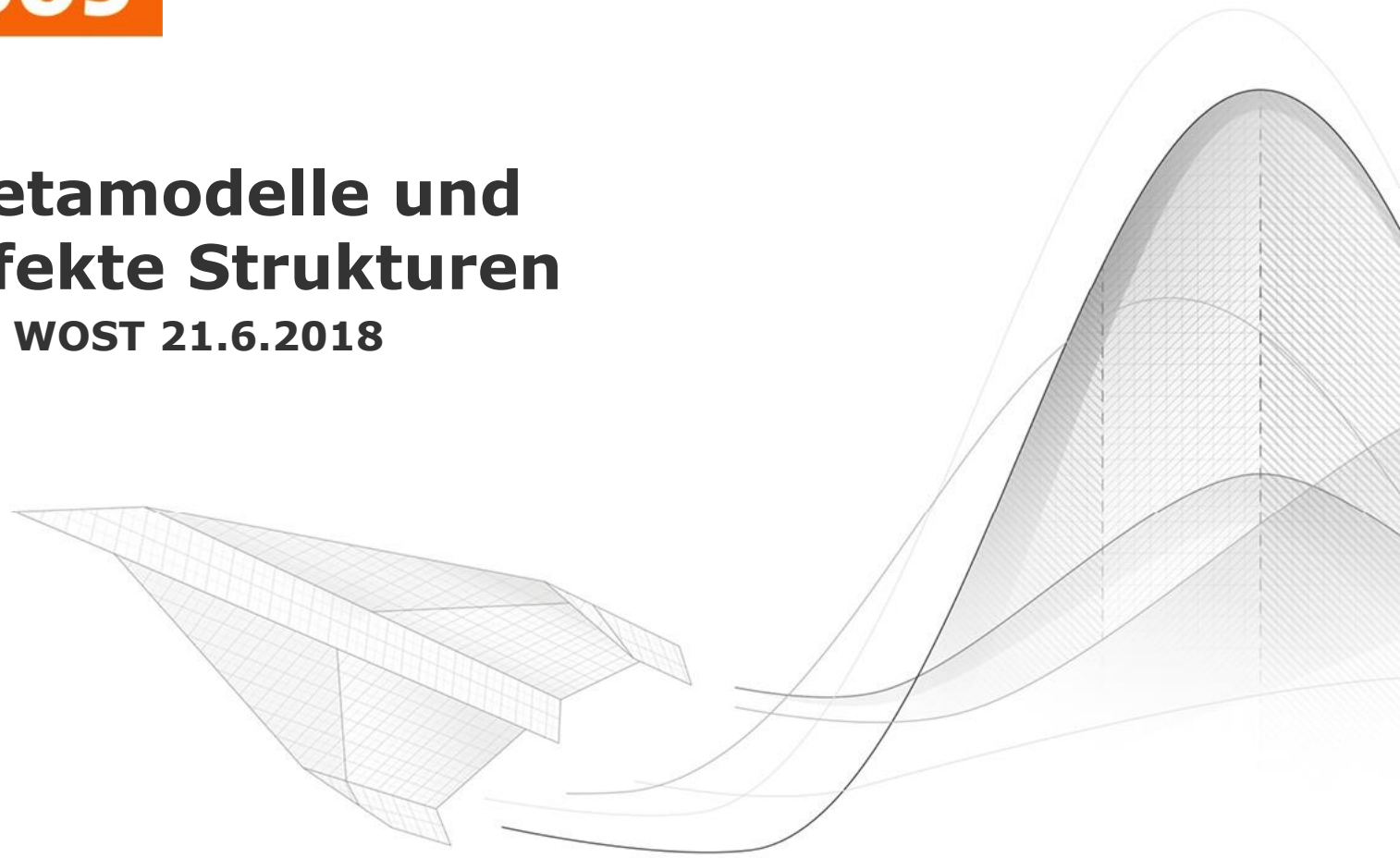
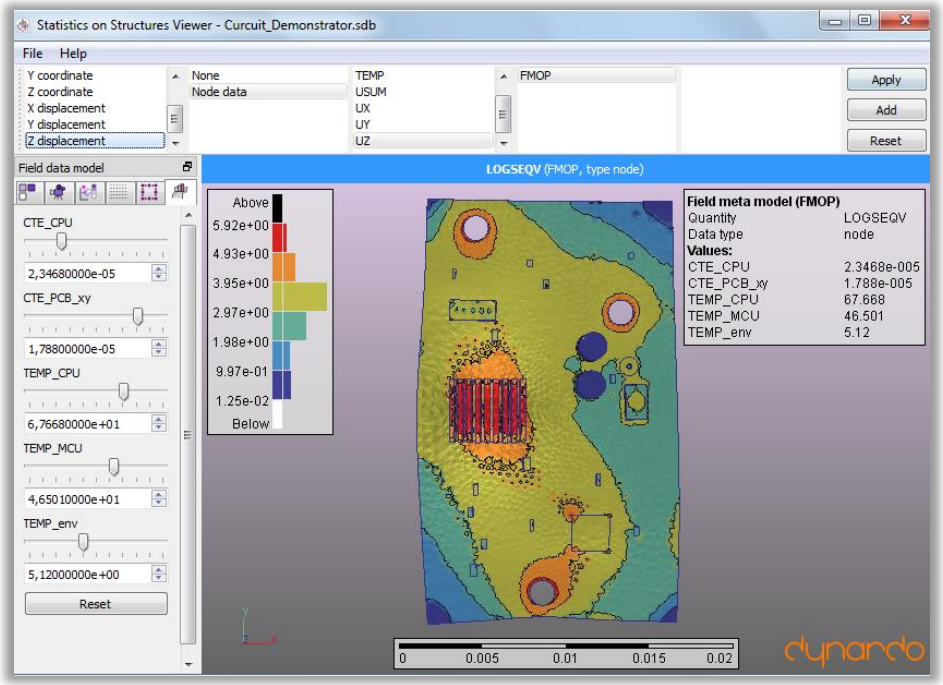
Teil 2: Feldmetamodelle und Imperfekte Strukturen

Workshop WOST 21.6.2018



Field meta models:
Statistics on Structures

- Graphical user interface for
  - Loading and visualizing FEM meshes (solver independent)
  - Analysis of measured (or simulated) data in 3D
  - Parameterize variations (e.g. geometric imperfections)
    - Empirical Random fields
    - Synthetic random fields
    - Free-form fields
  - Find nonlinear correlations through field meta models
    - FMOP

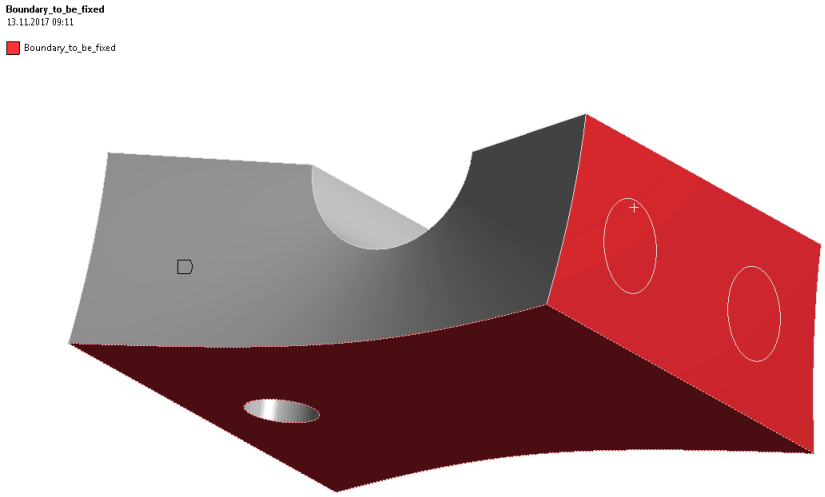


SoS for ANSYS Mechanical

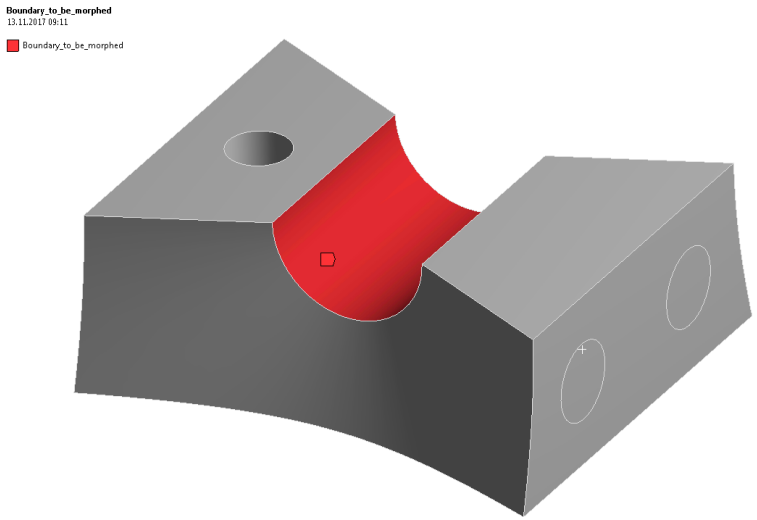
- Plugin in ANSYS Mechanical
  - Create random fields for geometric imperfections
  - Create free-form variation fields for geometric imperfections
  - Export Mechanical result data and mesh to SoS for FMOP
- Toolbar

ANSYS Mechanical

- Preparation of model: Distinguish 3 types of boundary:
  - Fixed support (not changed)
  - Boundary subject to loading (directly changed by SoS / used for parameterization)
  - Free boundary (used to interpolate between the other two)
- Create Named selections:
  - Fixed boundary

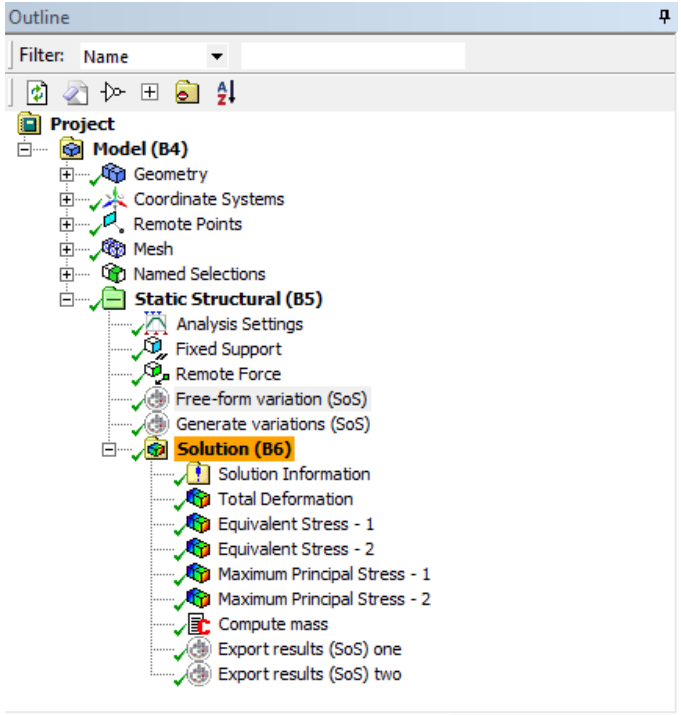


Parameterized boundary



ANSYS Mechanical
1. Define random field model

- Chose “Create” – “Synthetic random field model”
  - Used to create random variations !
  - Statistics: mean, stddev, correlation length

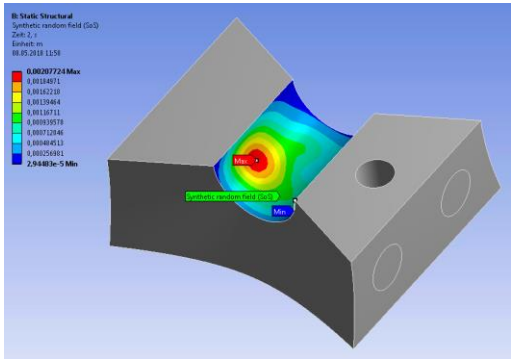


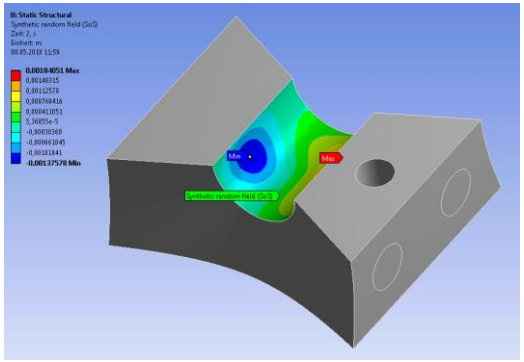
Details von "Synthetic random field (SoS)"

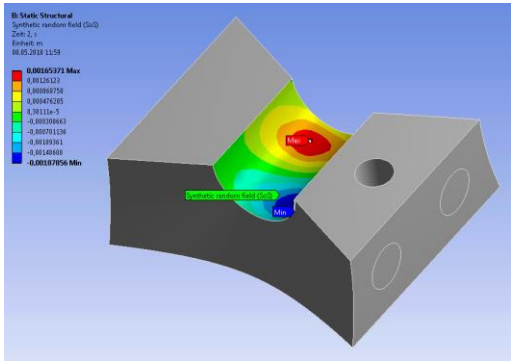
Boundary to be parameterized	
Scoping Method	Named Selection
Named Selection	Boundary_to_be_morphed
Fixed boundary	
Scoping Method	Named Selection
Named Selection	Boundary_to_be_fixed
Mesh part to be exported to SoS	
Scoping Method	Named Selection
Named Selection	NS_Solid
Definition	
Desired variability (%)	90
Maximum number of parameters	10
Correlation length (eg. ~20% of total length)	0,005 [m]
Standard deviation of geometric variation	0,005 [m]
Mean geometric variation	0 [m]
Visualization	
Visible variation shape index (0=mean)	0
Advanced options	
Use mesh stabilization	True
Test on mesh distortion	False
Move nodes along	Boundary normal
Solver options	
Number of CPUs used by SoS (0=all)	0
Internal directory	SoS_SynthRF

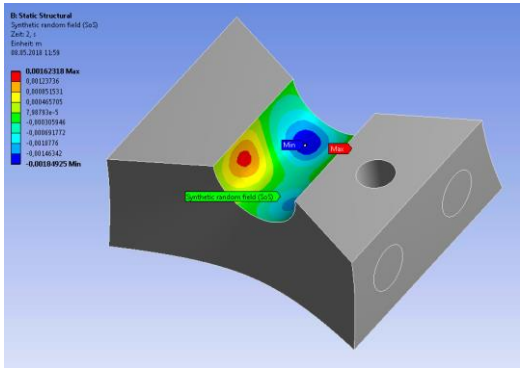
ANSYS Mechanical
Synthetic random field model

- Visual feedback on variation shapes:
- Shapes 1,2,3,4: wave like shaped variation patterns



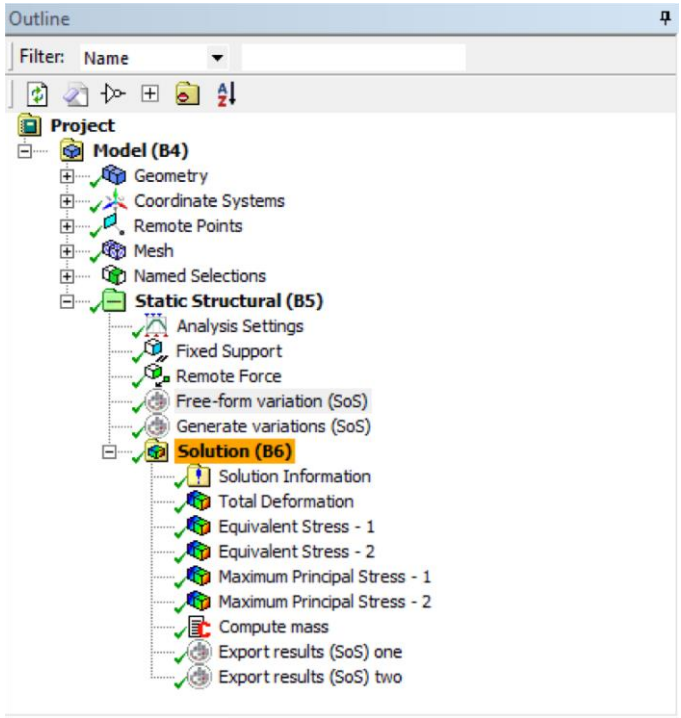






ANSYS Mechanical
2. Define Free-form variation model

- Chose “Create” – “Free Form Variation model”
- Used to create localized variation patterns (for creation of random geometries, for optimization or sensitivity analysis)

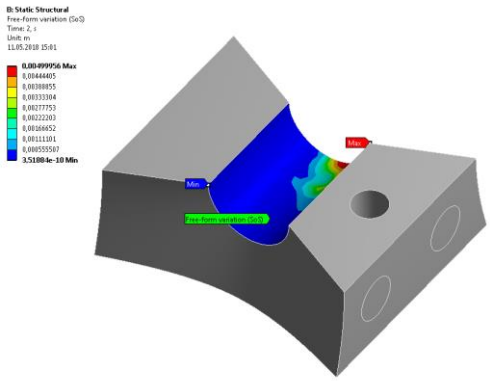


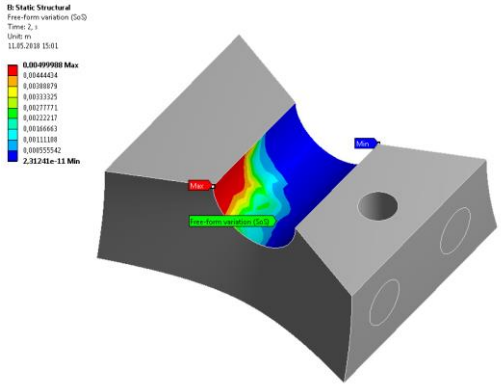
Details of "Free-form variation (SoS)"

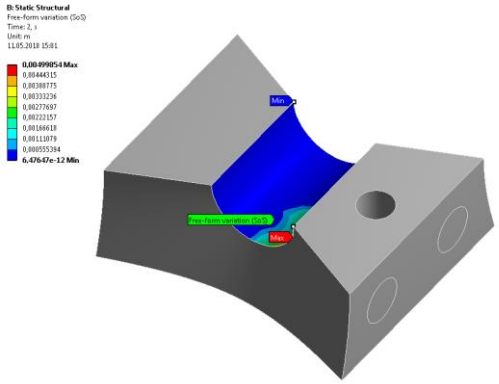
Boundary to be parameterized	
Scoping Method	Named Selection
Named Selection	Boundary_to_be_morphed
Fixed boundary	
Scoping Method	Named Selection
Named Selection	Boundary_to_be_fixed
Mesh part to be exported to SoS	
Scoping Method	Named Selection
Named Selection	NS_Solid
Definition	
Support points	Auto
Number of auto-parameters	20
Magnitude of free variation (if scale factor=1)	0,005 [m]
Visualization	
Visible variation shape index (> =1)	1
Advanced options	
Constant variation	Do not use
Use mesh stabilization	True
Test on mesh distortion	False
Shape of variation patterns	Normalized
Move nodes along	Boundary normal
Solver options	
Number of CPUs used by SoS (0=all)	0
Internal directory	SoS_FreeForm

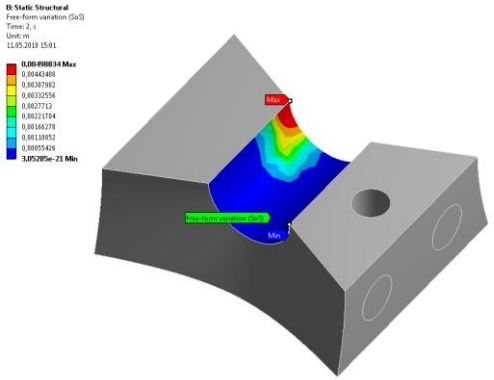
ANSYS Mechanical
Free-form variation model

- The variation shapes are always positive with a single maximum around the support point(s)



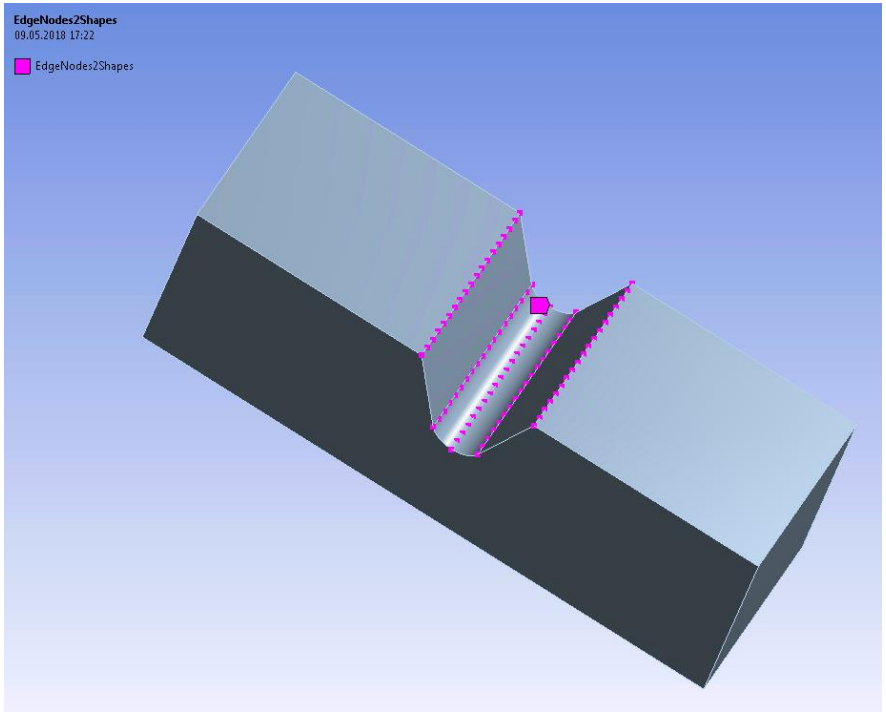






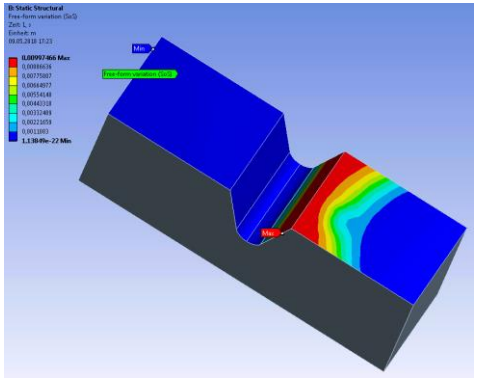
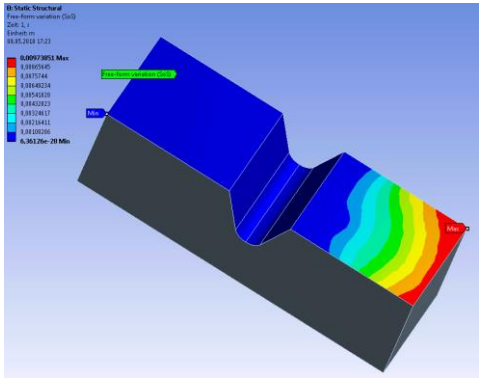
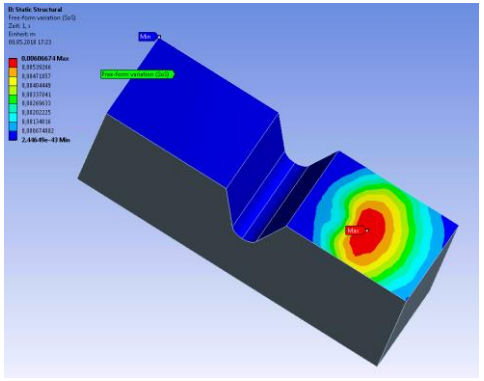
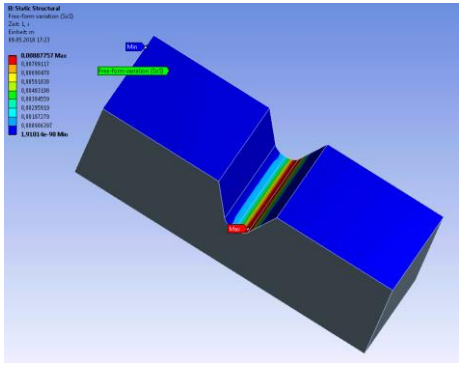
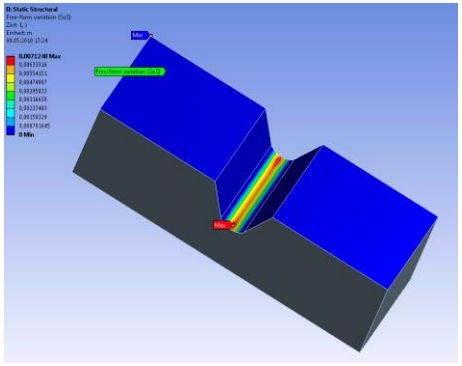
ANSYS Mechanical
Free-form variation model

- It is possible to define manually selected support points
- You can also select edges or faces in order to affect the shape of the associated variation



ANSYS Mechanical
Free-form variation model

- The variation shapes are always positive with a single maximum around the support point(s); Here with mixed automatic and manually defined shapes:

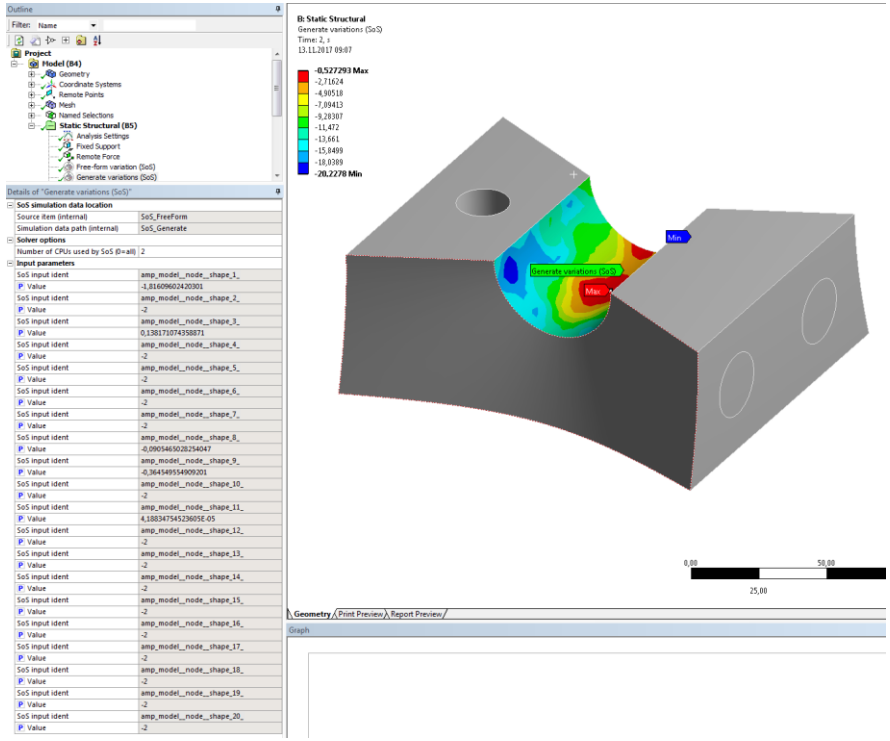
ANSYS Mechanical
2. Vary shape parameters

- Define shape scaling parameters (parametrization or values)
- Get visual response on applied variation

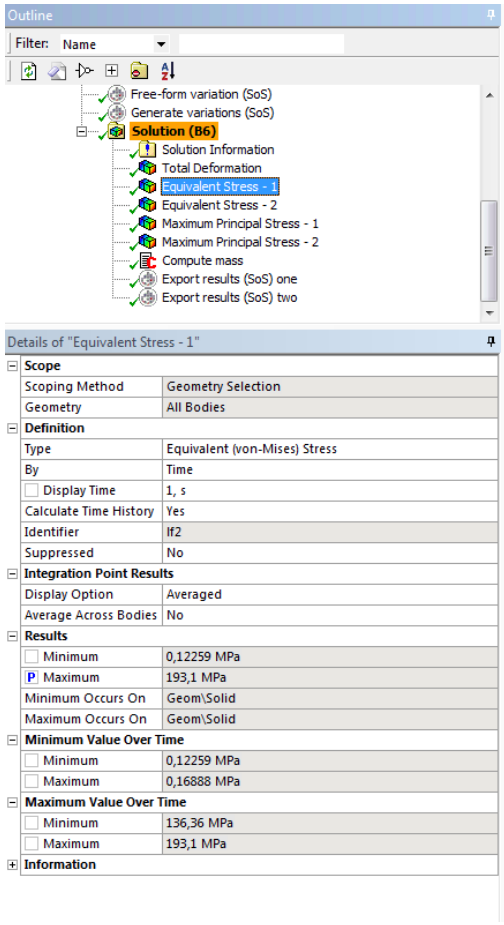
Free-form variation (SoS)
Generate variations (SoS)

Details of "Generate variations (SoS)"

SoS simulation data location	
Source item (internal)	SoS_FreeForm
Simulation data path (internal)	SoS_Generate
Solver options	
Number of CPUs used by SoS (0=all)	2
Input parameters	
SoS input ident	amp_model__node__shape_1_
P Value	-1,81609602420301
SoS input ident	amp_model__node__shape_2_
P Value	-2
SoS input ident	amp_model__node__shape_3_
P Value	0,138171074358871
SoS input ident	amp_model__node__shape_4_
P Value	-2
SoS input ident	amp_model__node__shape_5_
P Value	-2
SoS input ident	amp_model__node__shape_6_
P Value	-2
SoS input ident	amp_model__node__shape_7_
P Value	-2
SoS input ident	amp_model__node__shape_8_
P Value	-0,0905465028254047
SoS input ident	amp_model__node__shape_9_
P Value	-0,364549554909201
SoS input ident	amp_model__node__shape_10_
P Value	-2
SoS input ident	amp_model__node__shape_11_
P Value	4,18834754523605E-05

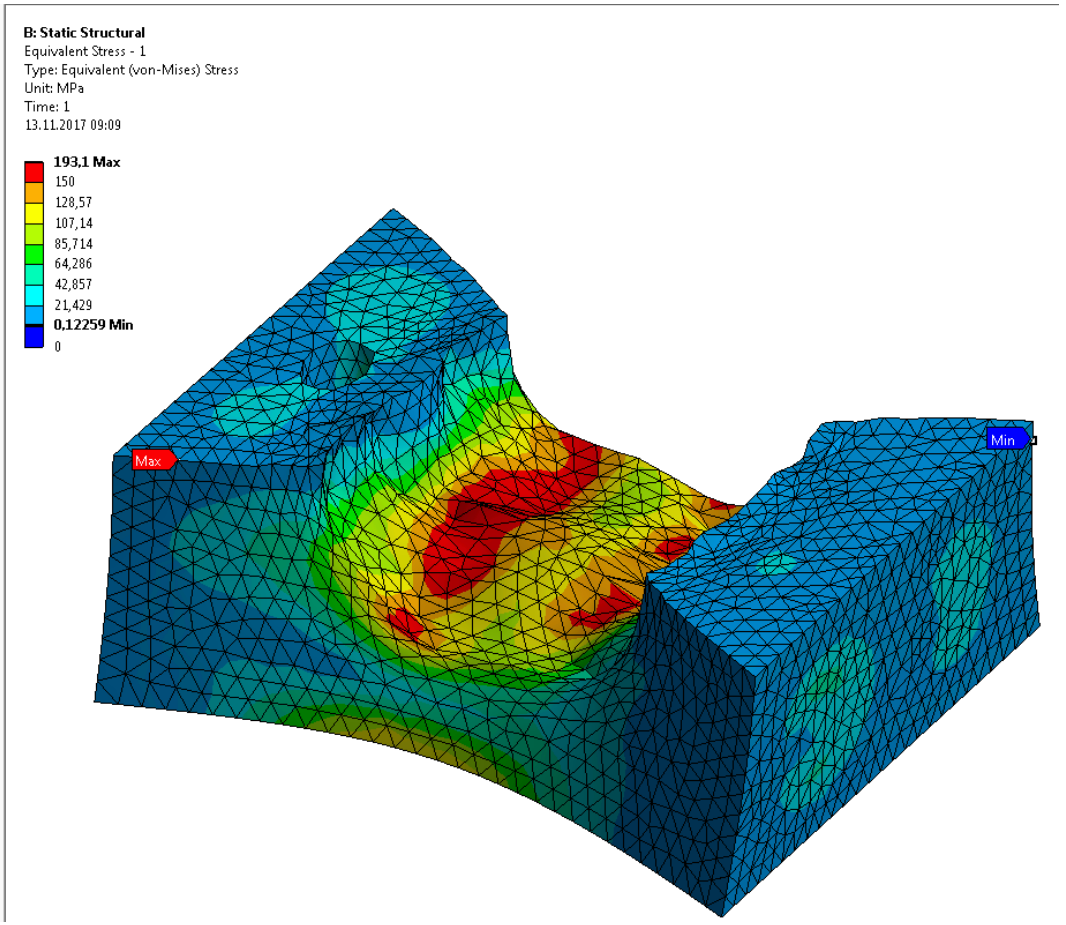


ANSYS Mechanical
3. Solve and display solution on deformed mesh



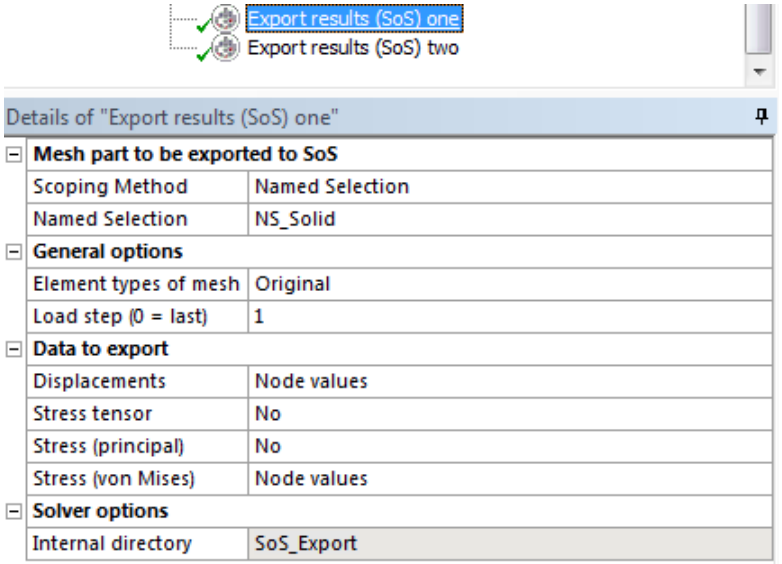
Details of "Equivalent Stress - 1"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Equivalent (von-Mises) Stress
By	Time
Display Time	1, s
Calculate Time History	Yes
Identifier	If2
Suppressed	No
Integration Point Results	
Display Option	Averaged
Average Across Bodies	No
Results	
Minimum	0,12259 MPa
Maximum	193,1 MPa
Minimum Occurs On	Geom\Solid
Maximum Occurs On	Geom\Solid
Minimum Value Over Time	
Minimum	0,12259 MPa
Maximum	0,16888 MPa
Maximum Value Over Time	
Minimum	136,36 MPa
Maximum	193,1 MPa
Information	



4. Export field results to SoS

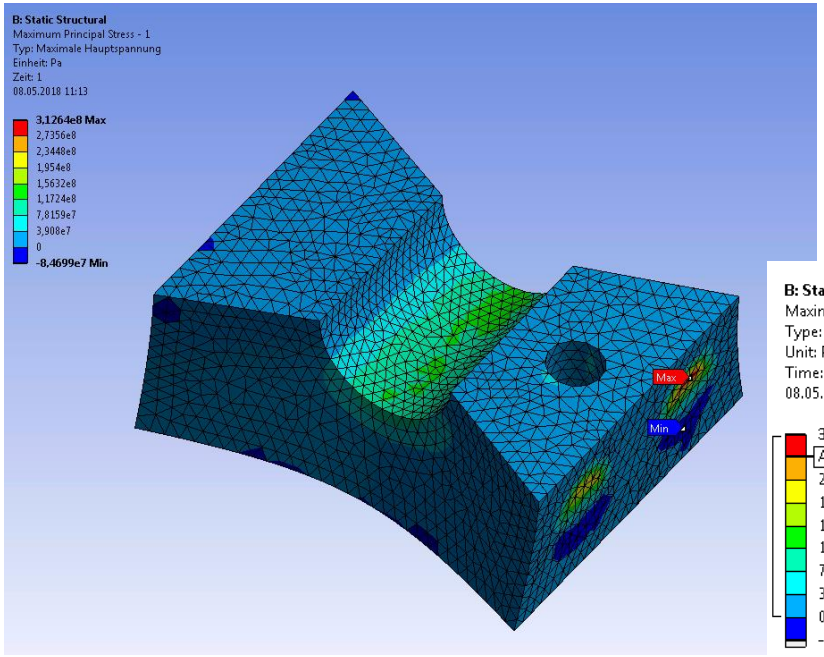
- Stress, displacements, etc.
- Export for different load cases
- Export to SoS-GUI for creation of FMOP etc.
- FMOP needs:
  - 50-200 design points
  - From one DP it choses the reference mesh
  - From all DP it imports the field data assigned to this mesh
  - The ACT plugin exports mesh and data
  - For FMOP it is sufficient to export linear elements (saves CPU time + RAM)
- Then open SoS-GUI, import all data and create FMOP

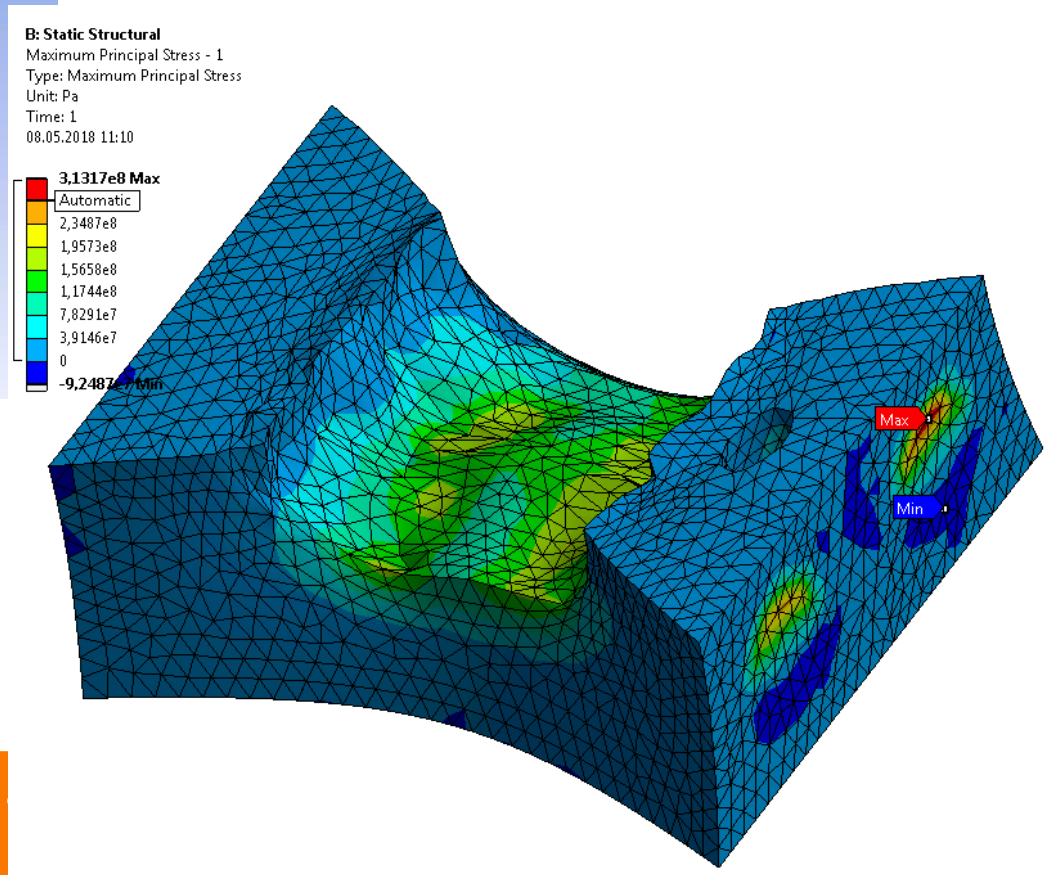


Export options:

- Mechanical (U, stresses)
- Thermal (Temperature)
- General (Any single result from RST/RTH)

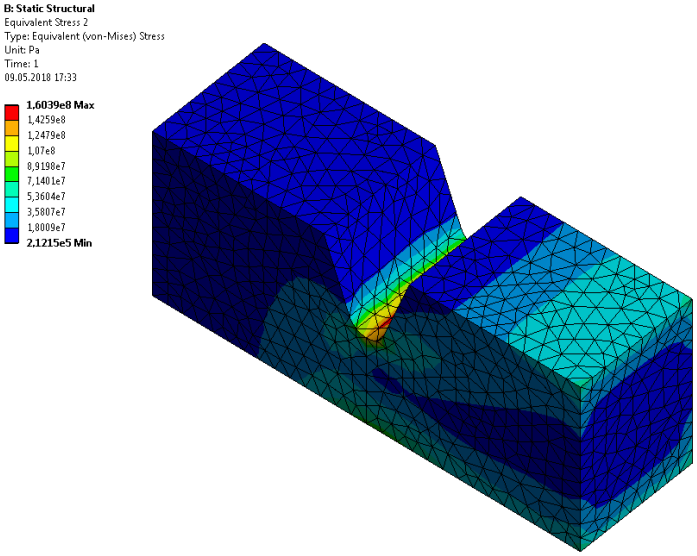
Example:
Minimize mass while keeping stress



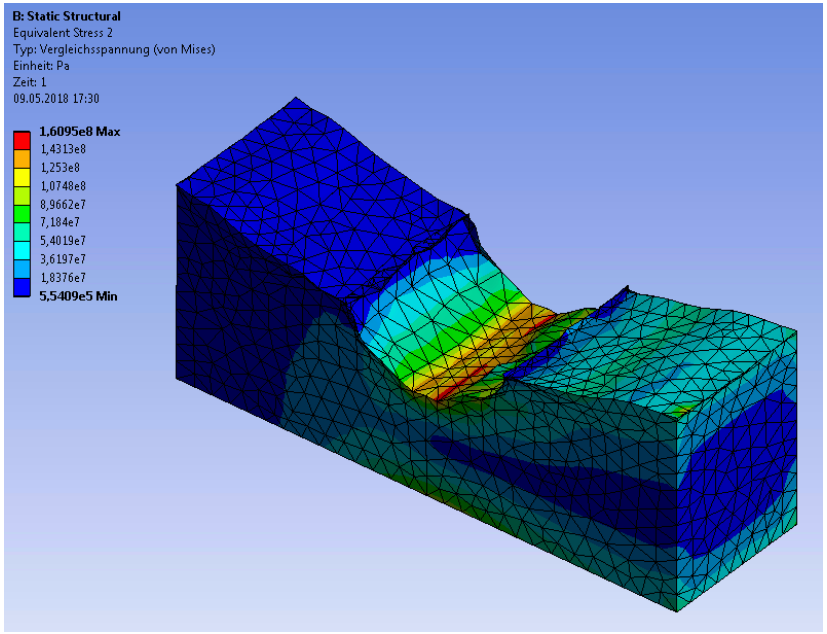


With 20 parameters
(20 automatically defined points)

Example:
Minimize mass while keeping stress



With 11 parameters (5 edges and 6 automatically defined points)



Software demonstration

dynardo

dynamic software & engineering

contact@dynardo.at
Dr. Sebastian Wolff