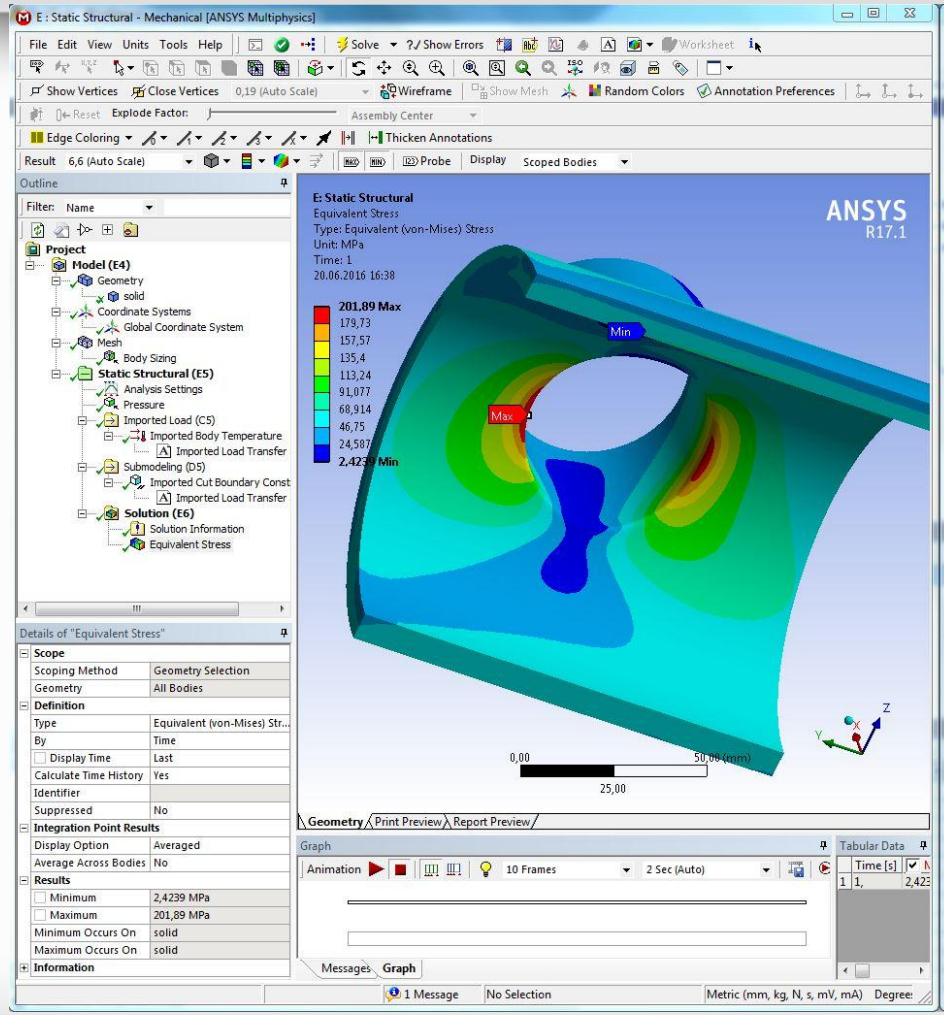
To obtain accurate stress in a local region, submodeling separates local analysis from the global model. This allows mesh refinement in a region that might not be possible on the full model without exceeding size limits.

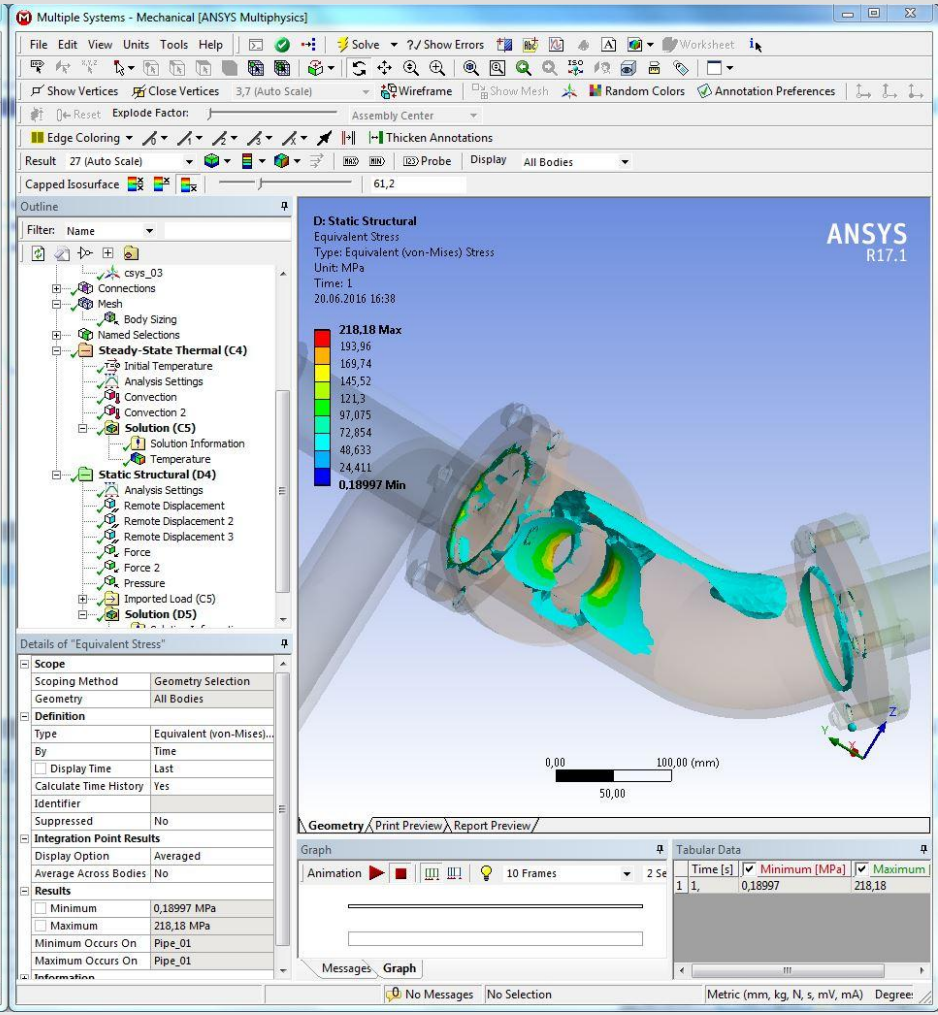
(Reduction technique)



fppt.com

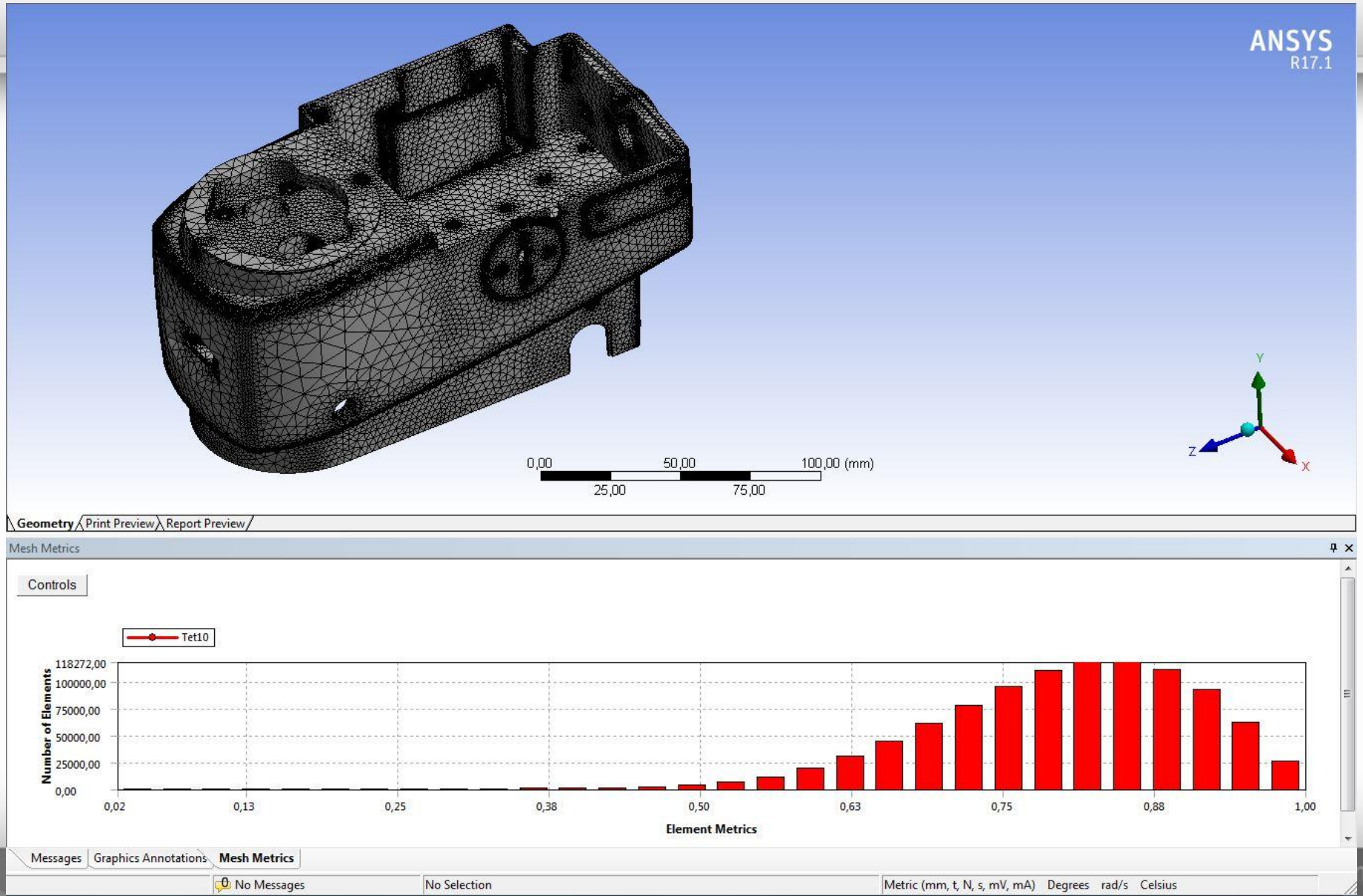
Submodeling example



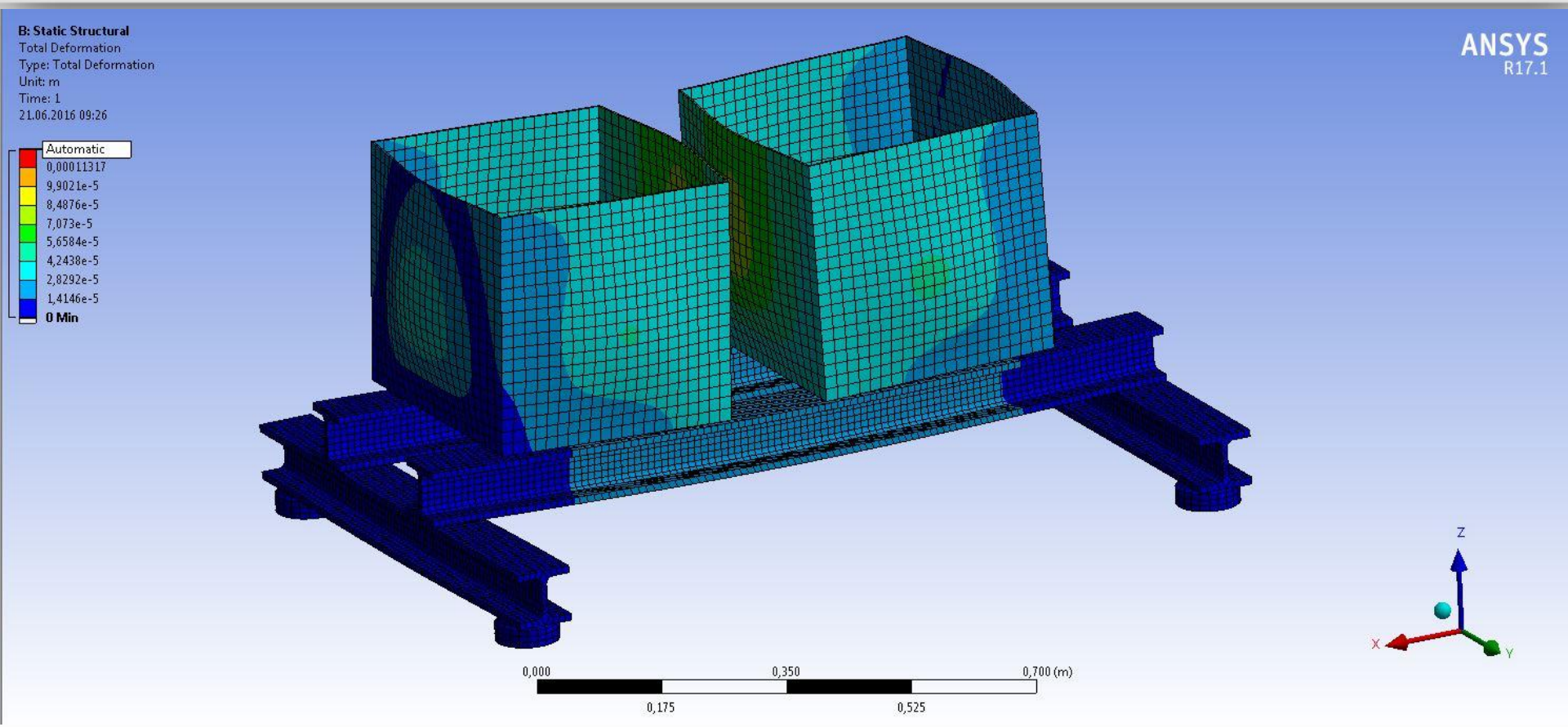


fppt.com

Meshing

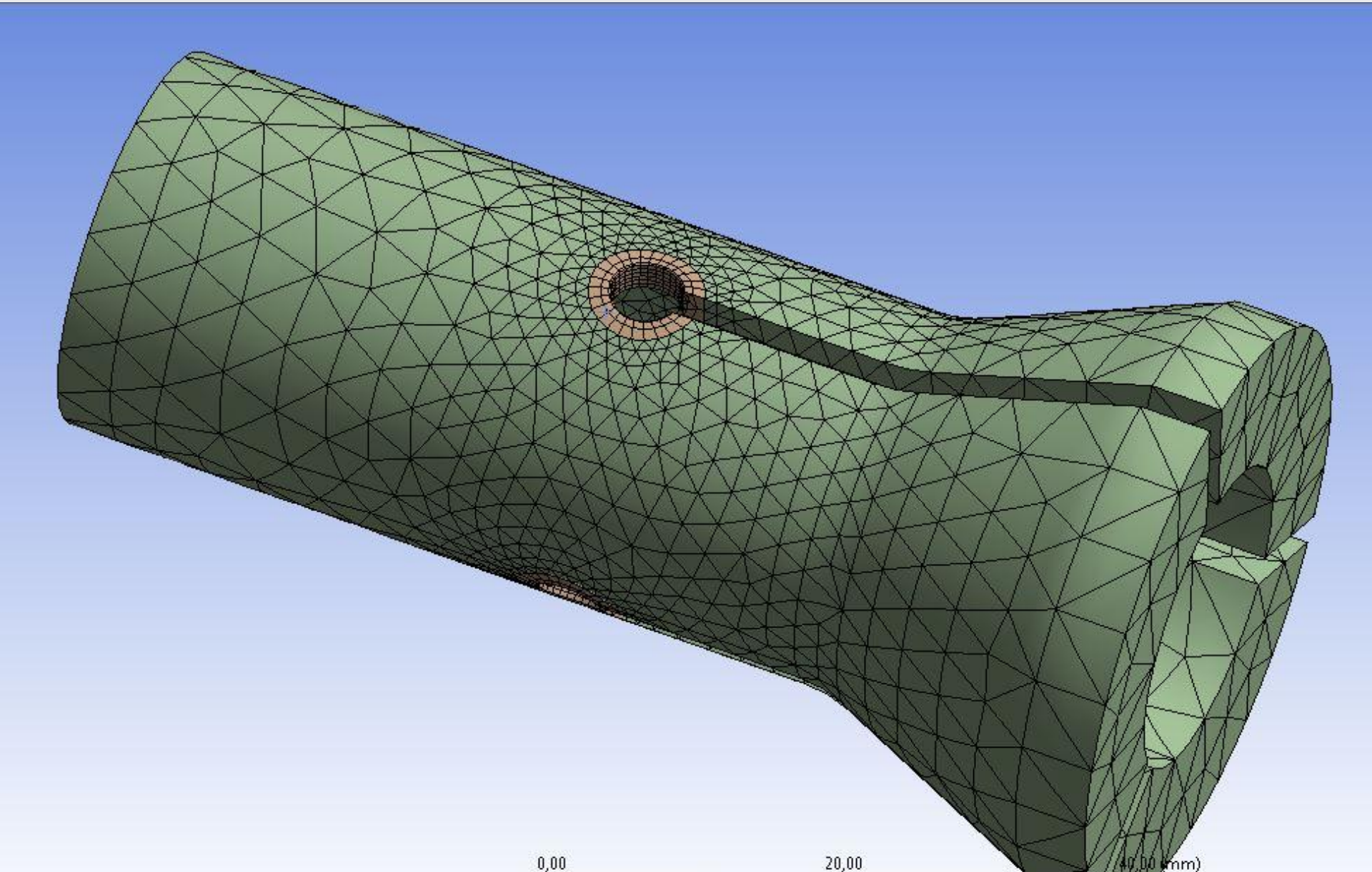

ANSYS
R17.1
0,00
50,00
100,00 (mm)
25,00
75,00
Y
Z
X
Geometry
Print Preview
Report Preview
Mesh Metrics
Controls
Tet10
Number of Elements
118272,00
100000,00
75000,00
50000,00
25000,00
0,00
0,02
0,13
0,25
0,38
0,50
0,63
0,75
0,88
1,00
Element Metrics
Messages
Graphics Annotations
Mesh Metrics
No Messages
No Selection
Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius

Water tanks on frame

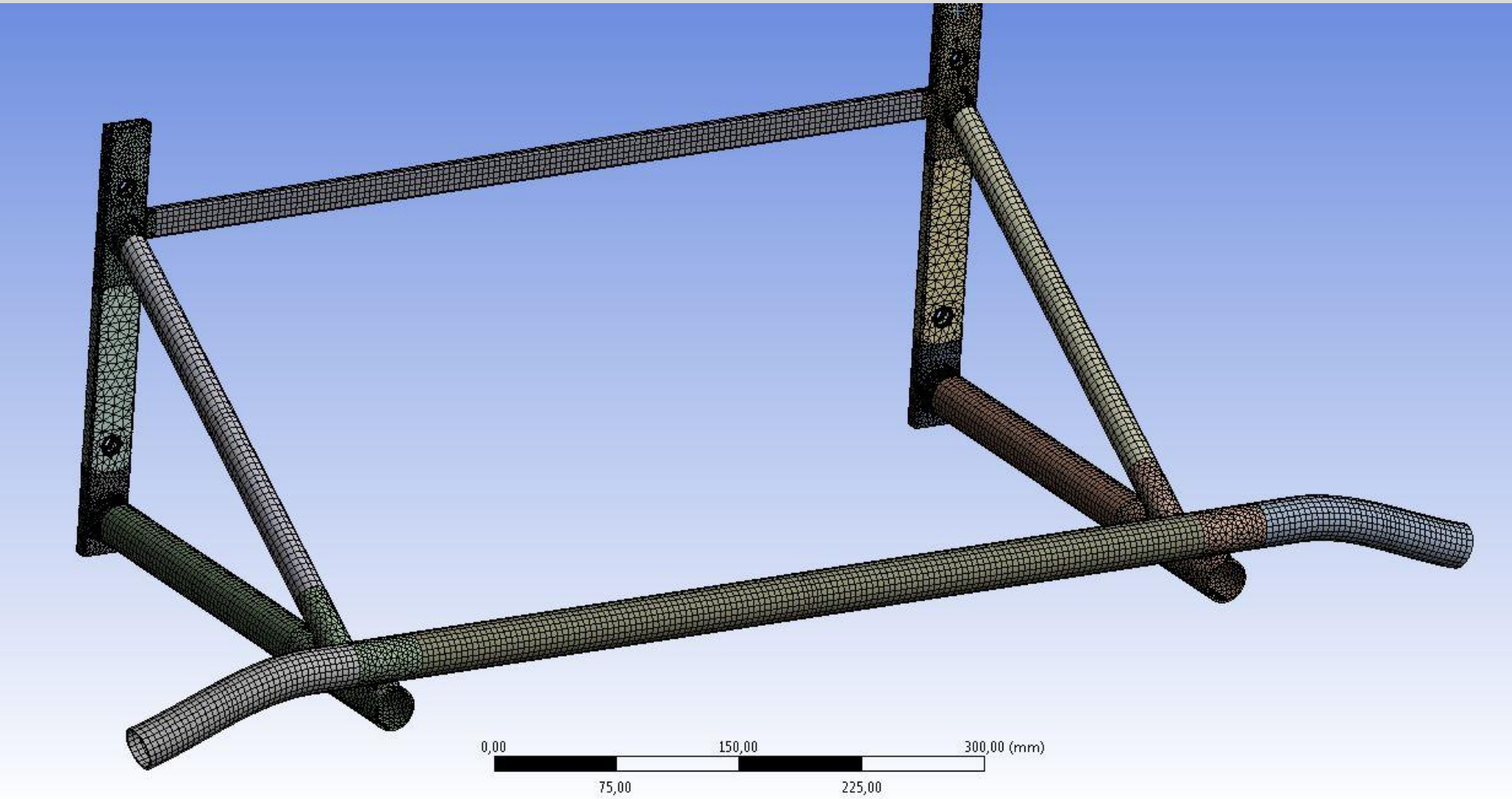


fppt.com

Share topology

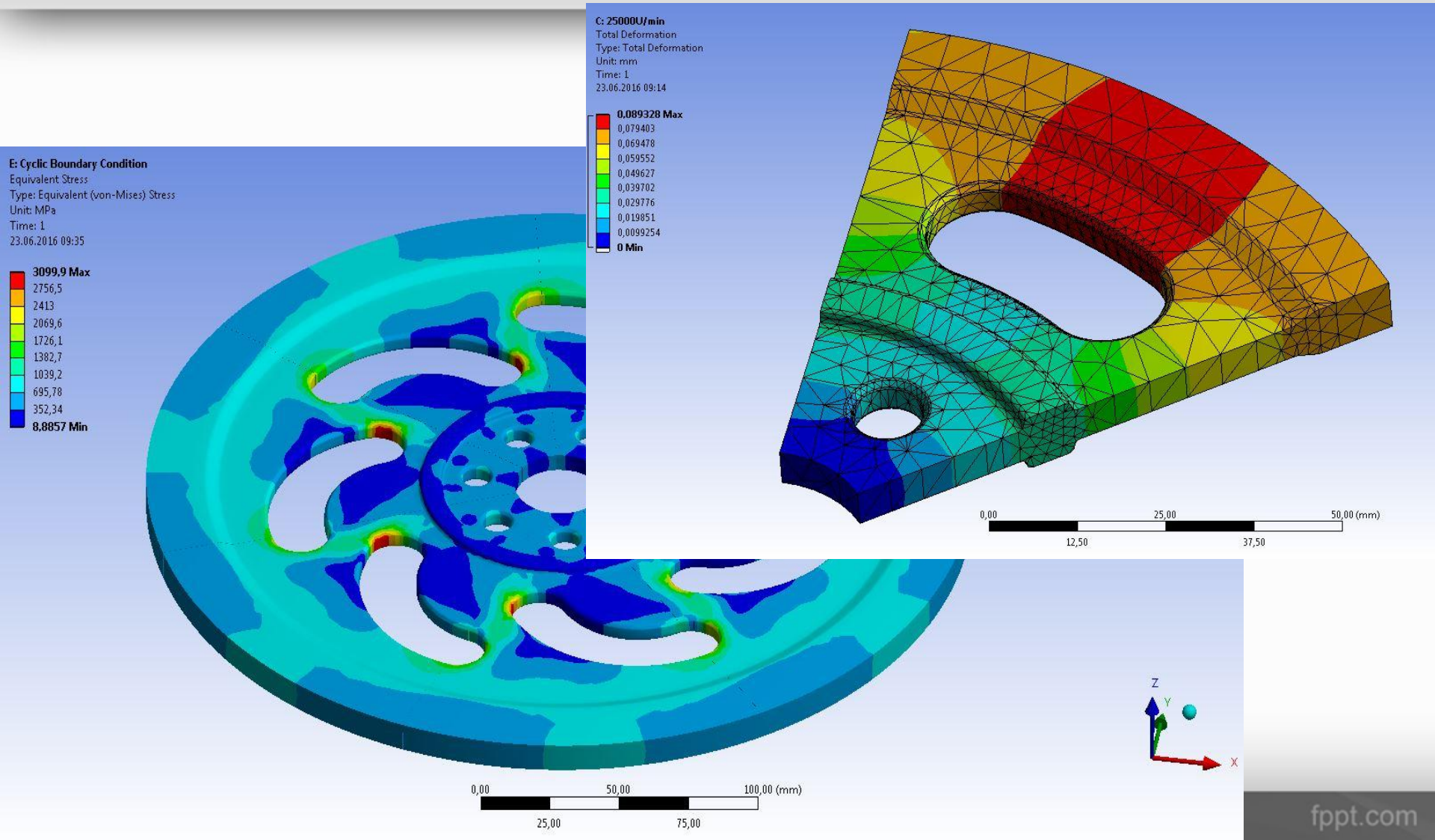


Advanced meshing



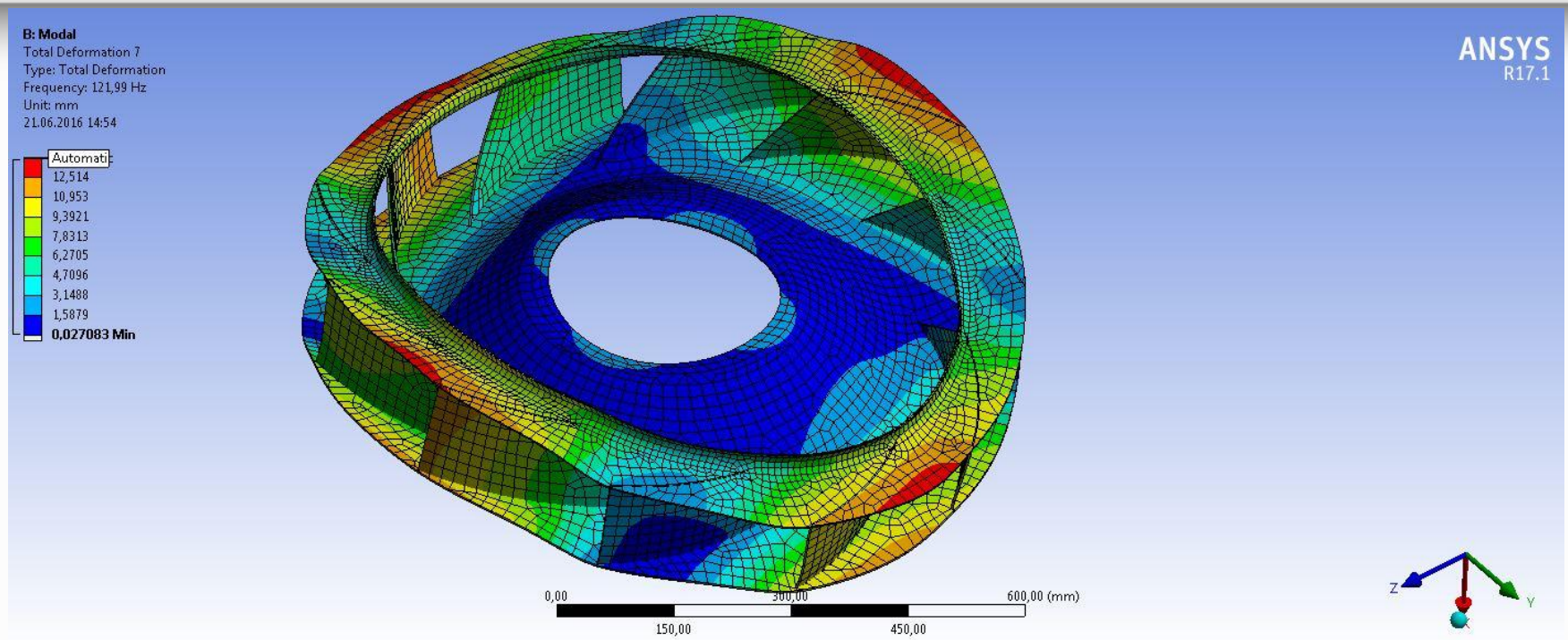
fppt.com

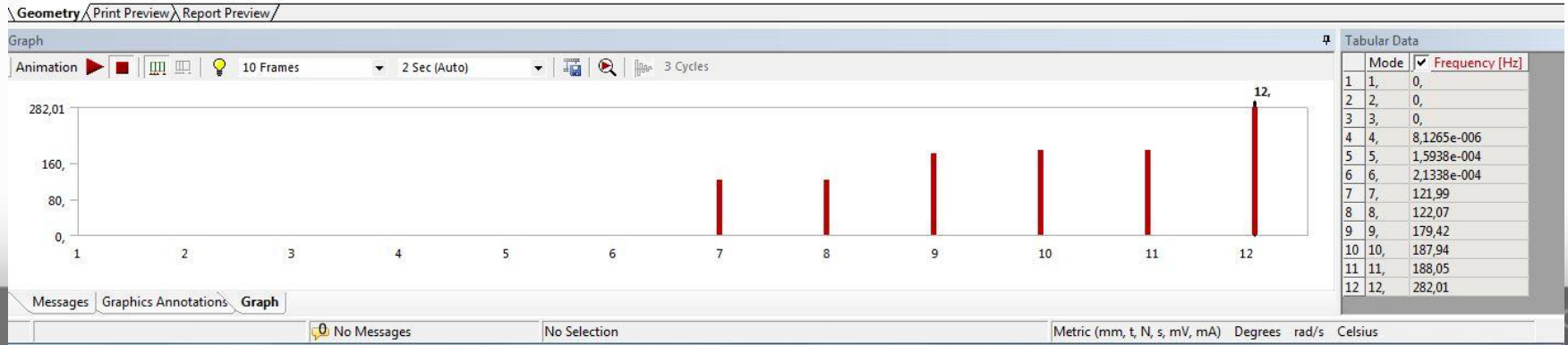
Circular symmetry



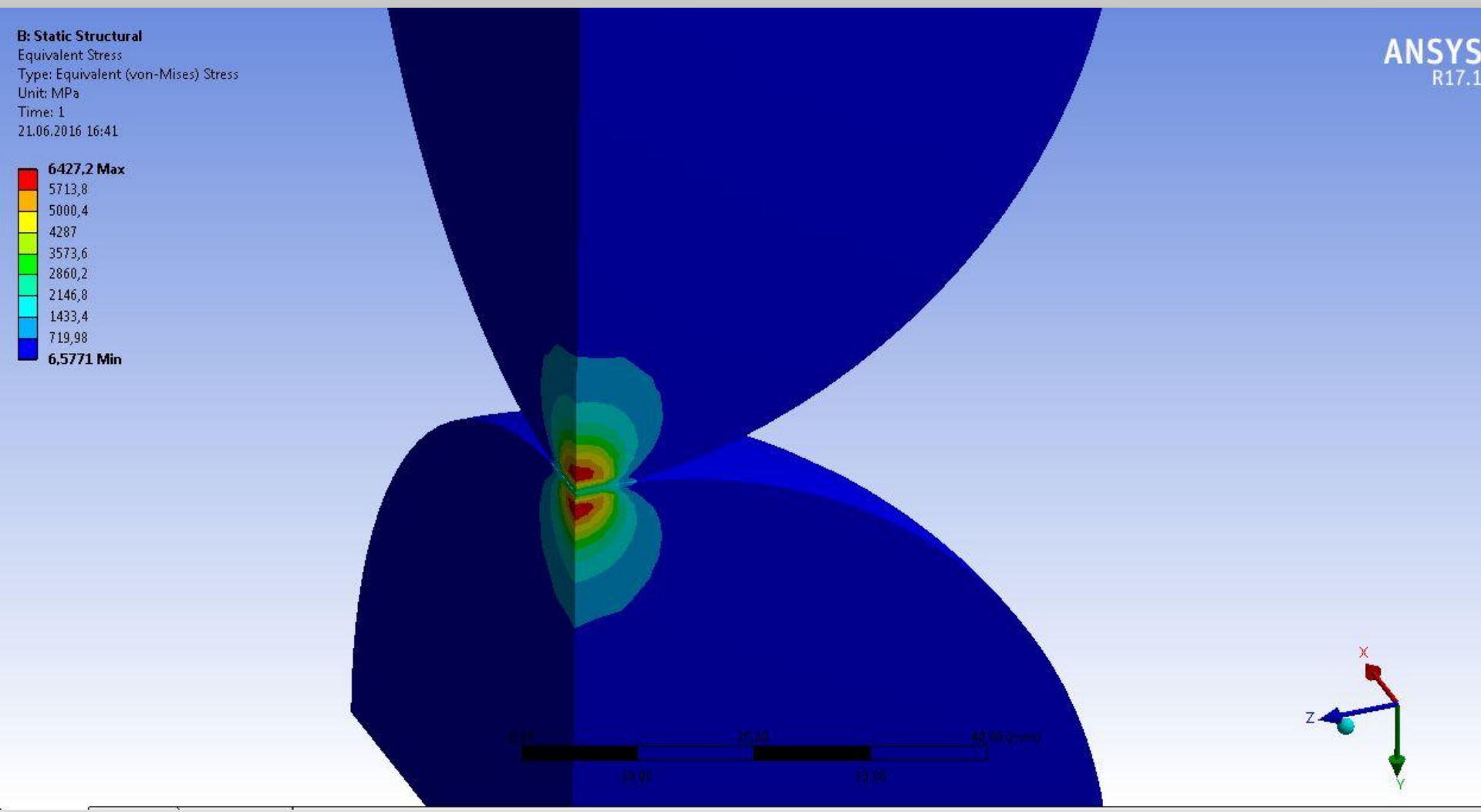
fppt.com

Modal analysis

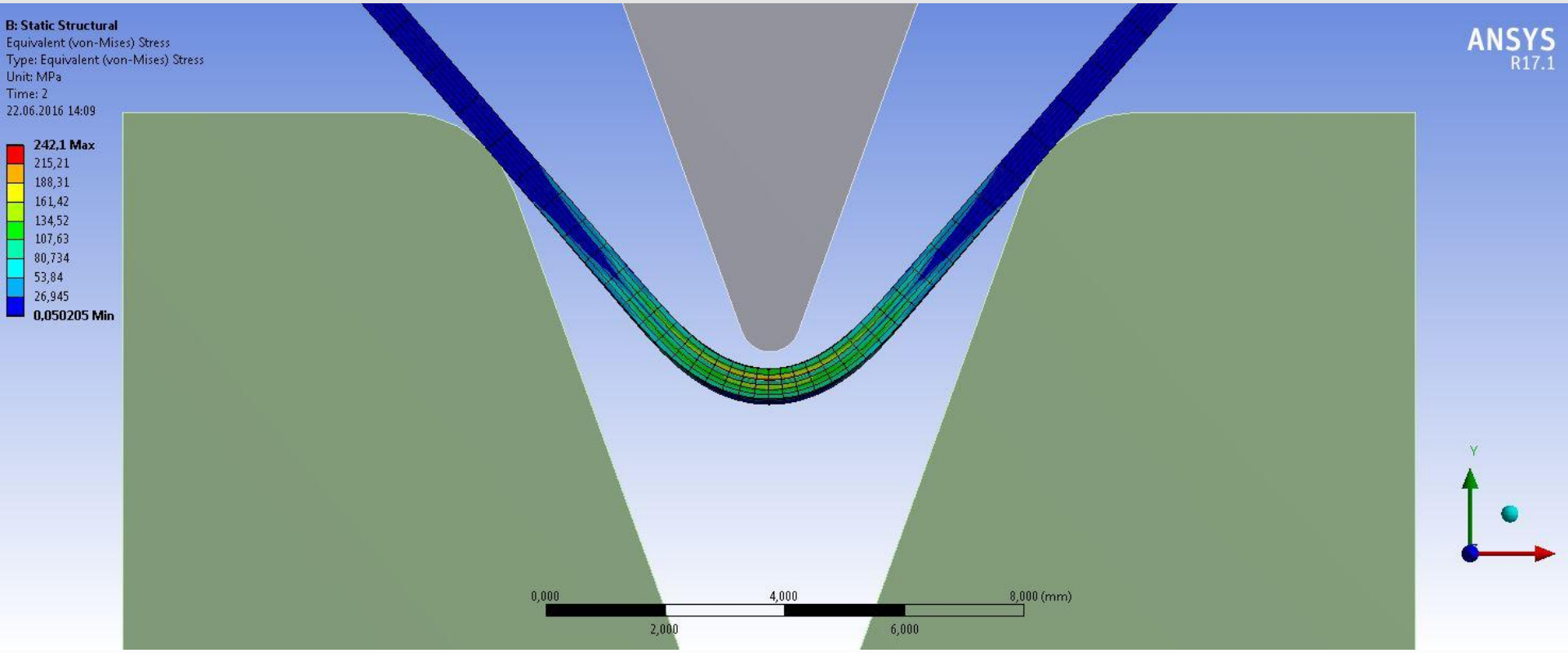




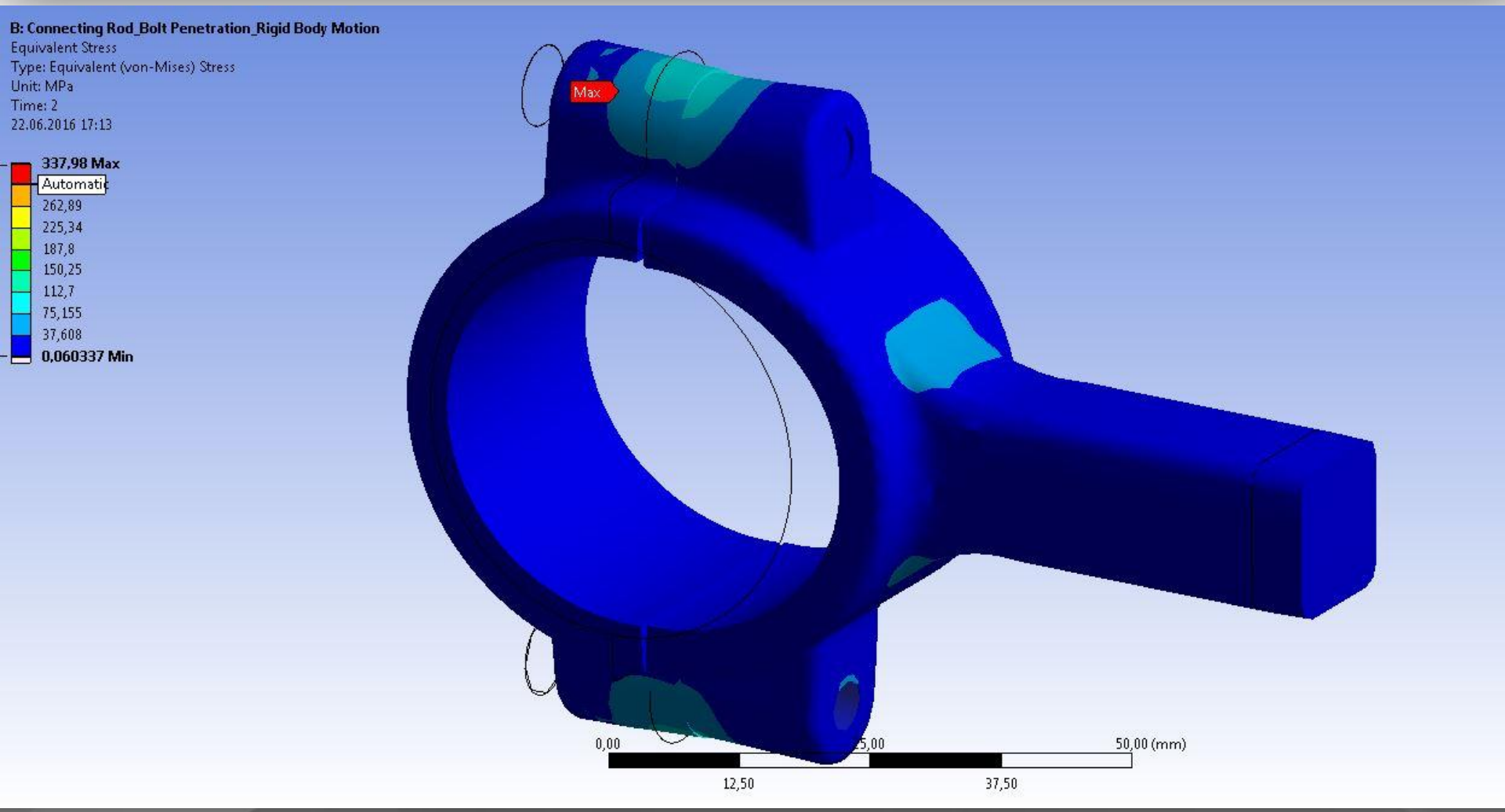
Simulation of the surface contact



fppt.com

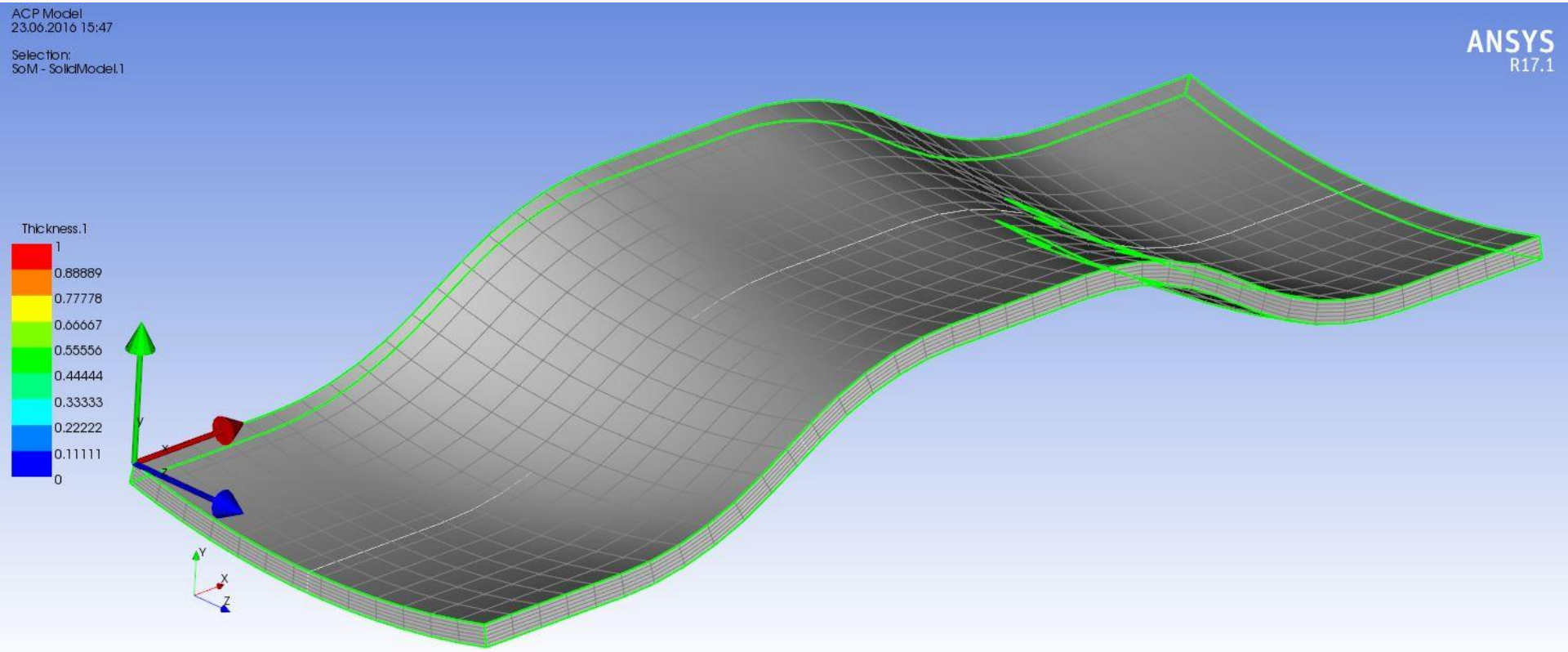

B: Static Structural
Equivalent (von-Mises) Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 2
22.06.2016 14:09
242,1 Max
215,21
188,31
161,42
134,52
107,63
80,734
53,84
26,945
0,050205 Min
ANSYS
R17.1
Y
0,000
4,000
8,000 (mm)
2,000
6,000

fppt.com


B: Connecting Rod_Bolt Penetration_Rigid Body Motion
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 2
22.06.2016 17:13
337,98 Max
Automatic
262,89
225,34
187,8
150,25
112,7
75,155
37,608
0,060337 Min
Max
0,00
25,00
50,00 (mm)
12,50
37,50

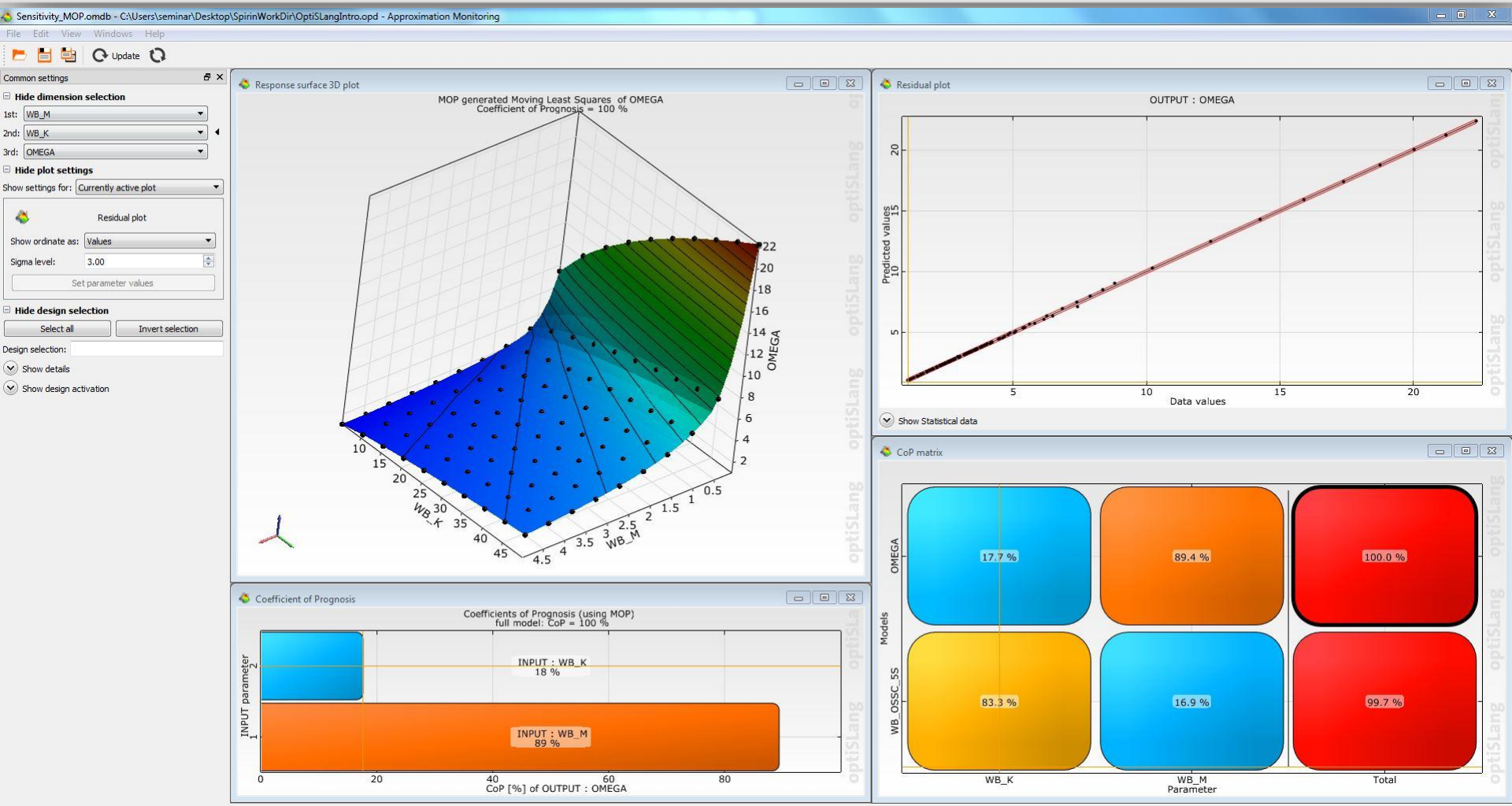
fppt.com

Composite material



fppt.com

Optimization tool



fppt.com

Description of topology optimization in general

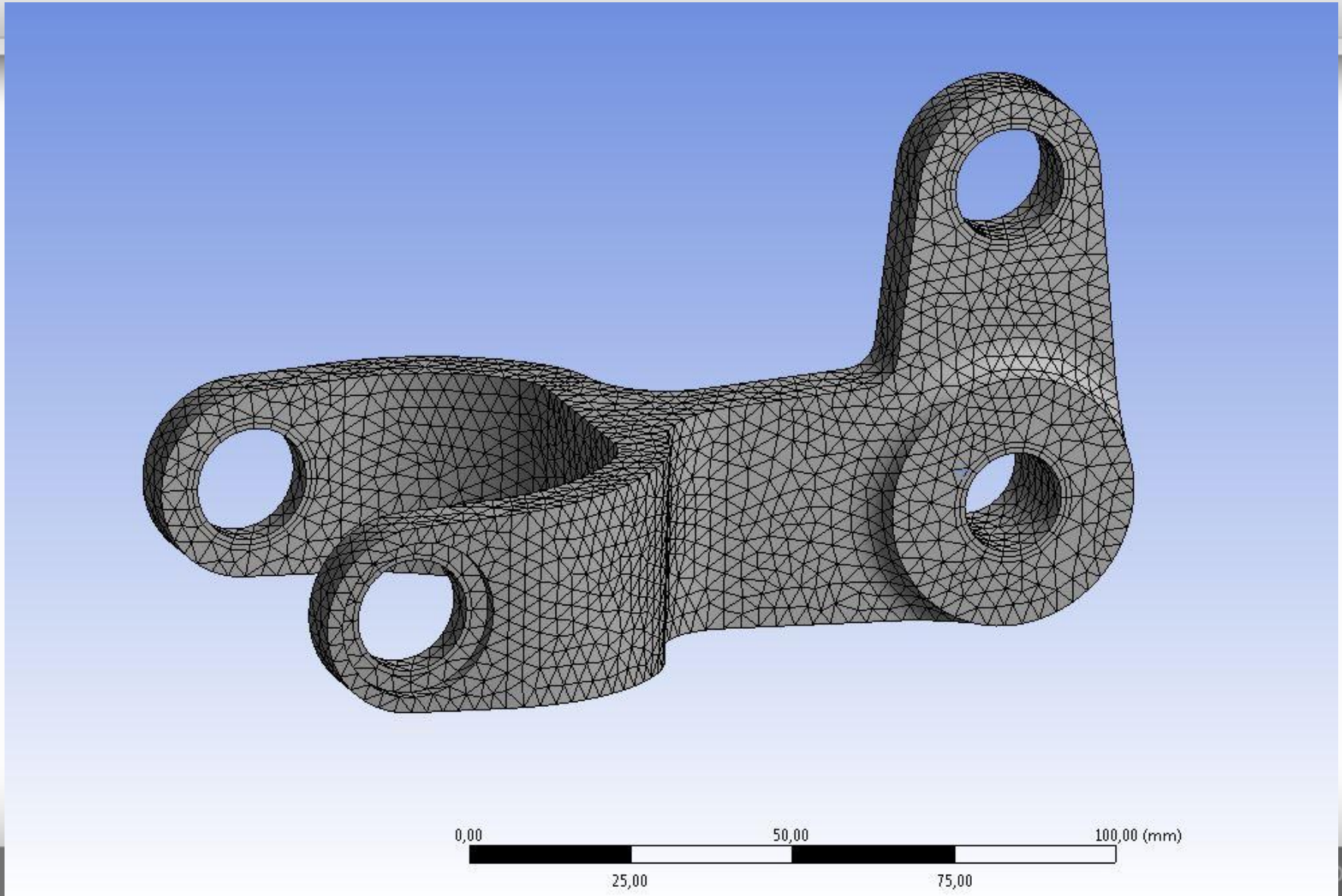
- What is the objective of a Topology Optimization?
  → Get a material distribution which provides for a given design space and for a single or multiple load case scenario an optimal part stiffness

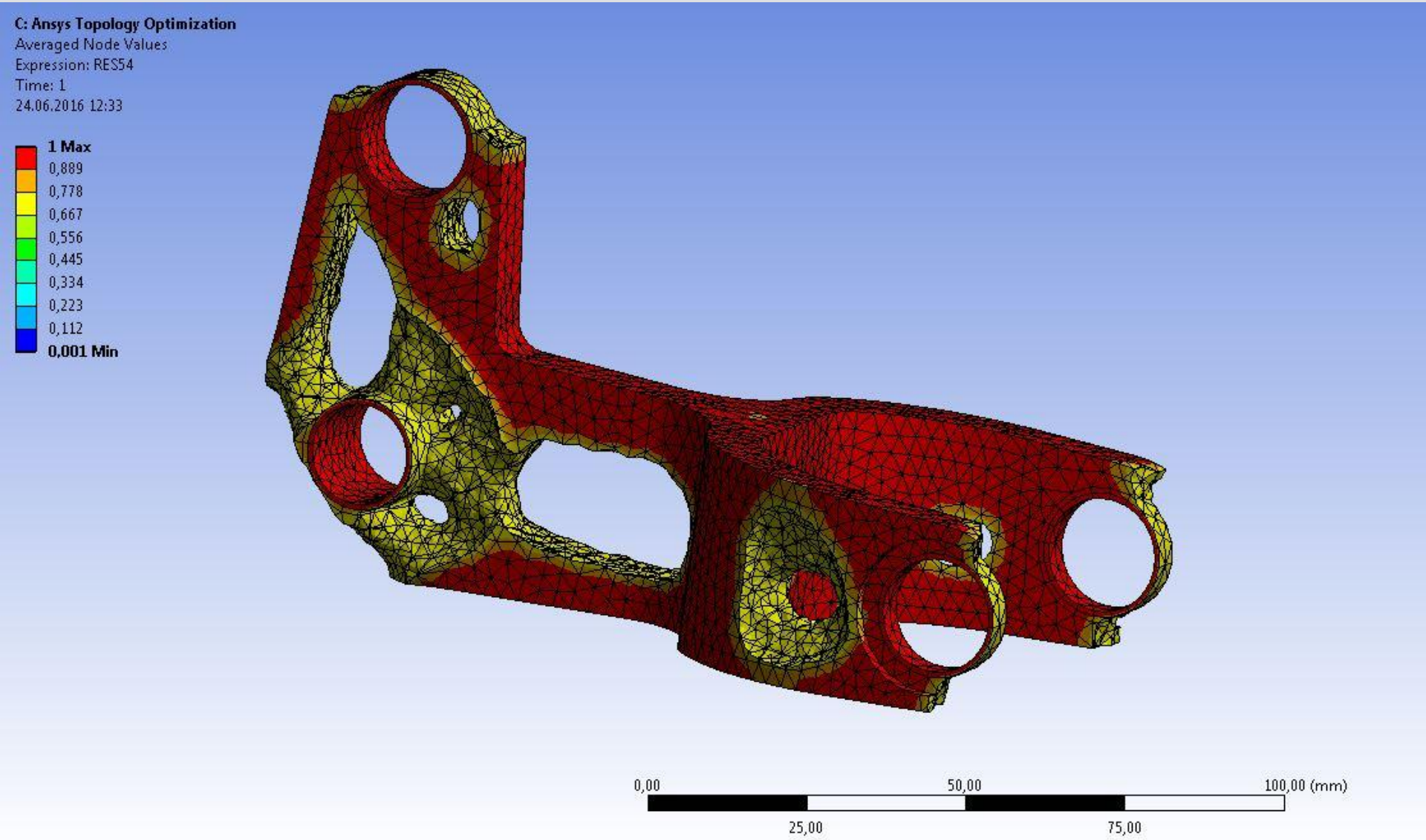
- The most common objective function in topology optimization is the energy of the elastic compliance. Minimizing the compliance is equivalent to maximizing the global stiffness

- As constraint (state variable) usually the volume of the part is defined. The design variable is the pseudo density, which is assigned to each element. Value „1“ describes that the element is active, „0“ means inactive

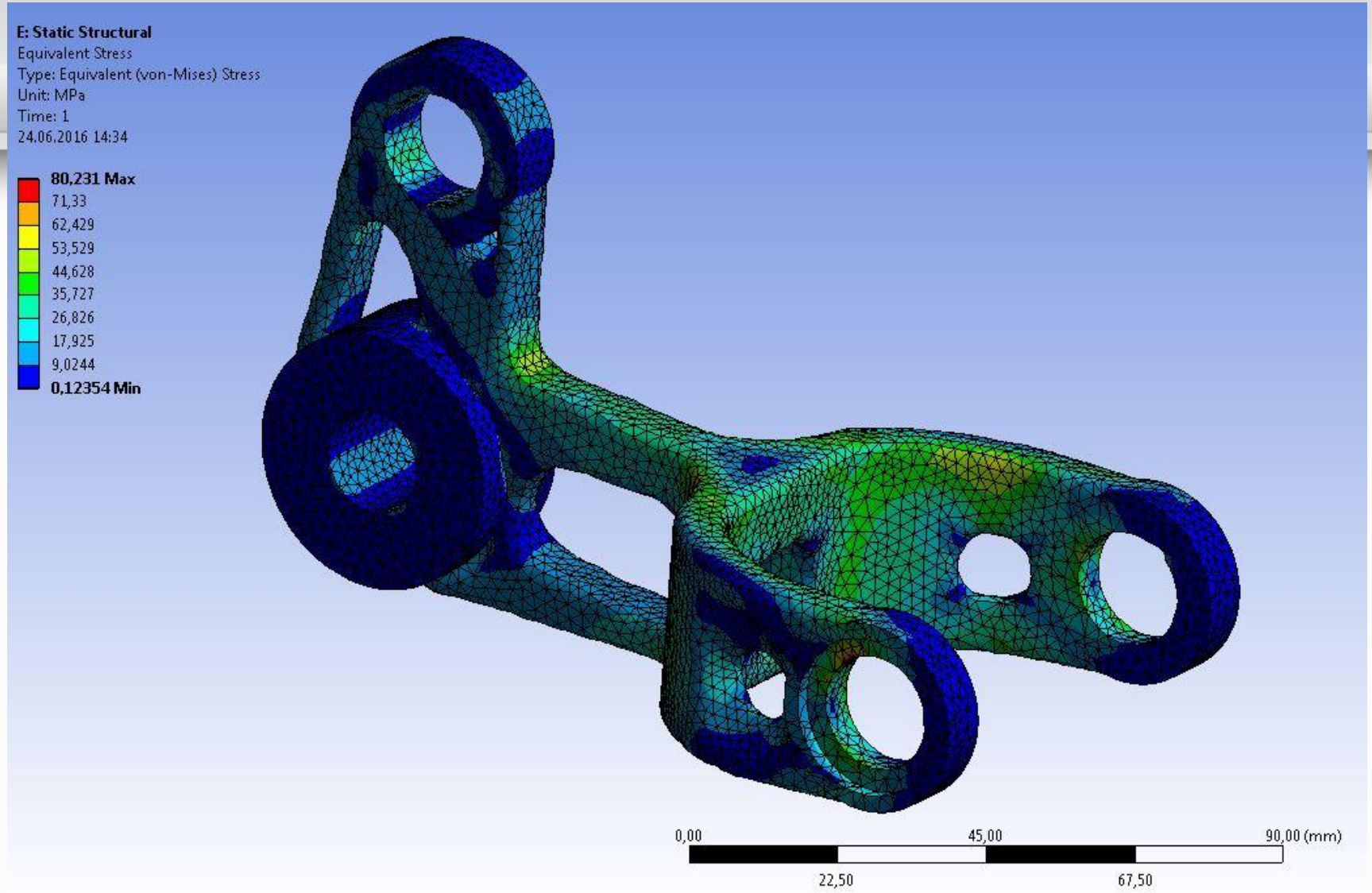
fppt.com

Topology optimization




C: Ansys Topology Optimization
Averaged Node Values
Expression: RES54
Time: 1
24.06.2016 12:33
1 Max
0,889
0,778
0,667
0,556
0,445
0,334
0,223
0,112
0,001 Min
0,00
50,00
100,00 (mm)
25,00
75,00

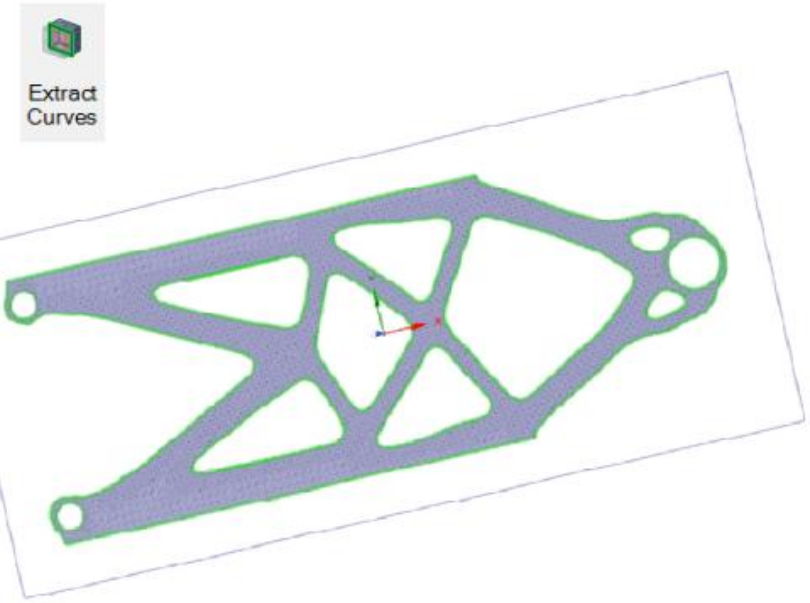
fppt.com

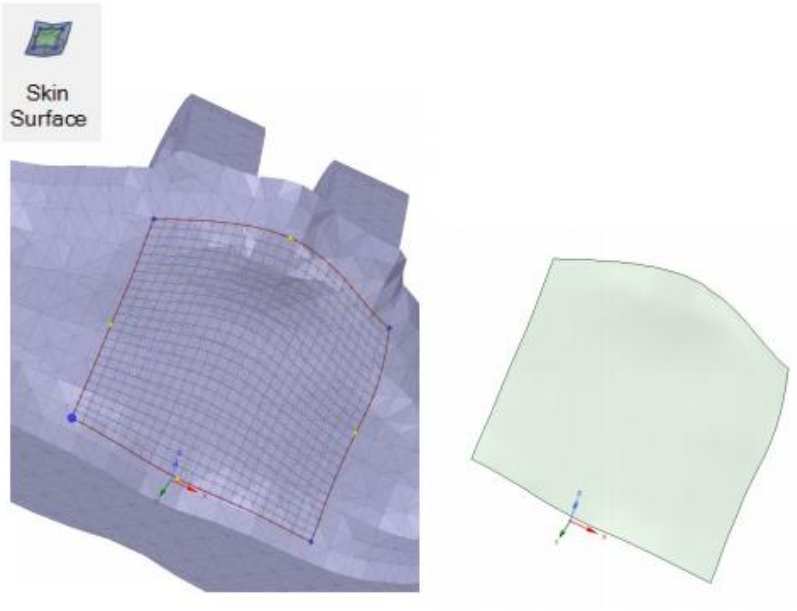

E: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
24.06.2016 14:34
80,231 Max
71,33
62,429
53,529
44,628
35,727
26,826
17,925
9,0244
0,12354 Min
0,00
45,00
90,00 (mm)
22,50
67,50

fppt.com

Reverse Engineering

- Extract Curves
- Skin Surface
- Default geometries (Cylinder, Sphere)





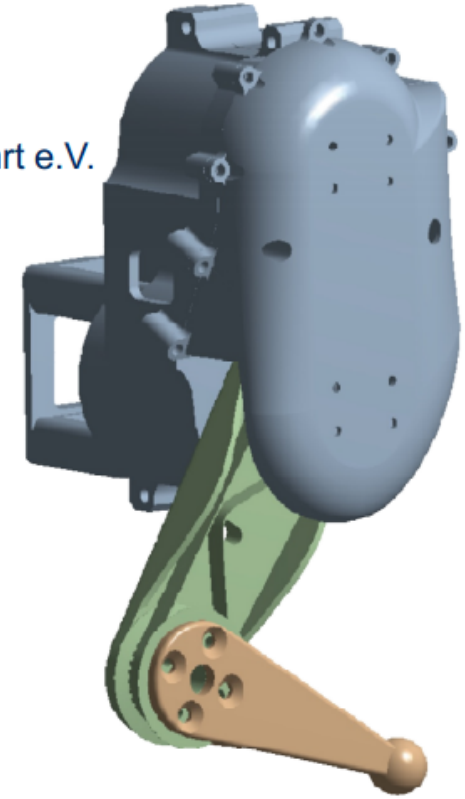
fppt.com

DLR jumping robot leg

- Demonstration of the ANSYS Topology Optimization, V17.0

- Model: Jumping robot leg

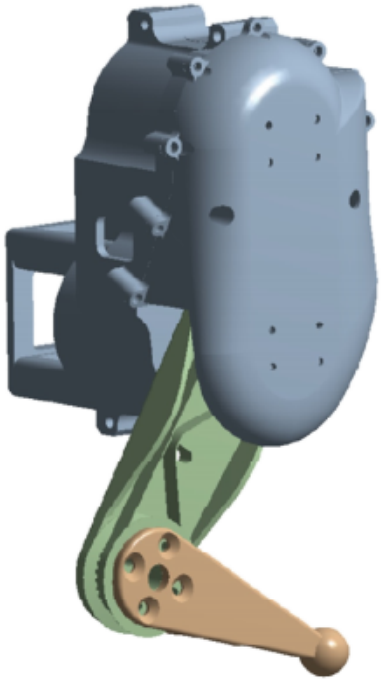
Source: Deutsches Zentrum für Luft- und Raumfahrt e.V.
in der Helmholtz-Gemeinschaft
Institut für Robotik und Mechatronik
Contact person: Werner Friedl



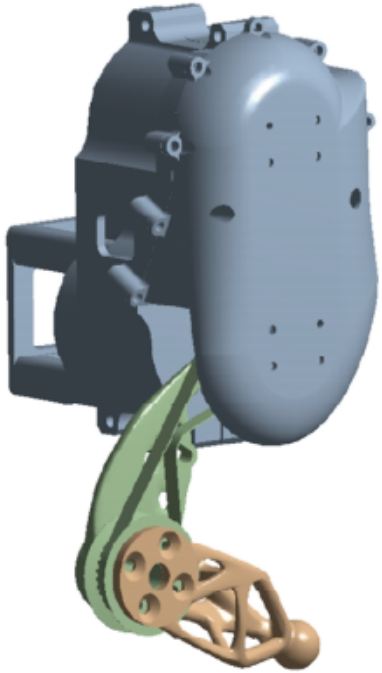
- Software:
  ANSYS Mechanical
  ANSYS SpaceClaim Direct Modeler

fppt.com

Comparision

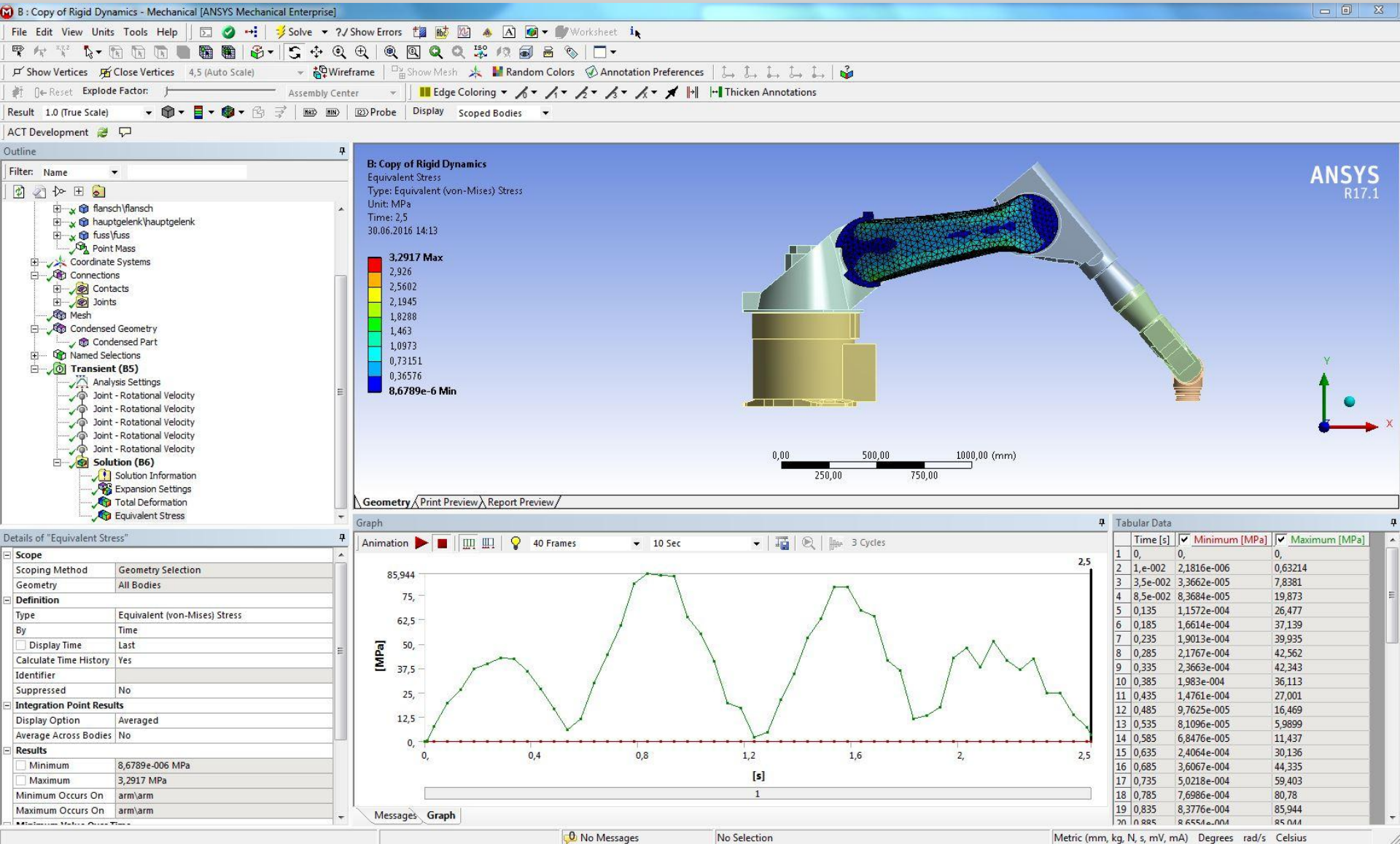


- Stress stays the same
- Deformation stays the same
- Mass reduced by 40%



fppt.com

High Performance Calculation



The main contributions

- Meshing
- Boundary conditions
- Contact
- Optimization
- HPC

fppt.com