

Advanced Integrate with Ansys optiSlang
and Mechanical Software



Powering Innovation That Drives Human Advancement

Hands-on: Process Integration, Sensitivity Study, Optimization Steel hook

Please note:

- These training materials were developed and tested in Ansys Release 2024 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.
- Although some workshop files may open successfully in previous releases, backward compatibility is somewhat unlikely and is not guaranteed.

Release 2023 R1

Agenda

Session	Slide Set	Time	Topic
1	0	5'	Agenda
	1	25'	Introduction to Ansys optiSLang
		10'	Ansys optiSLang in the Ansys Learning Hub – Find your Examples
		15'	Q/A
	2	30'	Sensitivity Study and Optimization – Theoretical Background
2	3	75'	Hands-on – Process Integration, Sensitivity Study and Postprocessing Steel Hook – optiSLang inside Workbench
		15'	Q/A
3	4	40'	Hands-on – Optimization Steel Hook – optiSLang inside Workbench
	5	20'	Robust Design Optimization – Theoretical Background
	6	40'	Hands-on – Robustness Evaluation Steel Hook – optiSLang inside Workbench
		15'	Q/A



Overview

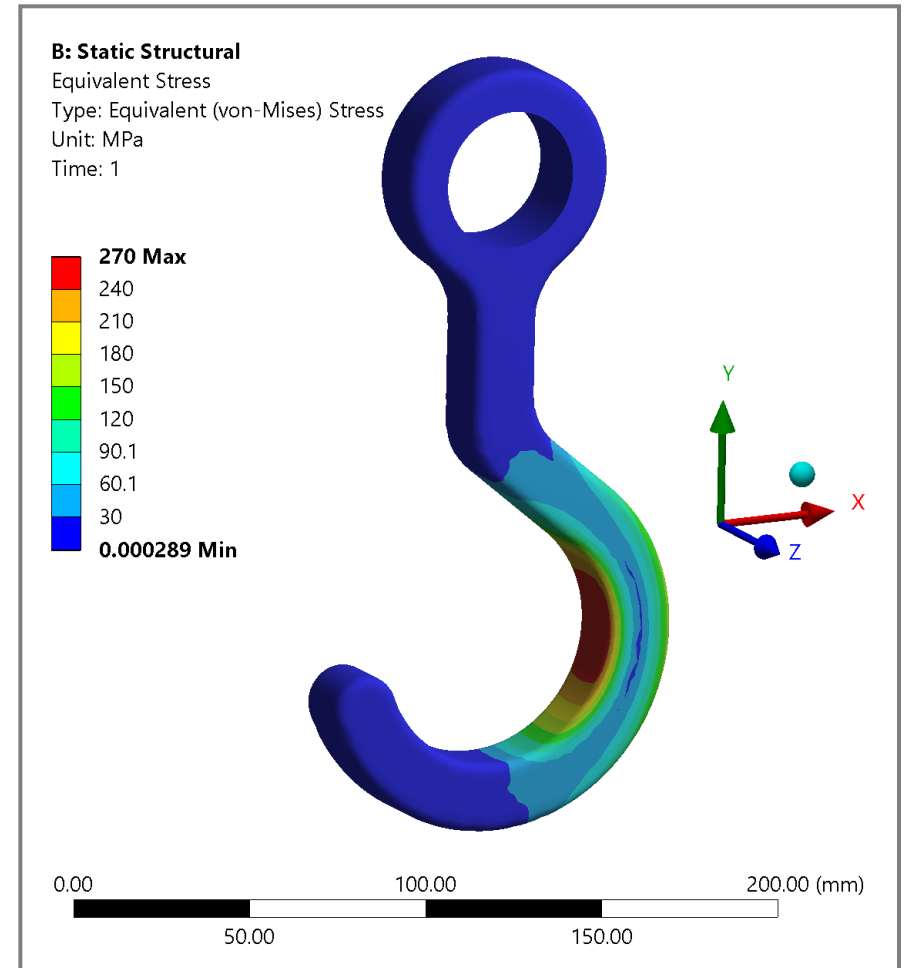
Problem Description

The task is to design a steel hook such that

- The von-Mises stress under a defined load does not exceed 300 MPa
- The mass is as small as possible
- Certain geometry parameters are within the predefined bounds.

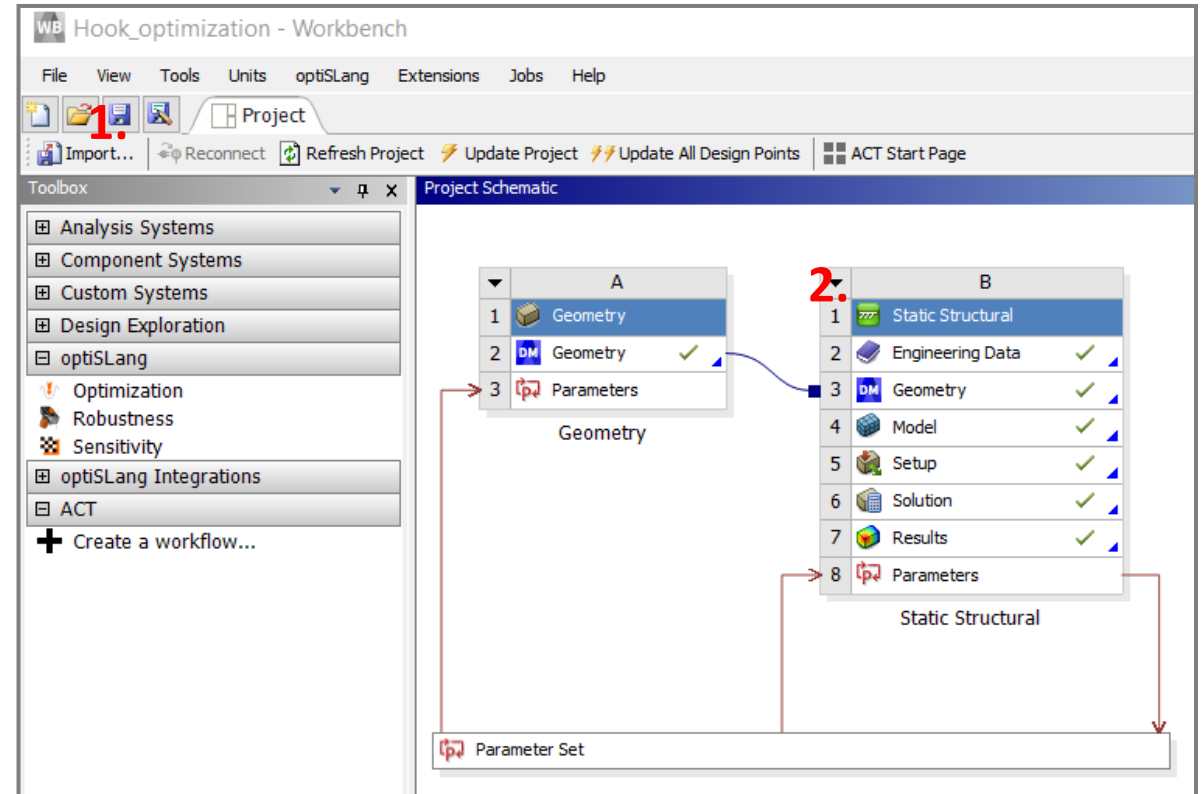
In this tutorial the

- setup of the required solver chain
- sensitivity analysis and
- optimization on MOP is discussed.



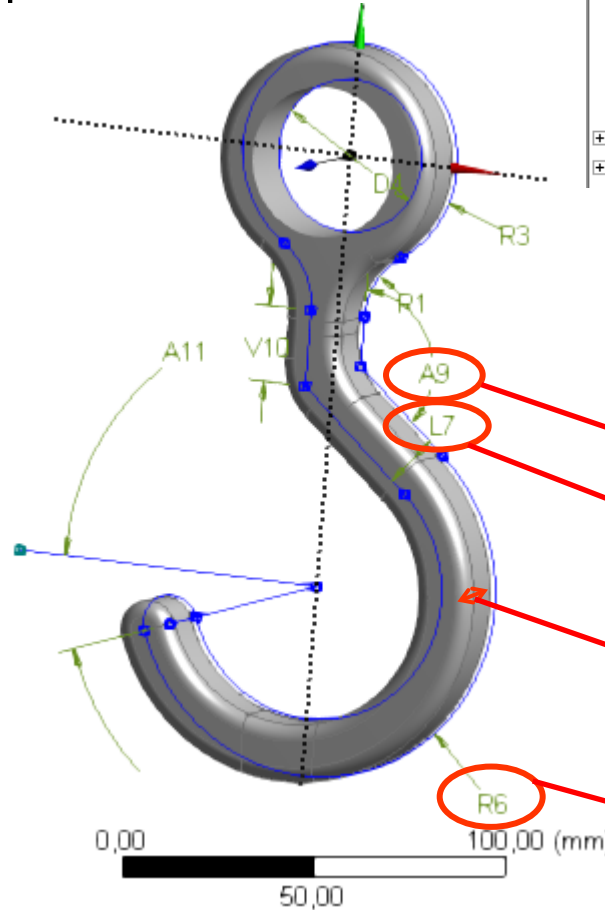
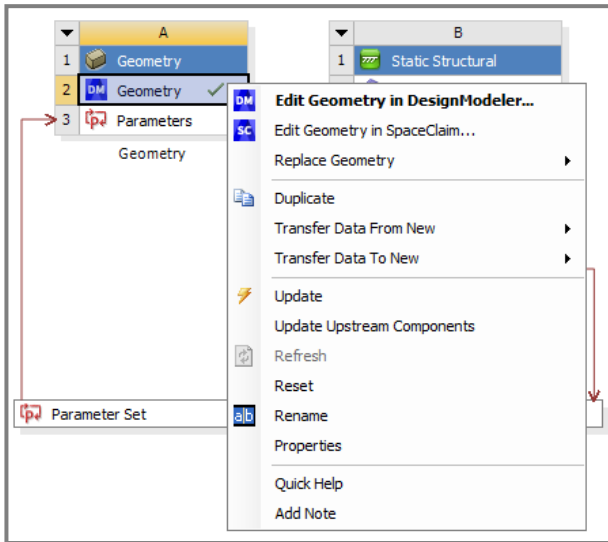
Solver: Ansys Mechanical

1. Open the archived Workbench project Hook_Sensitivity_23R1.wbpz and save it.
2. In Ansys Workbench the Ansys Mechanical is used as solver.



Input Parameters: Geometry

- Right-click on the Geometry cell and open DesignModeler to review the Design parameters.



Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 7	
D A9	130 °
D4	40 mm
D L7	15 mm
R1	20 mm
R3	30 mm
D R6	50 mm
V10	20 mm
Edges: 11	
References: 1	

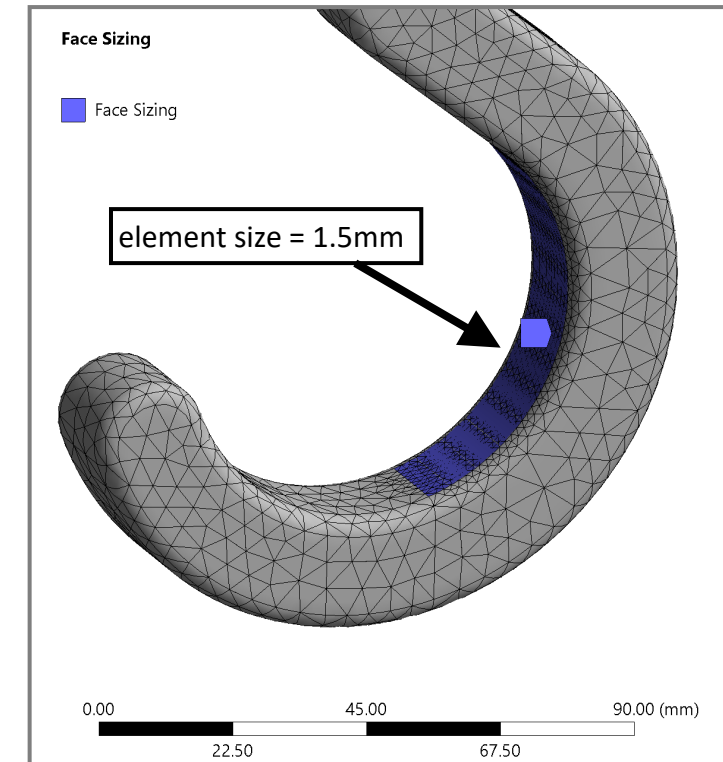
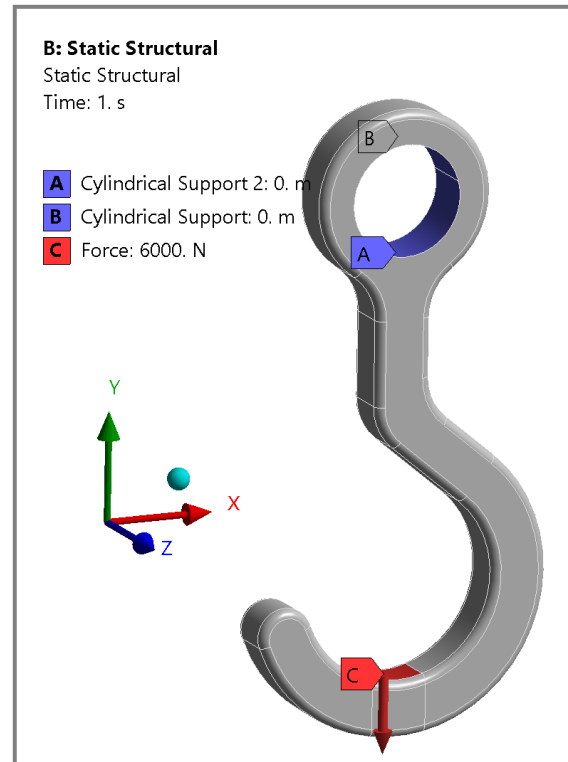
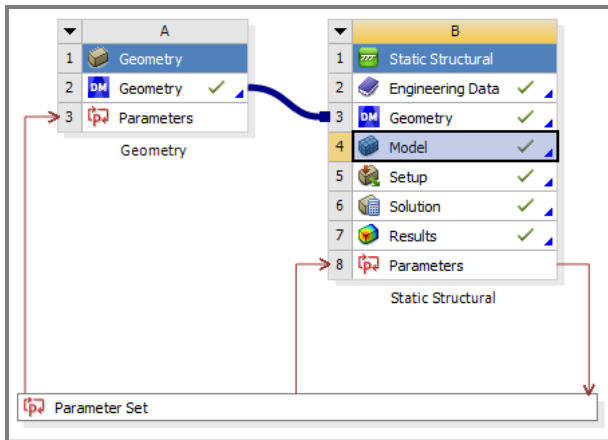
Details View	
Details of Extrude1	
Extrude	Extrude1
Geometry	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	Fixed
D FD1, Depth (>0)	20 mm
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	

DS_Angle	(120-150°)
DS_Thickness	(15-25 mm)
DS_Depth	(15-25 mm)
DS_LowerRadius	(45-55 mm)

Loading conditions

Review the solution setup in Ansys Mechanical (double-click on the Model cell)

- Loading force $F = 6000\text{ N}$
- Cylindrical support, tangential direction is free.



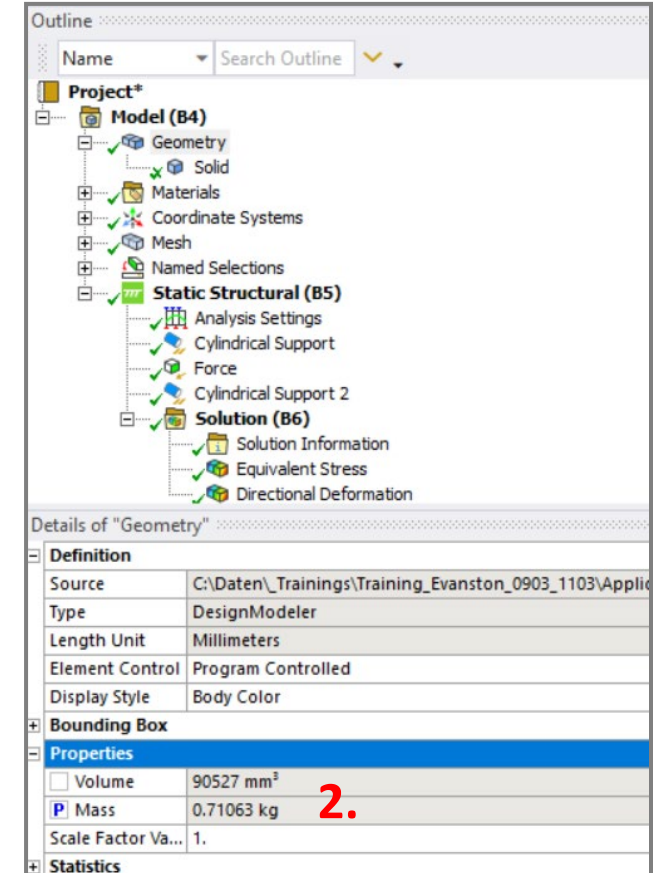
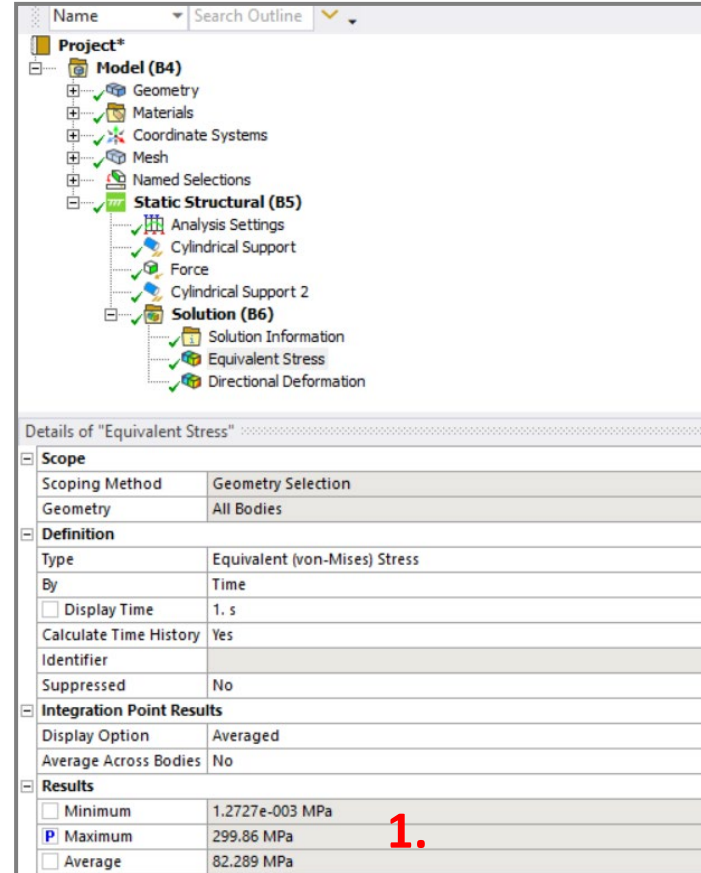
Output Values

Responses in Ansys Mechanical:

For the objective function and constraint condition

1. the maximal stress and
2. the mass

are parameterized.



The Ansys logo, featuring a stylized yellow and black 'A' followed by the word 'nsys' in black.

