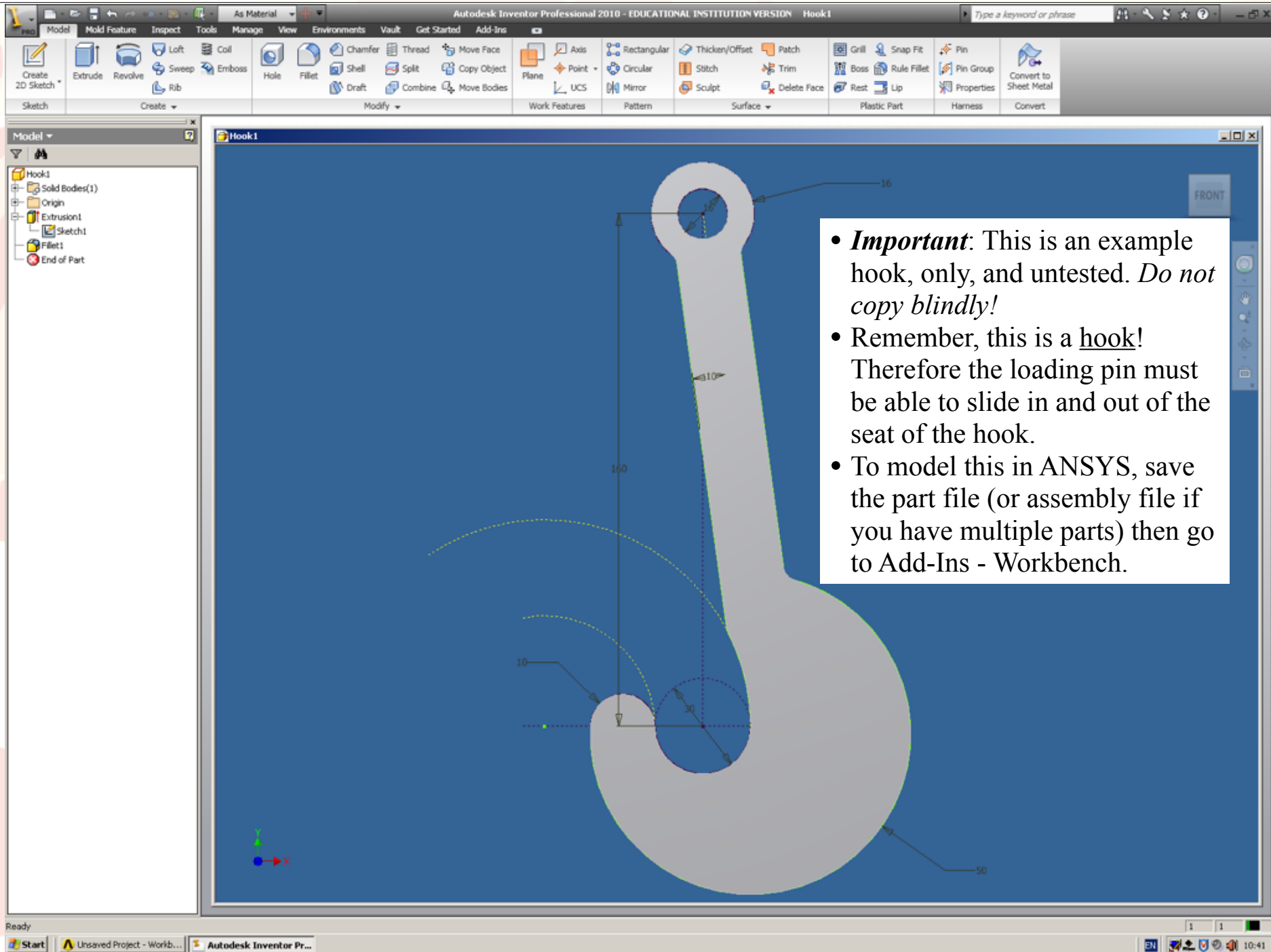




ANSYS

Dr Francis J. Franklin

F.J.Franklin@ncl.ac.uk



- **Important:** This is an example hook, only, and untested. *Do not copy blindly!*
- Remember, this is a hook! Therefore the loading pin must be able to slide in and out of the seat of the hook.
- To model this in ANSYS, save the part file (or assembly file if you have multiple parts) then go to Add-Ins - Workbench.

When ANSYS Workbench starts:

- If there is no Geometry in the Project Schematic window, then add one. If you started Workbench from Inventor, there should be one already.
- Add an Engineering Data toolbox, and a Static Structural toolbox.
- Drag Geometry Row 2 into Static Structural Row 3.
- Drag Engineering Data Row 2 in Static Structural Row 2.
- Left-click Model to select it, then right-click to get the menu, then Edit to enter ANSYS Modeller.

Messages		
	A	B
	Type	Text
1		
2	ANSYS News	ANSYS and SpaceClaim Streamline Engineering Design and Simulation Driven Product Development
3	ANSYS News	ANSYS Harnessing the Power of Ocean Waves
4	ANSYS News	Firehole Releases Version 2.0 of HeliusMCT™
5	ANSYS News	ANSYS Launches Immersed Boundary Module for Rapid Design Evaluation
6	ANSYS News	ANSYS and HBM-nCode Partner to Deliver Advanced Fatigue Capabilities
7	ANSYS News	ANSYS Ranked #33 on FORTUNE 100 Fastest-Growing Companies List
8	ANSYS News	ANSYS CEO to Present at Kaufman Bros. 12th Annual Investor Conference
9	ANSYS News	ANSYS Paves the Way for Economic and Environmental Improvements in Cement Manufacturing
10	ANSYS News	ANSYS, Inc. Reports Second Quarter 2009 Results and Updates Outlook
11	ANSYS News	Maxwell Technologies and ANSYS Release Ultracapacitor Components Library for Use in Simpler
12	ANSYS News	VNIIGAZ Selects ANSYS Multiphysics Software to Drive Innovation in Oil and Gas Pipeline Technology
13	ANSYS News	AREVA Names ANSYS Certified Supplier
14	ANSYS News	Delta Marine Steers Ships in the Right Direction Using ANSYS Engineering Simulation
15	ANSYS News	ANSYS CEO to Present at Oppenheimer Annual Communications, Technology & Internet Conference
16	ANSYS News	ANSYS, Inc. to Release Second Quarter 2009 Earnings August 6, 2009
17	ANSYS News	Mining Prospects Enhanced with ANSYS Engineering Simulation

B: Static Structural (ANSYS) - Mechanical [ANSYS Academic Teaching Advanced]

File Edit View Units Tools Help Solve

Environment: Metric (m, kg, N, s, V, A)
 Metric (cm, g, dyne, s, V, A)
 Metric (mm, kg, N, s, mV, mA)
 Metric (mm, t, N, s, mV, mA)
 Metric (mm, dat, N, s, mV, mA)
 Metric (µm, kg, µN, s, V, A)
 U.S. Customary (ft, lbm, lbf, °F, s, V, A)
 U.S. Customary (in, lbm, lbf, °F, s, V, A)

Outline: Project Model Ge Co Me St

Details of "Bearing Load"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Bearing Load
Define By	Components
Coordinate System	Global Coordinate System
☐ X Component	0. N
☒ Y Component	-30000 N
☐ Z Component	0. N
Suppressed	No

B: Static Structural (ANSYS)
 Bearing Load
 Time: 1. s
 04/10/2009 10:44

Bearing Load: 30000 N
 Components: 0., -30000, 0. N

Menu: Units:
 • Choose suitable units.

0.000 0.025 0.050 0.075 0.100 (m)

Geometry Worksheet Print Preview Report Preview

Graph Tabular Data

Press F1 for Help No Messages No Selection Metric (m, kg, N, s, V, A) Degrees rad/s Celsius

Start Unsaved Project - Workb... Unsaved Project - Workb... Autodesk Inventor Profe... B: Static Structural (...)

ANSYS



B: Static Structural (ANSYS) - Mechanical [ANSYS Academic Teaching Advanced]

File Edit View Units Tools Help | Solve | Mesh | Update | Mesh | Mesh Control | Options

Outline

- Project
 - Model (B4)
 - Geometry
 - Coordinate Systems
 - Mesh
 - Face Sizing
 - Static Structural (B5)
 - Analysis Settings
 - Frictionless Support
 - Cylindrical Support
 - Bearing Load
 - Solution (B6)
 - Solution Information
 - Equivalent Stress
 - Stress Tool
 - Safety Factor

Face Sizing
Static Structural
04/10/2009 10:53

☐ Face Sizing

Details of "Face Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	3 Faces
Definition	
Suppressed	No
Type	Element Size
Element Size	1. mm
Behavior	Soft

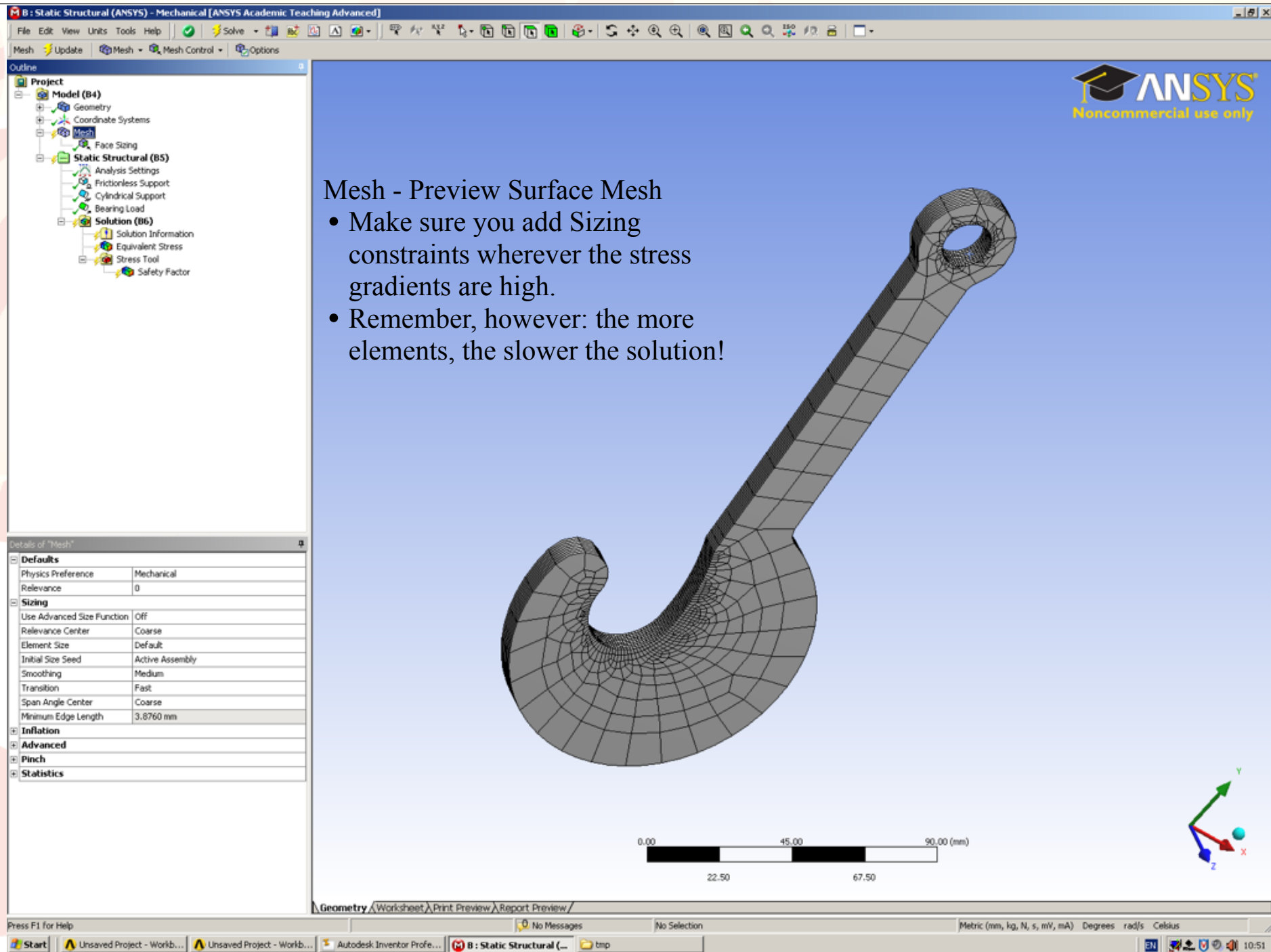
Geometry | Worksheet | Print Preview | Report Preview

Press F1 for Help | No Messages | No Selection | Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Start | Unsaved Project - Workb... | Unsaved Project - Workb... | Autodesk Inventor Profe... | B: Static Structural (... | tmp | Hook1AnsysMesh - Paint | 10:53

Under Mesh:

- Select faces where stresses need to be solved most accurately, and add Sizing constraints.
- Fill in the Element Size field.



B: Static Structural (ANSYS) - Mechanical [ANSYS Academic Teaching Advanced]

File Edit View Units Tools Help

Environment Inertial Loads Supports

Outline

- Project
 - Model (B4)
 - Geometry
 - Coordinate Systems
 - Mesh
 - Face Sizing
 - Static Structural (B5)
 - Analysis Settings
 - Frictionless Support
 - Cylindrical Support
 - Bearing Load
 - Solution (B6)
 - Solution Information
 - Equivalent Stress
 - Stress Tool
 - Safety Factor

B: Static Structural (ANSYS)
 Bearing Load
 Time: 1. s
 04/10/2009 10:54

Bearing Load: 30000 N
 Components: 0, -30000, 0, N

Under Static Structural:

- Add a bearing load to simulate the load from the pin. The bearing load is largest in the centre of the region and tails off towards the edges.
- Set the required force - change the field marked Vector to Components to get full control over the direction of the applied force.

Details of "Bearing Load"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Bearing Load
Define By	Components
Coordinate System	Components
☐ X Component	Vector
☐ Y Component	-30000 N
☐ Z Component	0, N
Suppressed	No

0.00 25.00 50.00 75.00 100.00 (mm)

Geometry Worksheet Print Preview Report Preview

Graph Tabular Data

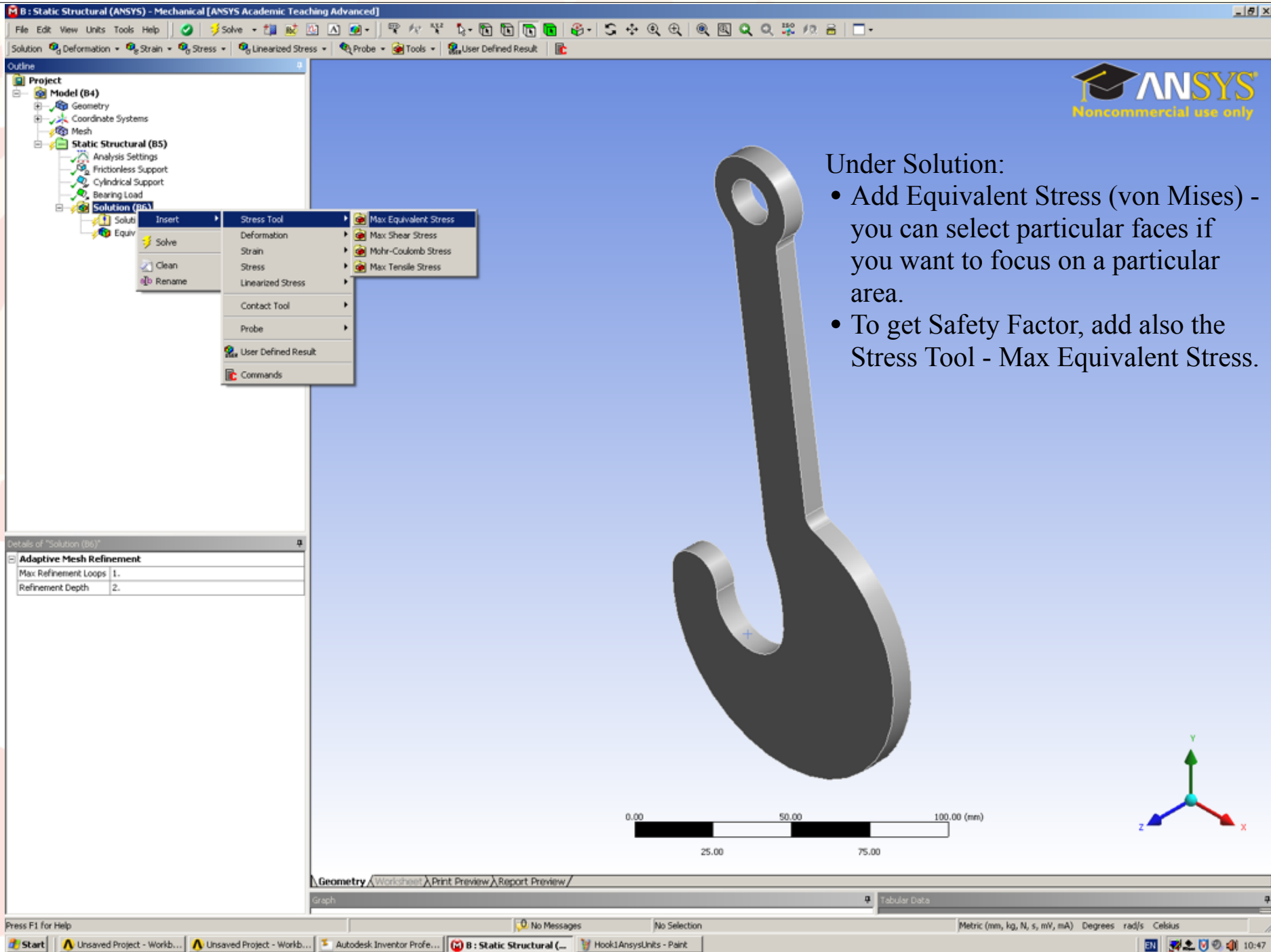
Press F1 for Help

No Messages No Selection

Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Start Unsaved Project - Workb... Unsaved Project - Workb... Autodesk Inventor Profe... B: Static Structural (Hook1AnsysMeshSizing -...

10:54



Under Solution:

- Add Equivalent Stress (von Mises) - you can select particular faces if you want to focus on a particular area.
- To get Safety Factor, add also the Stress Tool - Max Equivalent Stress.

B: Static Structural (ANSYS) - Mechanical [ANSYS Academic Teaching Advanced]

File Edit View Units Tools Help

Environment Inertial Loads Supports

Outline

- Project
 - Model (B4)
 - Geometry
 - Coordinate Systems
 - Mesh
 - Face Sizing
 - Static Structural (B5)**
 - Analysis Settings
 - Frictionless Support
 - Cylindrical Support
 - Bearing Load
 - Solution (B6)
 - Solution Information
 - Equivalent Stress
 - Stress Tool
 - Safety Factor

Details of "Static Structural (B5)"

Definition

Physics Type	Structural
Analysis Type	Static Structural
Solver Target	ANSYS Mechanical

Options

Environment Temperature	22. °C
Generate Input Only	No

Static Structural (ANSYS)

Static Structural
Time: 1. s
04/10/2009 10:55

- A** Frictionless Support
- B** Cylindrical Support: 0. mm
- C** Bearing Load: 30000 N

Also under Static Structural:

- Add a frictionless support to one side, to stop the hook moving sideways in the z-direction.
- Add a cylindrical support at the top, but allow the support to move tangentially. (Unfortunately, this allows tension across the interface!)

0.00 25.00 50.00 75.00 100.00 (mm)

Geometry Worksheet Print Preview Report Preview

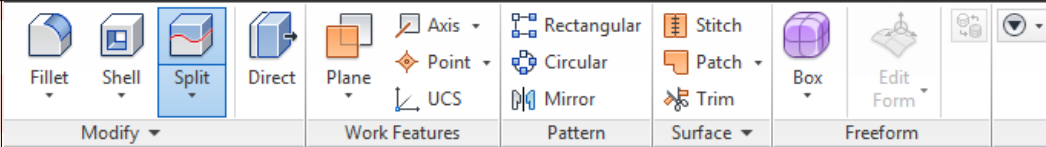
Graph Tabular Data

Press F1 for Help

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

Start Unsaved Project - Workb... Unsaved Project - Workb... Autodesk Inventor Profe... B: Static Structural (Hook1AnsysBearing - Paint

10:55

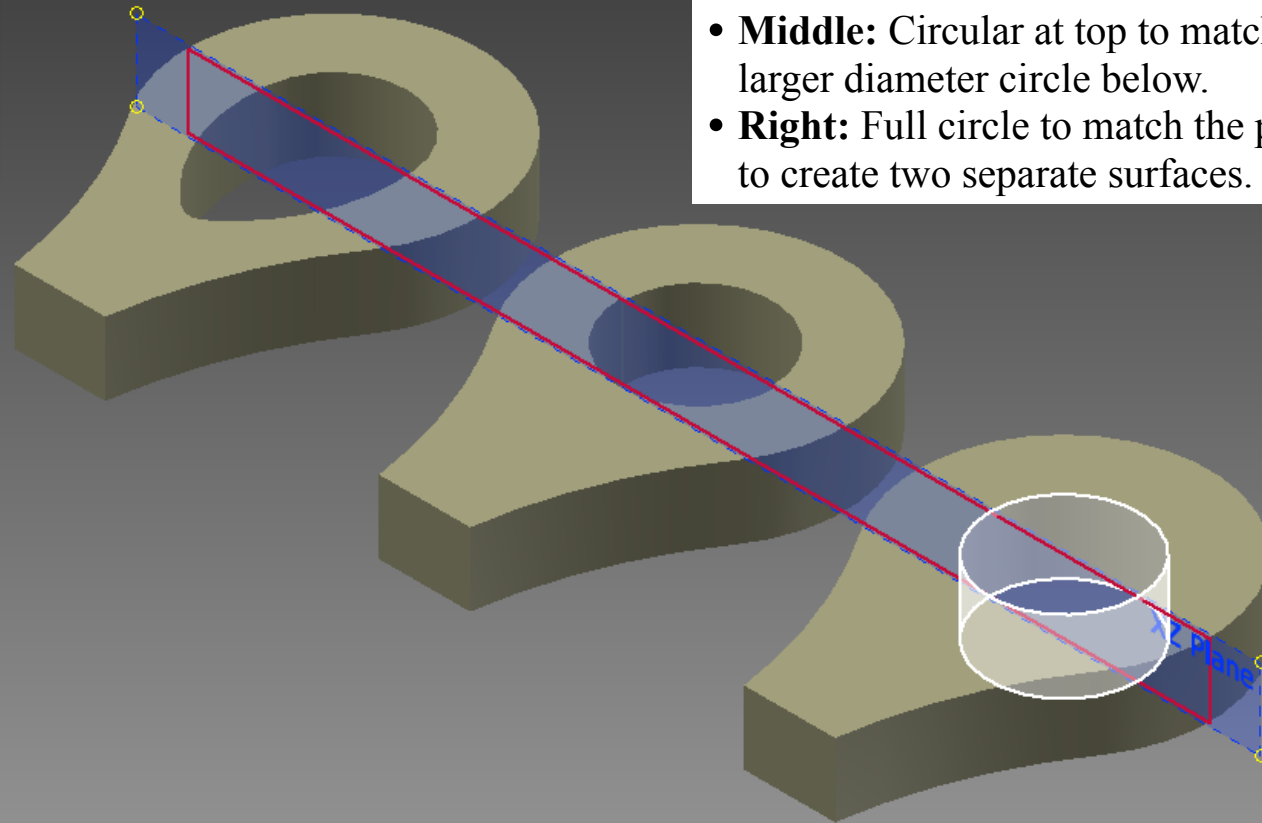
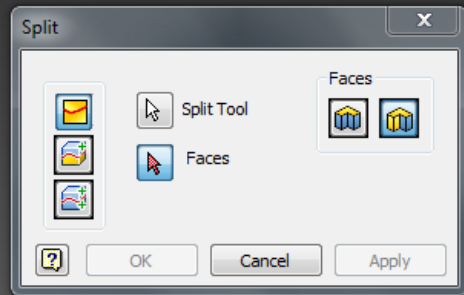


Important: These are example hook heads only, and untested. *Do not copy blindly!*

~~~ Break The Circle ~~~

Three methods shown here:

- **Left:** Circular at top to match the pin; larger elsewhere.
- **Middle:** Circular at top to match the pin; larger diameter circle below.
- **Right:** Full circle to match the pin; use **Split** to create two separate surfaces.





B: Static Structural

Force

Time: 1. s

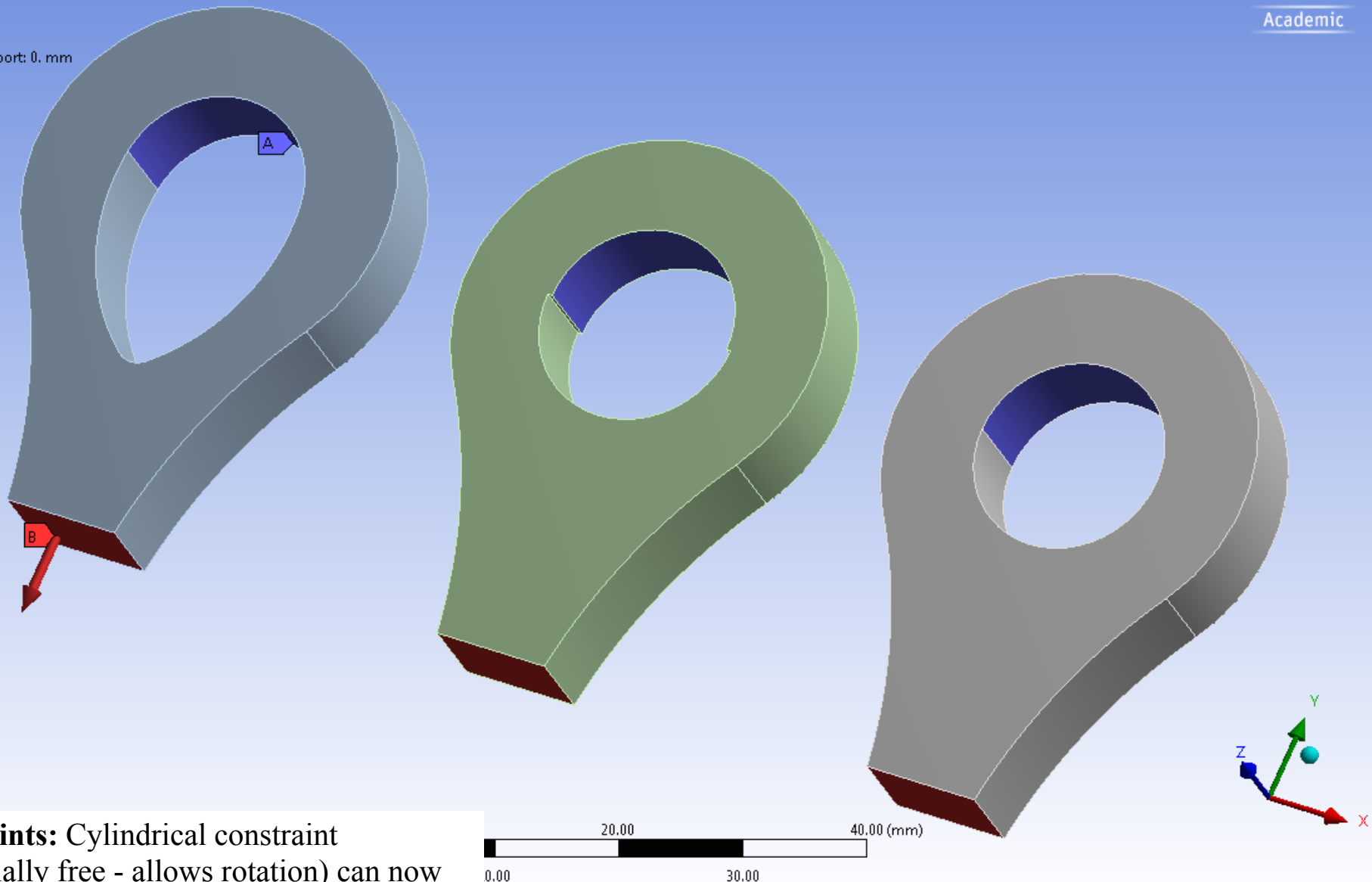
07/10/2015 16:42

A Cylindrical Support: 0. mm

B Force: 90000 N

ANSYS
R15.0

Academic



- **Constraints:** Cylindrical constraint (tangentially free - allows rotation) can now be applied only to top surface.
- **Forces:** Force applied to necks; here 30kN vertically down (-ve y-axis) is applied to each.

**B: Static Structural**

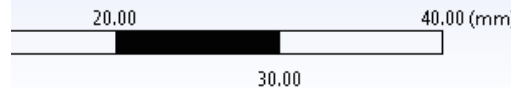
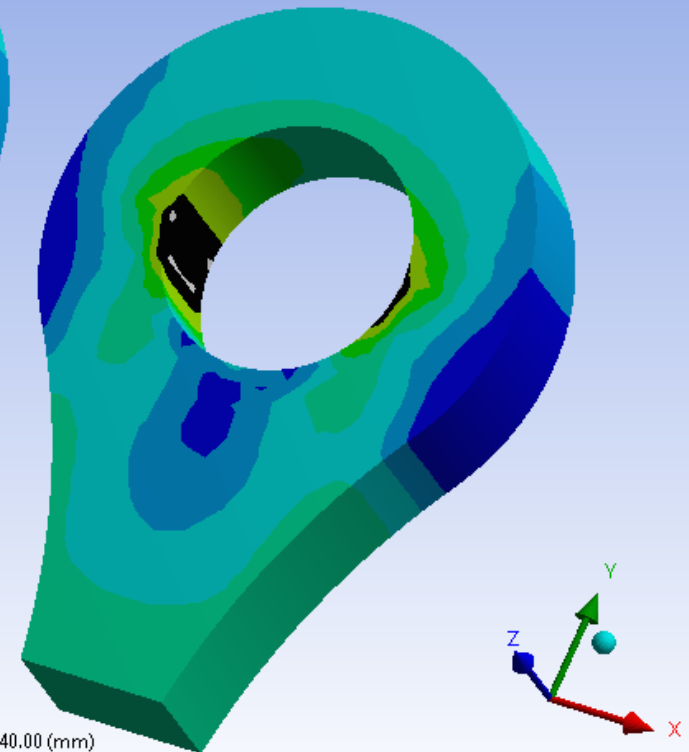
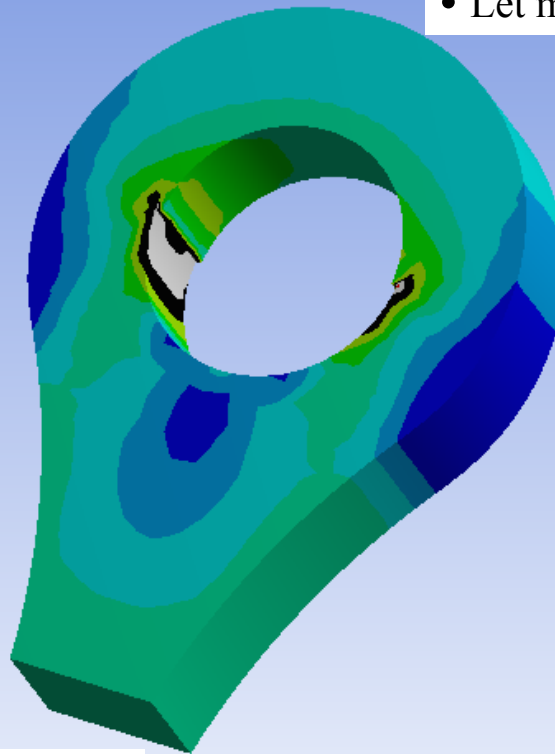
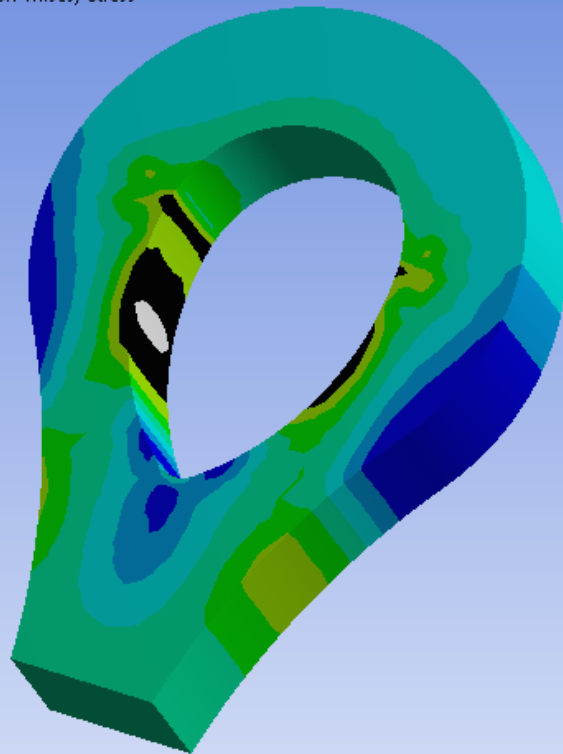
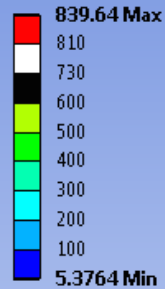
Equivalent Stress

Type: Equivalent (von-Mises) Stress

Unit: MPa

Time: 1

07/10/2015 16:40

**Will my hook yield?**

- Just because the yield stress is exceeded does not mean the test machine will detect yield.
- If the red/white regions are within 0.5mm of the surface, you can *probably* ignore them.
- Let me repeat: *probably*.

- **Colour bar:** Choose values and colours that let you see clearly the stress ranges you are interested in. Here RED indicates stress over UTS, and WHITE indicates stress over yield.
- **Mesh:** Make sure the mesh is sufficiently detailed where stresses change rapidly.
- Notice how the **sharp edges** in the middle design **cause high stresses** even at this low load (30kN).

**B: Static Structural**

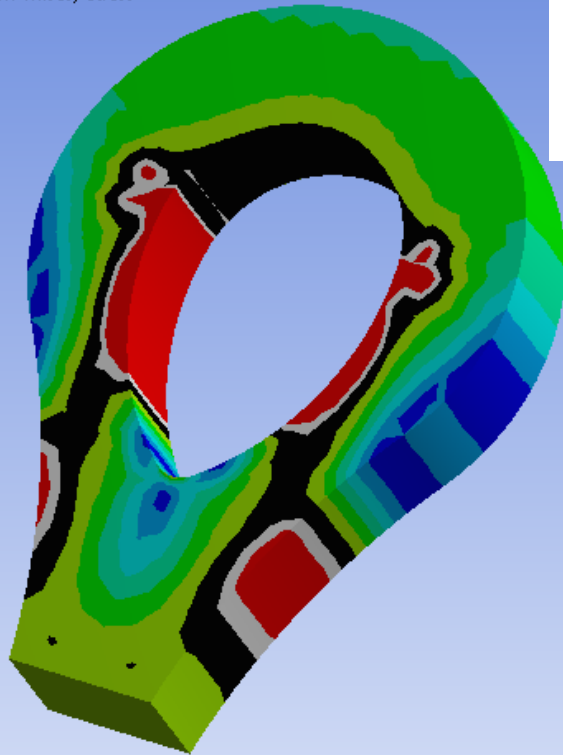
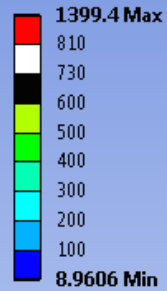
Equivalent Stress

Type: Equivalent (von-Mises) Stress

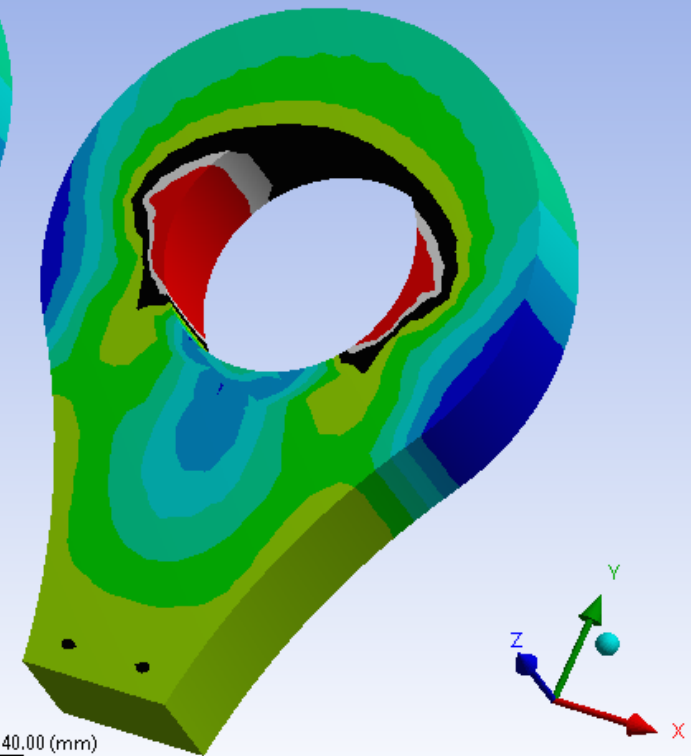
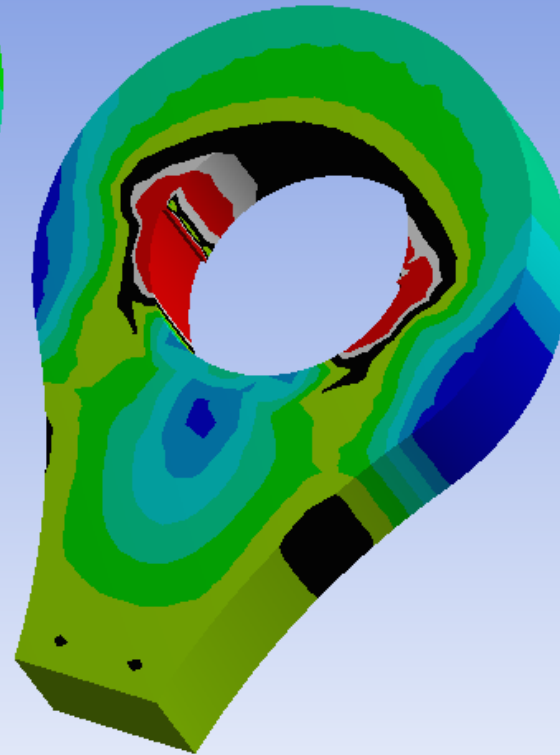
Unit: MPa

Time: 1

07/10/2015 16:38

**Will my hook fail?**

- Just because the UTS is exceeded does not mean the hook will fail.
- Significant red/white regions indicate that there will be significant plasticity, but ANSYS does not model plasticity well.
- My guess: If the red is not continuous across a section, failure is *unlikely*.



- **Colour bar:** Using the same colour bar makes comparison easy.
- Note the region of **low stress** below the hole and the very **high stresses** at the **inner sides** where the bending moment is largest.
- All designs have *significant* white/red regions, so yield by this point (50kN) is *likely*.

**B: Static Structural**

Equivalent Stress

Type: Equivalent (von-Mises) Stress

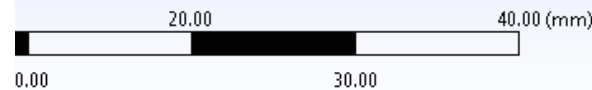
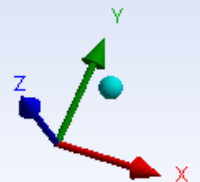
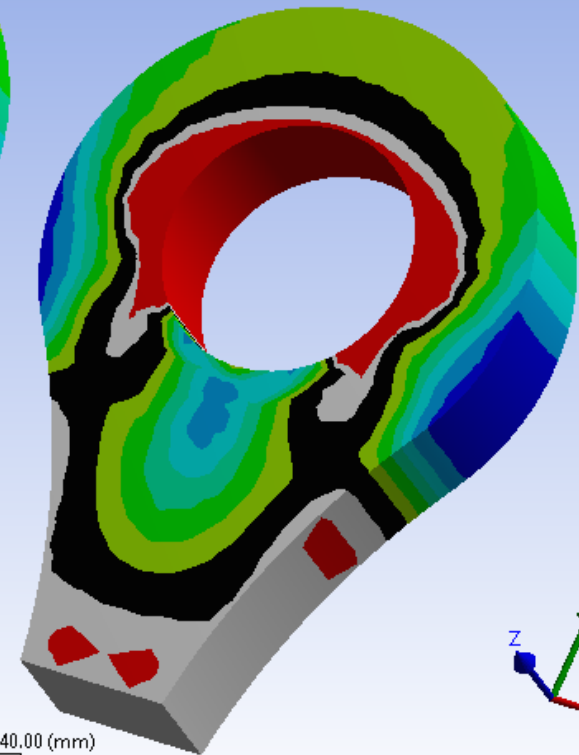
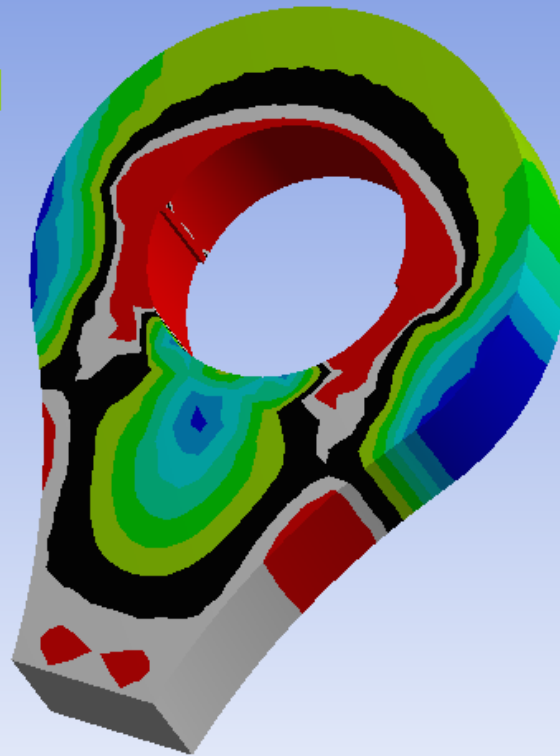
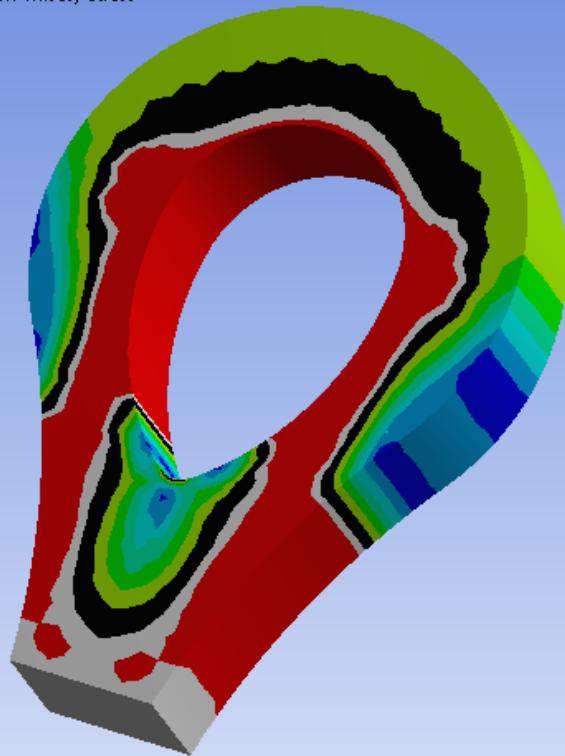
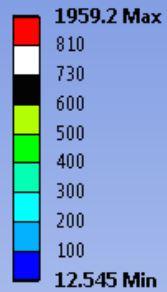
Unit: MPa

Time: 1

07/10/2015 16:37

ANSYS
R15.0

Academic



- All designs are white/red through the entire neck, so yield by this point (70kN) is *certain*.
- In the design on the left, the red is continuous from the inner surface to the outer surface, so failure is *likely*.

B: Static Structural (ANSYS) - Mechanical [ANSYS Academic Teaching Advanced]

File Edit View Units Tools Help

Environment Inertial Loads Supports

Outline

- Project
 - Model (B4)
 - Geometry
 - TopPin:1
 - BottomPin:1
 - Hook:1:1
 - Coordinate Systems
 - Connections
 - No Separation - TopPin:1 To Hook:1:1
 - No Separation - BottomPin:1 To Hook:1:1
 - Mesh
 - Face Sizing
 - Face Sizing - Pins
 - Static Structural (B5)
 - Analysis Settings
 - Force
 - Fixed Support
 - Frictionless Support
 - Solution (B6)
 - Solution Information
 - Equivalent Stress - Hook Only
 - Stress Tool
 - Safety Factor - Hook Only
 - Normal Stress - Hook Only

Details of "Static Structural (B5)"

Definition

Physics Type	Structural
Analysis Type	Static Structural
Solver Target	ANSYS Mechanical

Options

Environment Temperature	22. °C
Generate Input Only	No

B: Static Structural (ANSYS)

Static Structural

Time: 1. s

17/10/2009 11:43

- Fixed Support
- Frictionless Support
- Force: 30000 N

It is also possible to import *assemblies* as the geometry for ANSYS models.

In this example, special pins for loading the hook are created as separate parts in Inventor and the whole assembly imported in the Geometry component in the ANSYS Project Schematic.

- *Fixed Supports* are applied to the outer extrusion on the top pin.
- A vertical *Force* is applied to the bottom pin.
- *Frictionless Supports* are placed on the vertical surfaces of the outer extrusion on the bottom pin to constrain it to move vertically.

0.00 30.00 60.00 (mm)

15.00 45.00

Geometry Worksheet Print Preview Report Preview

Graph

Tabular Data Graph

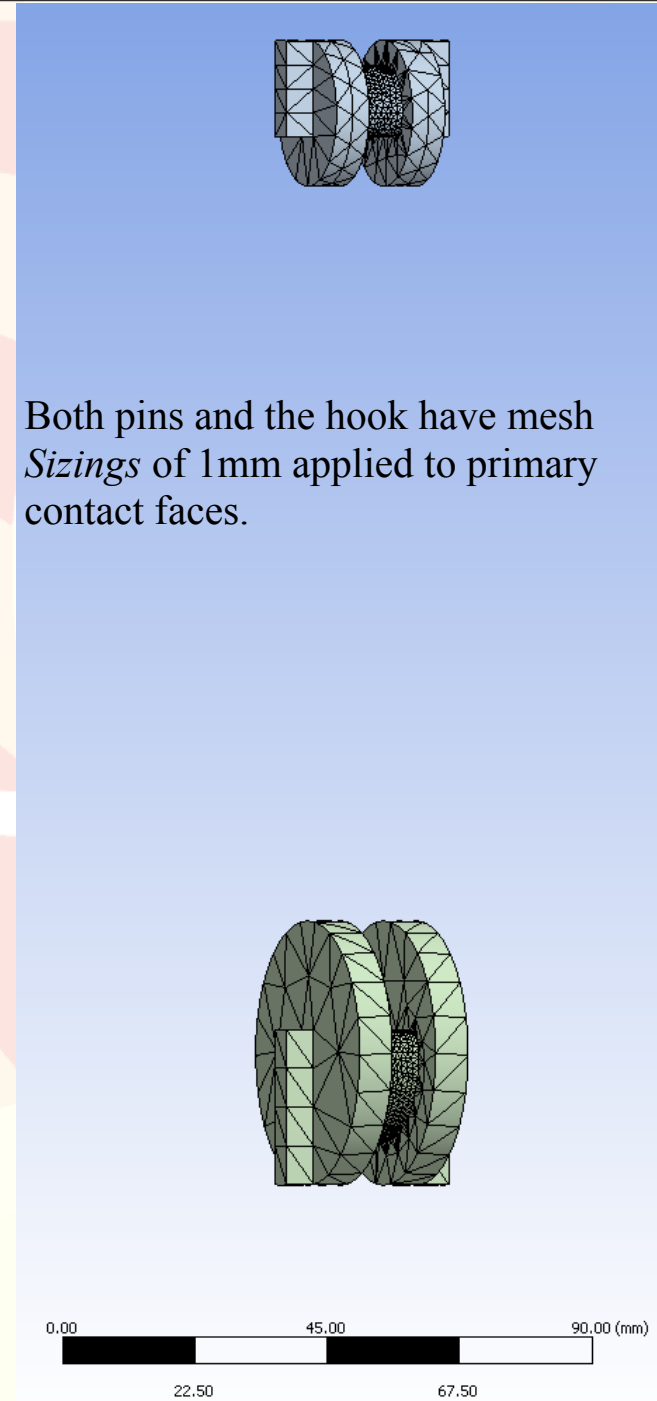
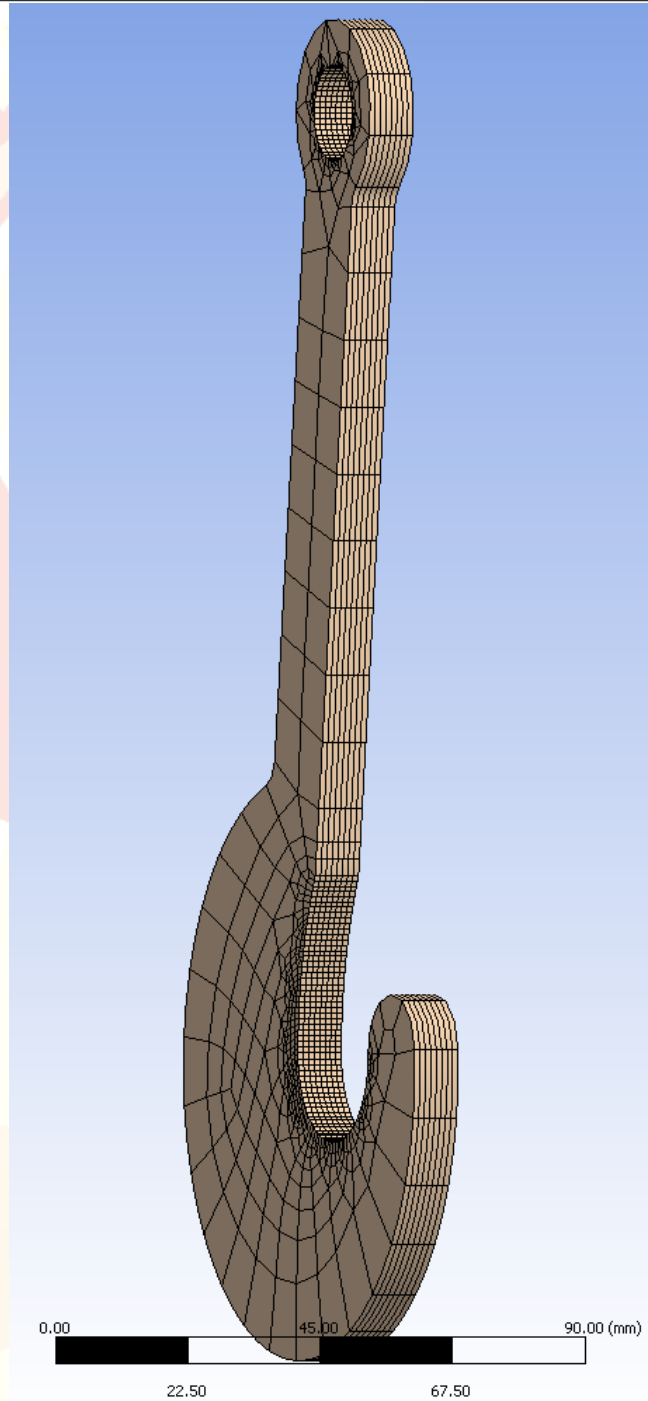
Press F1 for Help

2 Messages No Selection

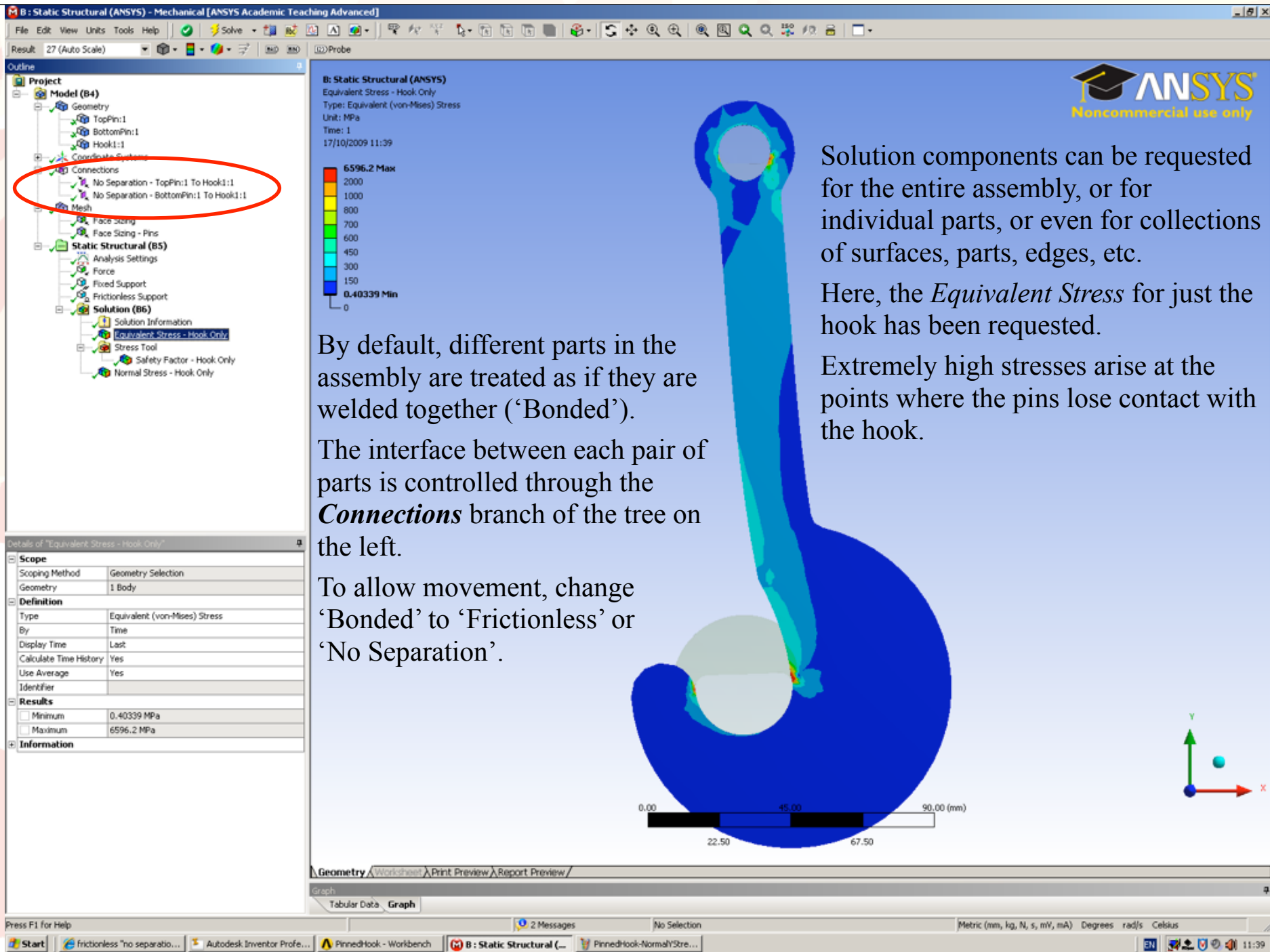
Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Start Frictionless "no separatio..." Autodesk Inventor Profe... PinnedHook - Workbench B: Static Structural (PinnedHook-Mesh-Hook - ...

11:45



Both pins and the hook have mesh *Sizings* of 1mm applied to primary contact faces.



By default, different parts in the assembly are treated as if they are welded together ('Bonded').

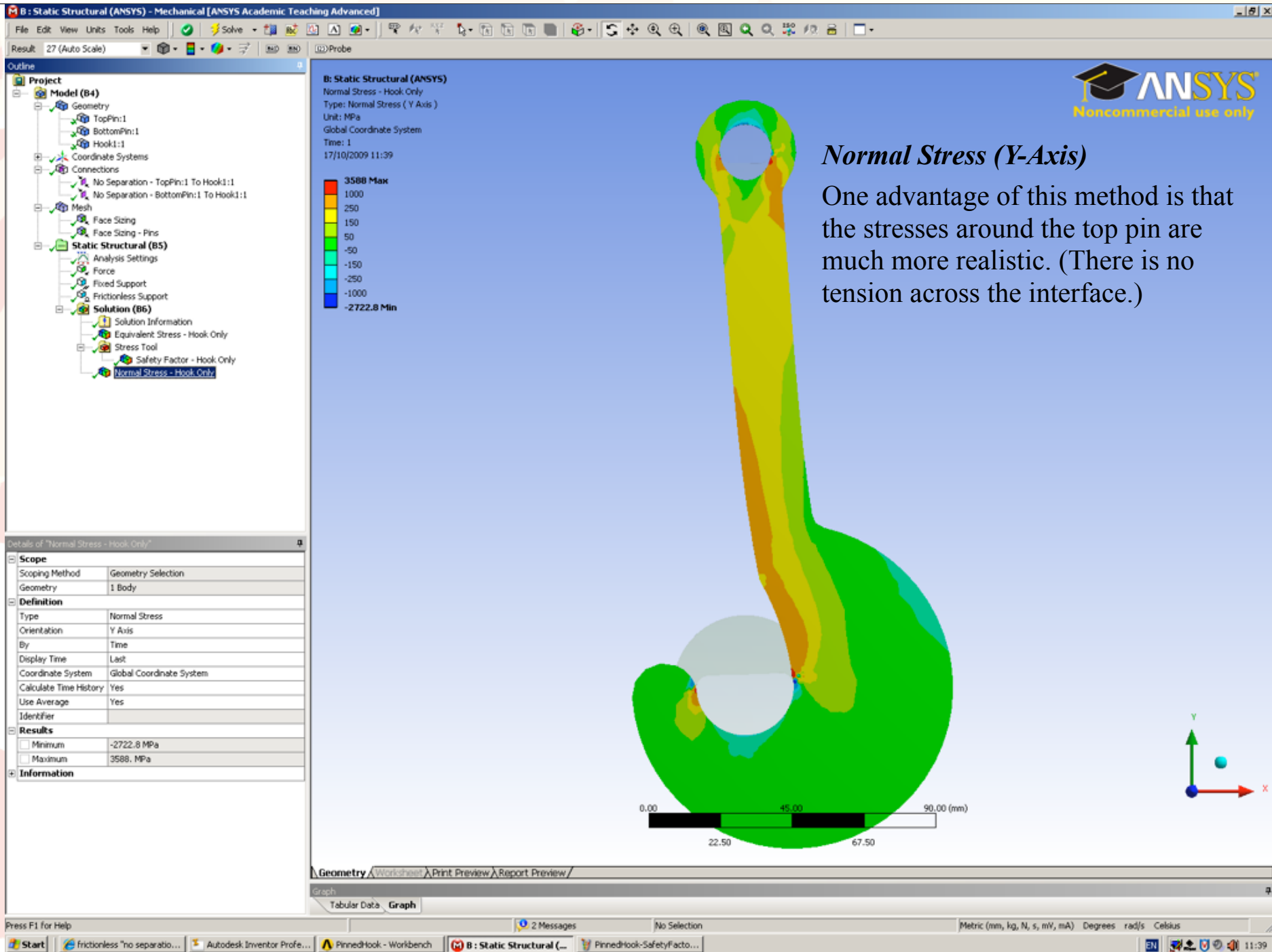
The interface between each pair of parts is controlled through the **Connections** branch of the tree on the left.

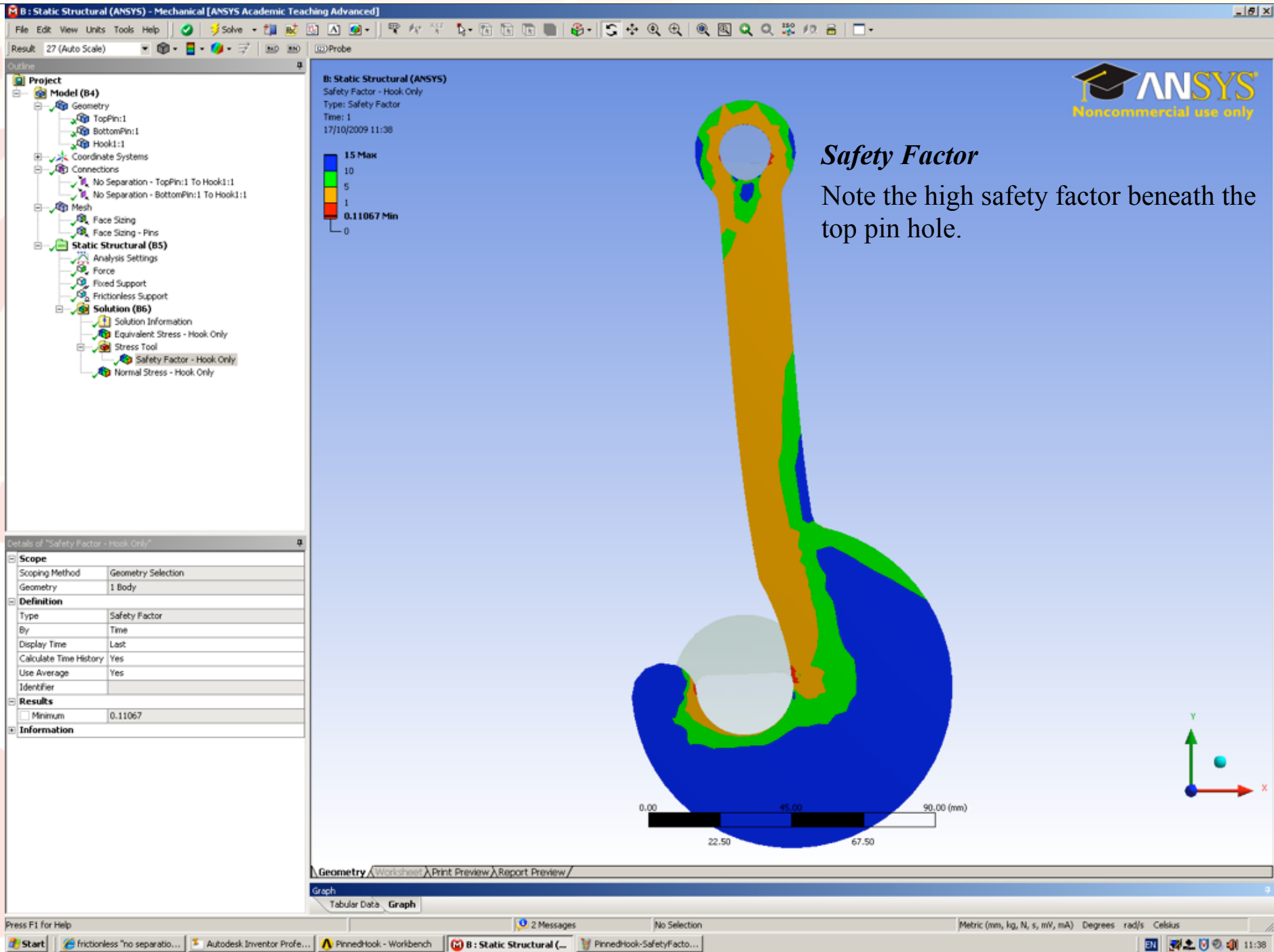
To allow movement, change 'Bonded' to 'Frictionless' or 'No Separation'.

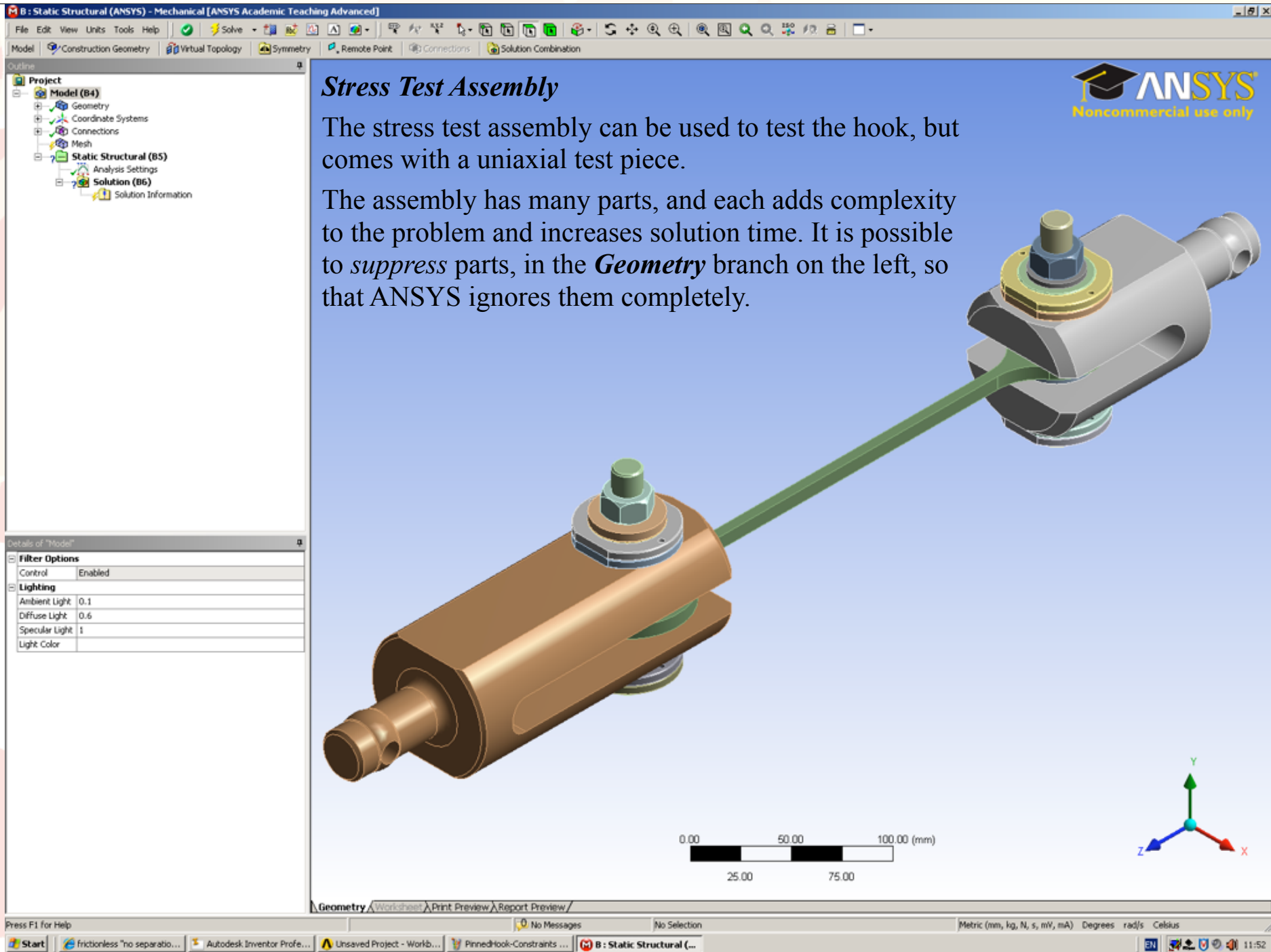
Solution components can be requested for the entire assembly, or for individual parts, or even for collections of surfaces, parts, edges, etc.

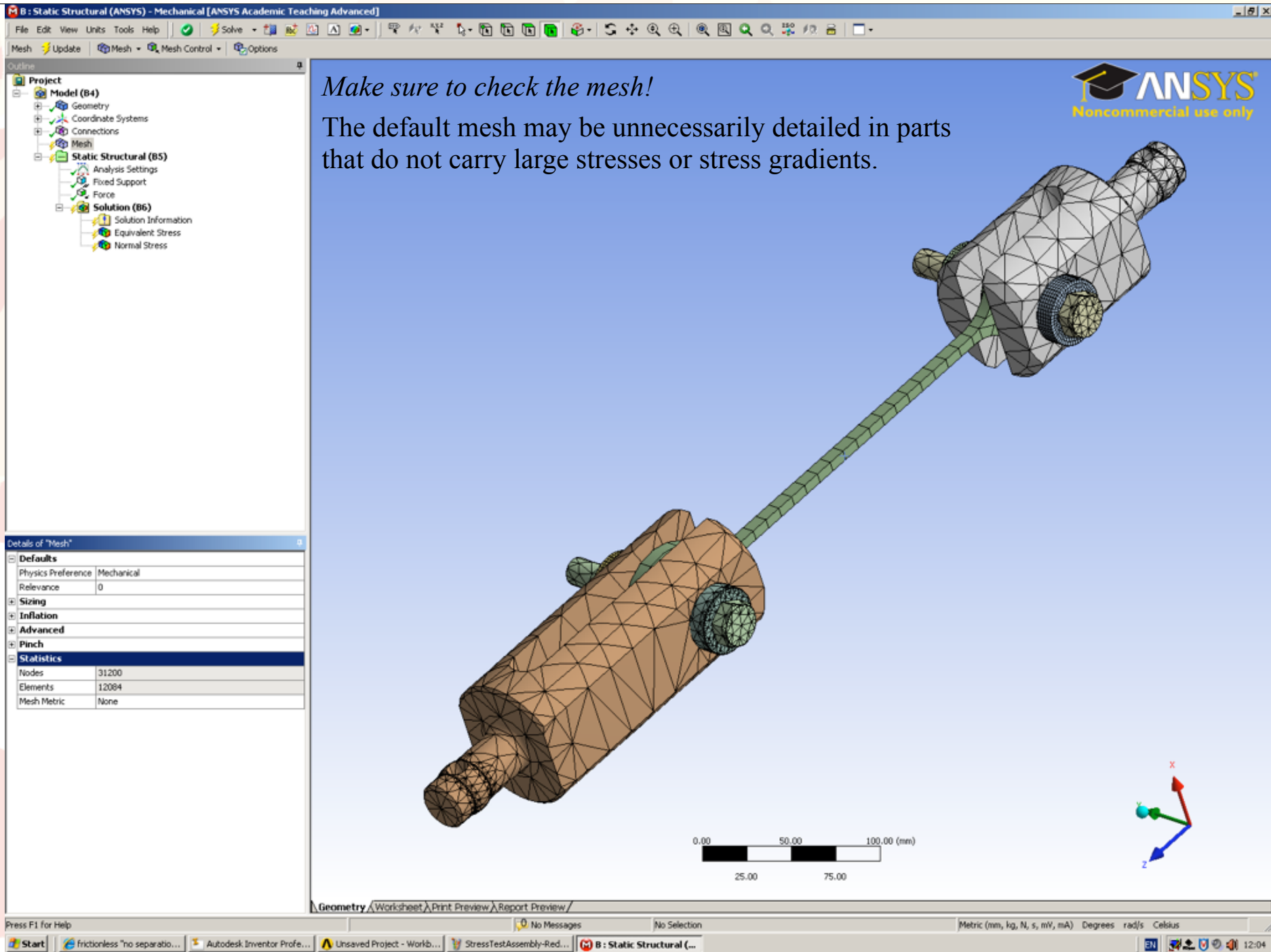
Here, the *Equivalent Stress* for just the hook has been requested.

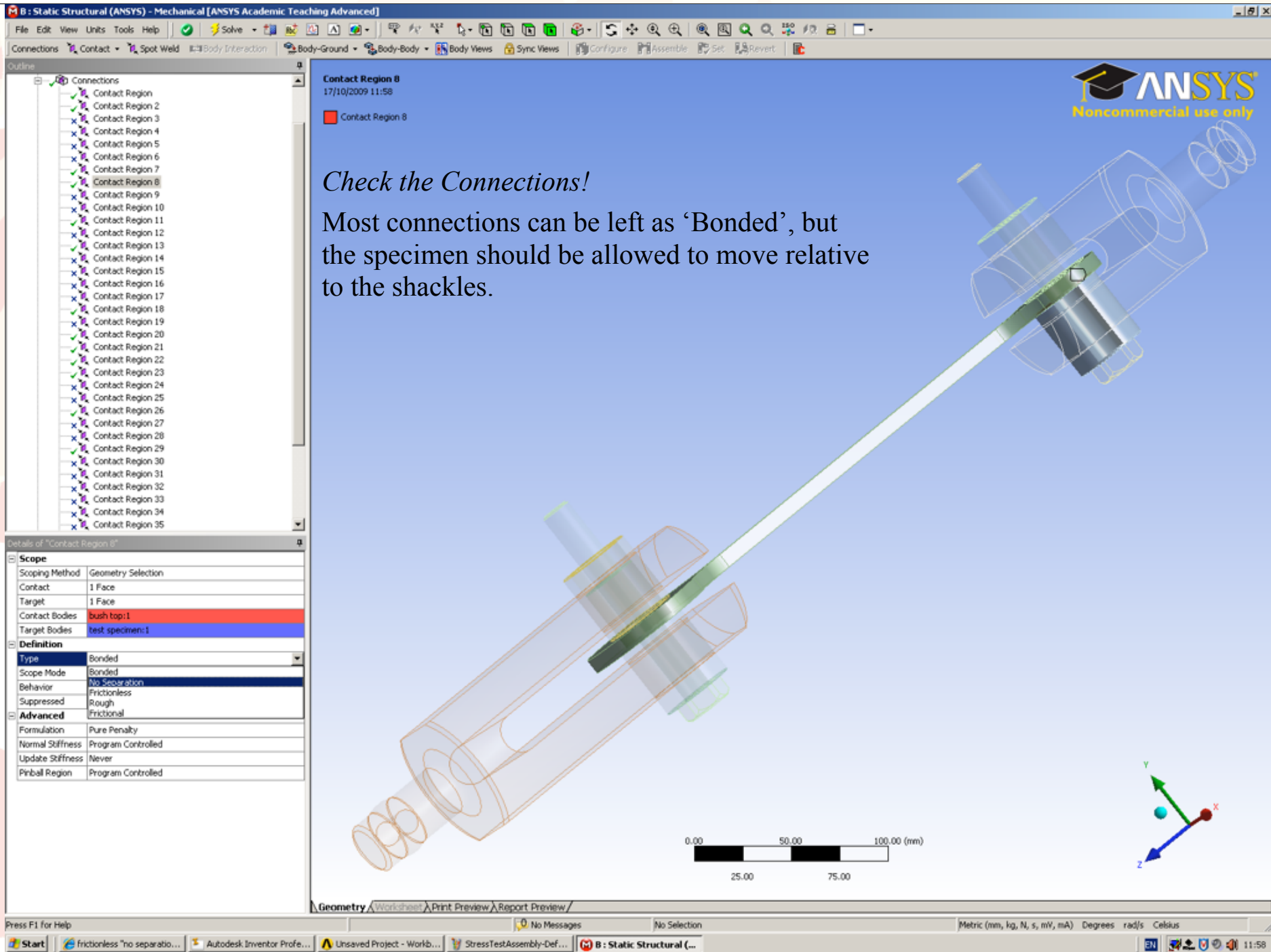
Extremely high stresses arise at the points where the pins lose contact with the hook.











Check the Connections!

Most connections can be left as 'Bonded', but the specimen should be allowed to move relative to the shackles.

