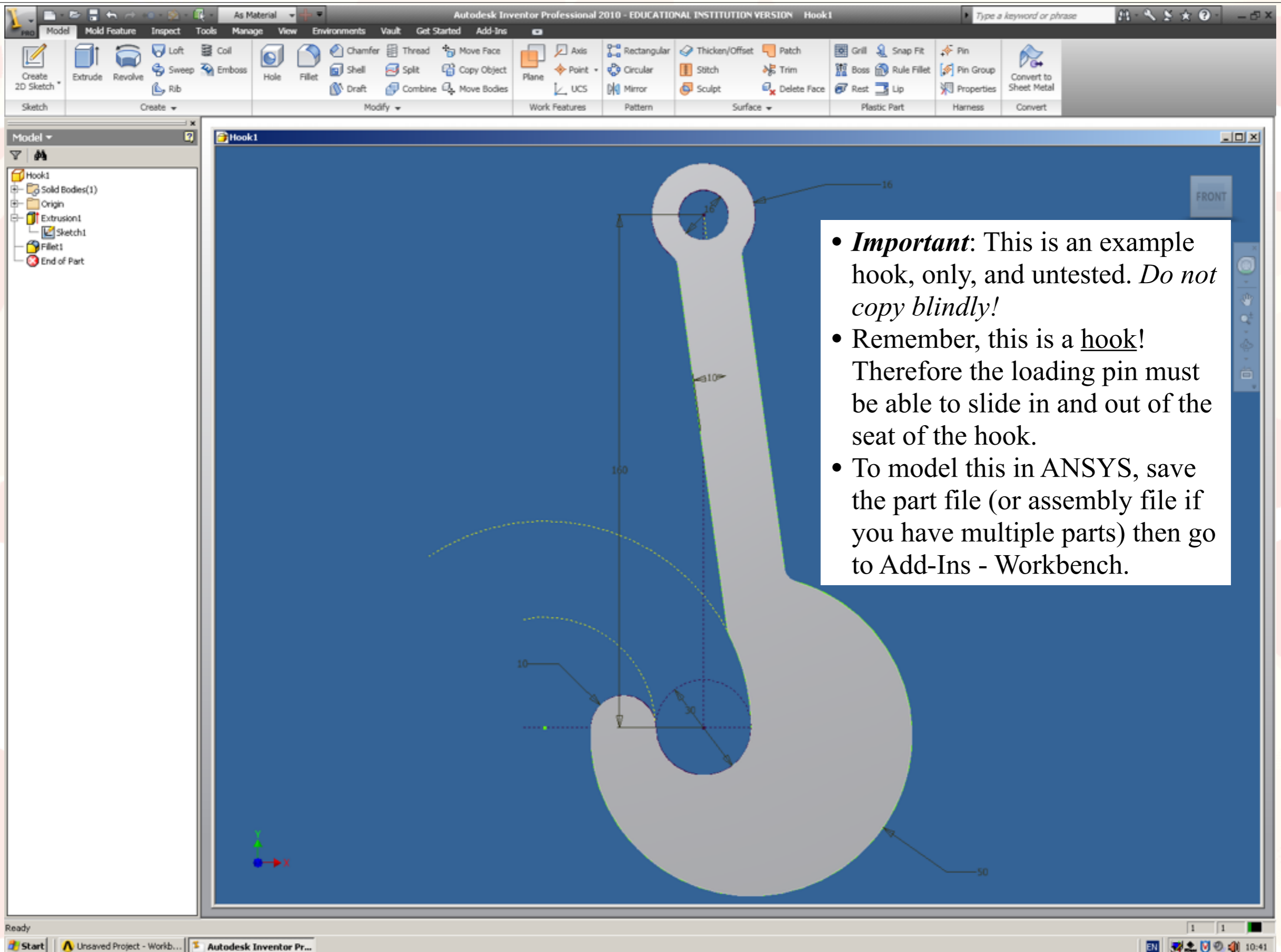




# **ANSYS**

**Dr Francis J. Franklin**

**F.J.Franklin@ncl.ac.uk**



The screenshot displays the ANSYS Workbench interface. The 'Project Schematic' window shows three analysis systems: A (Geometry), B (Static Structural (ANSYS)), and C (Engineering Data). The 'Messages' window at the bottom shows a list of news items with columns for 'Type', 'Text', and 'Date/Time'.

	A	B	C
1	Type	Text	Date/Time
2	ANSYS News	<a href="#">ANSYS and SpaceClaim Streamline Engineering Design and Simulation Driven Product Development</a>	09/09/2009 15:07:05
3	ANSYS News	<a href="#">ANSYS Harnessing the Power of Ocean Waves</a>	02/09/2009 15:06:28
4	ANSYS News	<a href="#">Firehole Releases Version 2.0 of HeliusMCT™</a>	01/09/2009 15:05:53
5	ANSYS News	<a href="#">ANSYS Launches Immersed Boundary Module for Rapid Design Evaluation</a>	31/08/2009 15:05:17
6	ANSYS News	<a href="#">ANSYS and HBM-nCode Partner to Deliver Advanced Fatigue Capabilities</a>	24/08/2009 15:04:28
7	ANSYS News	<a href="#">ANSYS Ranked #33 on FORTUNE 100 Fastest-Growing Companies List</a>	20/08/2009 15:03:22
8	ANSYS News	<a href="#">ANSYS CEO to Present at Kaufman Bros. 12th Annual Investor Conference</a>	19/08/2009 14:59:01
9	ANSYS News	<a href="#">ANSYS Paves the Way for Economic and Environmental Improvements in Cement Manufacturing</a>	10/08/2009 16:10:36
10	ANSYS News	<a href="#">ANSYS, Inc. Reports Second Quarter 2009 Results and Updates Outlook</a>	06/08/2009 16:10:35
11	ANSYS News	<a href="#">Maxwell Technologies and ANSYS Release Ultracapacitor Components Library for Use in Simplorer</a>	04/08/2009 16:10:34
12	ANSYS News	<a href="#">VNIIGAZ Selects ANSYS Multiphysics Software to Drive Innovation in Oil and Gas Pipeline Technology</a>	28/07/2009 16:10:36
13	ANSYS News	<a href="#">AREVA Names ANSYS Certified Supplier</a>	21/07/2009 14:54:47
14	ANSYS News	<a href="#">Delta Marine Steers Ships in the Right Direction Using ANSYS Engineering Simulation</a>	22/07/2009 14:54:07
15	ANSYS News	<a href="#">ANSYS CEO to Present at Oppenheimer Annual Communications, Technology &amp; Internet Conference</a>	15/07/2009 14:53:27
16	ANSYS News	<a href="#">ANSYS, Inc. to Release Second Quarter 2009 Earnings August 6, 2009</a>	13/07/2009 14:52:31
17	ANSYS News	<a href="#">Mining Prospects Enhanced with ANSYS Engineering Simulation</a>	09/07/2009 20:27:10

When ANSYS Workbench starts:

- If there is no Analysis Systems toolbox at all, go to Tools: License Preferences and make sure the Use Academic Licenses button is selected, then restart Workbench.
- If Analysis Systems is there, but there is no Static Structural (ANSYS), then quit ANSYS Workbench and restart.
- 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.



**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, mA)  
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

**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.

Details of "Bearing Load"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face

Definition	
Type	Bearing Load
Define By	Components
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Component	0. N
<input checked="" type="checkbox"/> Y Component	-30000 N
<input type="checkbox"/> Z Component	0. N
Suppressed	No

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 (-

EN 10:45



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

Face Sizing

**Under Mesh:**

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

Details of "Face Sizing" - Sizing	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	3 Faces
<b>Definition</b>	
Suppressed	No
Type	Element Size
Element Size	1. mm
Behavior	Soft

0.00 22.50 45.00 67.50 90.00 (mm)

ANSYS Noncommercial use only



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

File Edit View Units Tools Help

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

ANSYS Noncommercial use only

### Mesh - Preview Surface Mesh

- Make sure you add Sizing constraints wherever the stress gradients are high.
- Remember, however: the more elements, the slower the solution!

Details of "Mesh"	
<b>Defaults</b>	
Physics Preference	Mechanical
Relevance	0
<b>Sizing</b>	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	3.8760 mm
<b>Inflation</b>	
<b>Advanced</b>	
<b>Pinch</b>	
<b>Statistics</b>	

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 (-) | bmp

10:51



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

File Edit View Units Tools Help Solve

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
<input type="checkbox"/> X Component	Vector
<input type="checkbox"/> Y Component	-30000 N
<input type="checkbox"/> 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 (-) bmp Hook1AnsysMeshSizing -...

ANSYS





**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)**  
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!)

Details of "Static Structural (B5)"

Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	ANSYS Mechanical
Options	
<input type="checkbox"/> Environment Temperature	22. °C
<input type="checkbox"/> Generate Input Only	No

0.00 50.00 100.00 (mm)  
25.00 75.00

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 ( tmp Hook1AnsysBearing - Paint EN 10:55



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

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)":

<b>Definition</b>	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	ANSYS Mechanical
<b>Options</b>	
<input type="checkbox"/> Environment Temperature	22. °C
<input type="checkbox"/> Generate Input Only	No

Legend:

- A Fixed Support
- B Frictionless Support
- C Force: 30000 N

3D Model Description: The model shows a hook-like assembly. A vertical force (C) is applied to the bottom pin. The top pin is fixed (A). Frictionless supports (B) are applied to the vertical surfaces of the outer extrusion at the bottom pin.

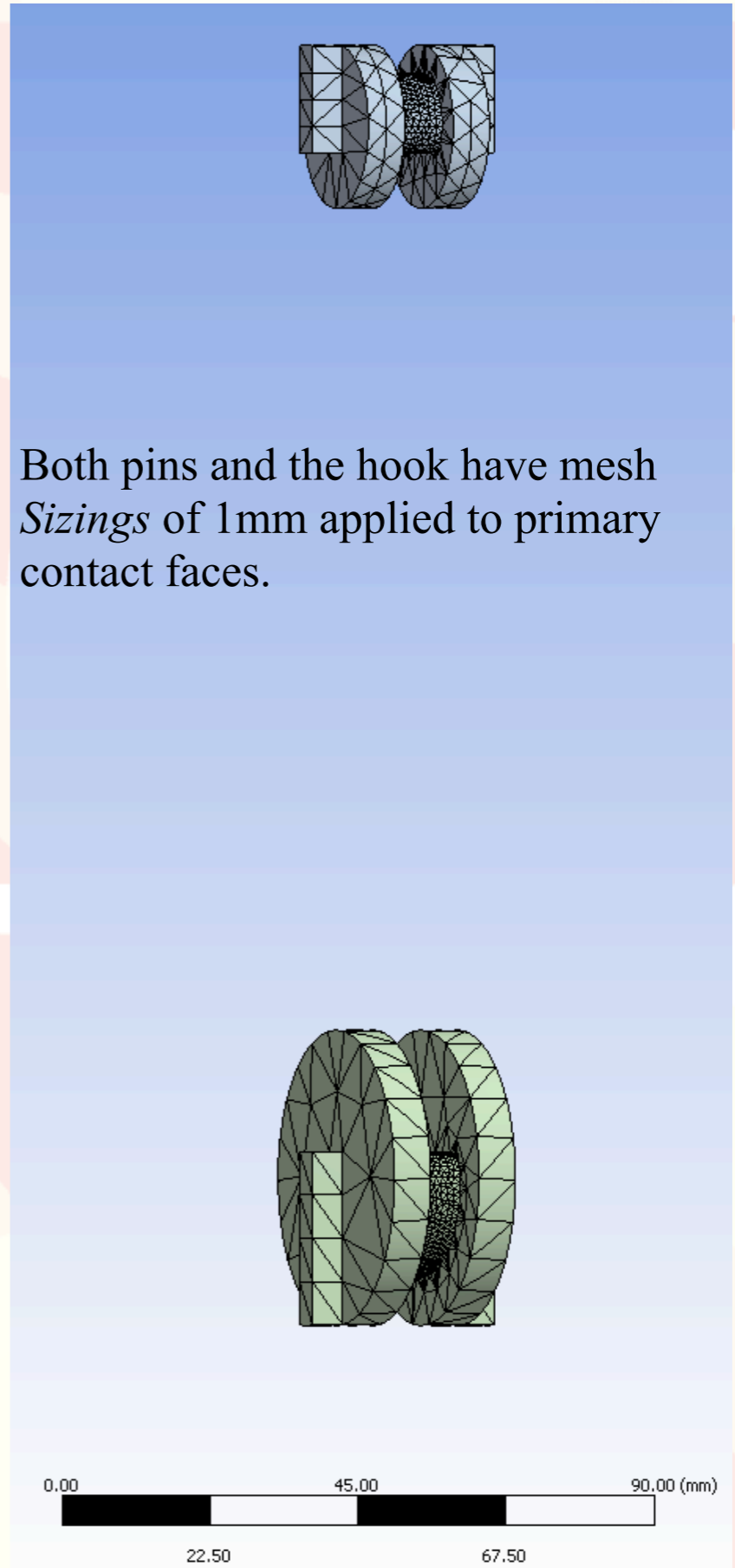
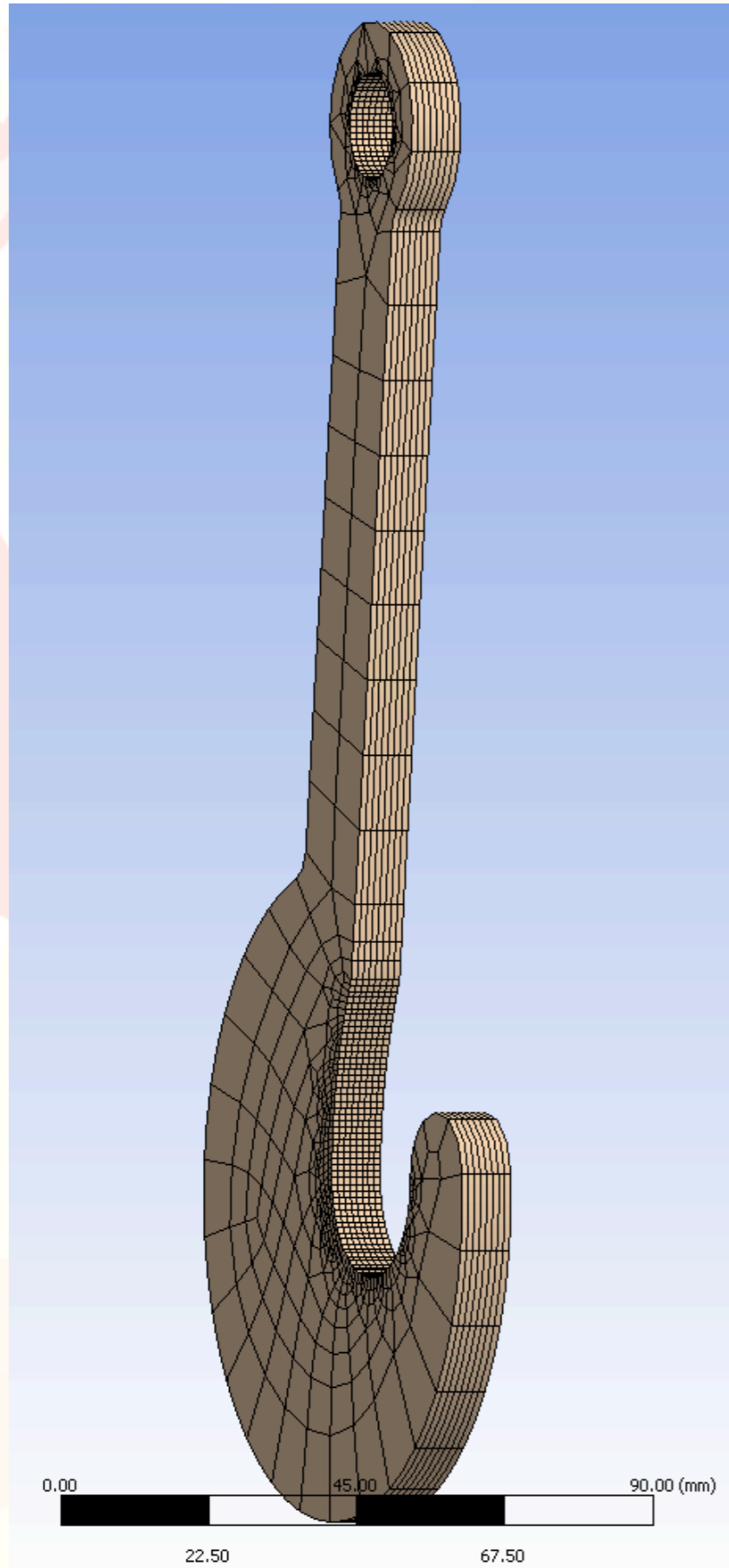
Scale: 0.00 to 60.00 (mm). Markers at 15.00, 30.00, 45.00.

ANSYS Noncommercial use only

- *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.

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.



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

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

Result: 27 (Auto Scale) | Probe

**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

**B: Static Structural (ANSYS)**  
Equivalent Stress - Hook Only  
Type: Equivalent (von-Mises) Stress  
Unit: MPa  
Time: 1  
17/10/2009 11:39

6596.2 Max  
2000  
1000  
800  
700  
600  
450  
300  
150  
0.40339 Min  
0

**Details of "Equivalent Stress - Hook Only"**

<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Body
<b>Definition</b>	
Type	Equivalent (von-Mises) Stress
By	Time
Display Time	Last
Calculate Time History	Yes
Use Average	Yes
Identifier	
<b>Results</b>	
Minimum	0.40339 MPa
Maximum	6596.2 MPa
<b>Information</b>	

**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 | 11:39

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.



**B: Static Structural (ANSYS)**  
Normal Stress - Hook Only  
Type: Normal Stress ( Y Axis )  
Unit: MPa  
Global Coordinate System  
Time: 1  
17/10/2009 11:39

**3588 Max**  
1000  
250  
150  
50  
-50  
-150  
-250  
-1000  
**-2722.8 Min**

**Normal Stress (Y-Axis)**  
One advantage of this method is that the stresses around the top pin are much more realistic. (There is no tension across the interface.)

Details of "Normal Stress - Hook Only"	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Body
<b>Definition</b>	
Type	Normal Stress
Orientation	Y Axis
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Use Average	Yes
Identifier	
<b>Results</b>	
<input type="checkbox"/> Minimum	-2722.8 MPa
<input type="checkbox"/> Maximum	3588. MPa
<b>Information</b>	



**B: Static Structural (ANSYS)**  
Safety Factor - Hook Only  
Type: Safety Factor  
Time: 1  
17/10/2009 11:38

**15 Max**  
10  
5  
1  
**0.11067 Min**  
0

**Safety Factor**  
Note the high safety factor beneath the top pin hole.

Details of "Safety Factor - Hook Only"	
<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Body
<b>Definition</b>	
Type	Safety Factor
By	Time
Display Time	Last
Calculate Time History	Yes
Use Average	Yes
Identifier	
<b>Results</b>	
<input type="checkbox"/> Minimum	0.11067
<b>Information</b>	

0.00 22.50 45.00 67.50 90.00 (mm)

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 11:38



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

File Edit View Units Tools Help Solve Virtual Topology Symmetry Remote Point Connections Solution Combination

Outline

- Project
  - Model (B4)
    - Geometry
    - Coordinate Systems
    - Connections
    - Mesh
    - Static Structural (B5)
      - Analysis Settings
      - Solution (B6)
        - Solution Information

Details of "Model"

Filter Options	
Control	Enabled
Lighting	
Ambient Light	0.1
Diffuse Light	0.6
Specular Light	1
Light Color	

**Stress Test Assembly**

The stress test assembly can be used to test the hook, but comes with a uniaxial test piece.

The assembly has many parts, and each adds complexity to the problem and increases solution time. It is possible to *suppress* parts, in the **Geometry** branch on the left, so that ANSYS ignores them completely.

ANSYS  
Noncommercial use only

0.00 25.00 50.00 75.00 100.00 (mm)

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 frictionless "no separatio... Autodesk Inventor Profe... Unsaved Project - Workb... PinnedHook-Constraints ... B: Static Structural (...)



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
    - Connections
    - Mesh
  - Static Structural (B5)
    - Analysis Settings
    - Fixed Support
    - Force
  - Solution (B6)
    - Solution Information
    - Equivalent Stress
    - Normal Stress

Details of "Mesh"

Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Inflation	
Advanced	
Pinch	
Statistics	
Nodes	31200
Elements	12084
Mesh Metric	None

ANSYS Noncommercial use only

Make sure to check the mesh!

The default mesh may be unnecessarily detailed in parts that do not carry large stresses or stress gradients.

0.00 25.00 50.00 75.00 100.00 (mm)

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 frictionless \*no separatio... Autodesk Inventor Profe... Unsaved Project - Workb... StressTestAssembly-Red... B: Static Structural (...)

EN 12:04



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

File Edit View Units Tools Help

Connections Contact Spot Weld Body Interaction Body-Ground Body-Body Body Views Sync Views Configure Assemble Set Revert

Outline

- Connections
  - Contact Region
  - Contact Region 2
  - Contact Region 3
  - Contact Region 4
  - Contact Region 5
  - Contact Region 6
  - Contact Region 7
  - Contact Region 8
  - Contact Region 9
  - Contact Region 10
  - Contact Region 11
  - Contact Region 12
  - Contact Region 13
  - Contact Region 14
  - Contact Region 15
  - Contact Region 16
  - Contact Region 17
  - Contact Region 18
  - Contact Region 19
  - Contact Region 20
  - Contact Region 21
  - Contact Region 22
  - Contact Region 23
  - Contact Region 24
  - Contact Region 25
  - Contact Region 26
  - Contact Region 27
  - Contact Region 28
  - Contact Region 29
  - Contact Region 30
  - Contact Region 31
  - Contact Region 32
  - Contact Region 33
  - Contact Region 34
  - Contact Region 35

Details of "Contact Region 8"

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	bush top:1
Target Bodies	test specimen:1
Definition	
Type	Bonded
Scope Mode	Bonded
Behavior	No Separation
Suppressed	Frictionless
Advanced	Frictional
Formulation	Pure Penalty
Normal Stiffness	Program Controlled
Update Stiffness	Never
Pinball Region	Program Controlled

Contact Region 8  
17/10/2009 11:58

Contact Region 8

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

ANSYS  
Noncommercial use only

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 frictionless 'no separatio... Autodesk Inventor Profe... Unsaved Project - Workb... StressTestAssembly-Def... B: Static Structural (...)

EN 11:58



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

File Edit View Units Tools Help

Result: 1.2 (Auto Scale)

Outline

- Project
  - Model (B4)
    - Geometry
    - Coordinate Systems
    - Connections
    - Mesh
      - Face Sizing
    - Static Structural (B5)
      - Analysis Settings
      - Fixed Support
      - Force
    - Solution (B6)
      - Solution Information
      - Equivalent Stress - Test Specimen
      - Normal Stress - Test Specimen

B: Static Structural (ANSYS)  
Normal Stress - Test Specimen  
Type: Normal Stress (Z Axis)  
Unit: MPa  
Global Coordinate System  
Time: 1  
17/10/2009 12:14

212.94 Max  
210  
208  
174.33  
140.66  
106.99  
73.322  
39.653  
5.9831  
-27.686 Min

ANSYS  
Noncommercial use only

Hmm...

Details of "Normal Stress - Test Specimen"

<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Body
<b>Definition</b>	
Type	Normal Stress
Orientation	Z Axis
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Use Average	Yes
Identifier	
<b>Results</b>	
<input type="checkbox"/> Minimum	-27.686 MPa
<input type="checkbox"/> Maximum	212.94 MPa
<b>Information</b>	

Geometry Worksheet Print Preview Report Preview

Graph

Tabular Data Graph

Press F1 for Help

3 Messages No Selection

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

Start frictionless "no separatio... Autodesk Inventor Profe... Unsaved Project - Workb... StressTestAssembly-Mes... B: Static Structural (...)

EN 12:15



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

File Edit View Units Tools Help Solve

Result: 1.2 (Auto Scale) Probe

Outline

- Project
  - Model (B4)
    - Geometry
    - Coordinate Systems
    - Connections
    - Mesh
      - Face Sizing
  - Static Structural (B5)
    - Analysis Settings
    - Fixed Support
    - Force
  - Solution (B6)
    - Solution Information
    - Equivalent Stress - Test Specimen
    - Normal Stress - Test Specimen
    - Directional Deformation

**B: Static Structural (ANSYS)**  
Directional Deformation  
Type: Directional Deformation ( X Axis )  
Unit: mm  
Global Coordinate System  
Time: 1  
17/10/2009 12:25

26.492 Max  
20.603  
14.714  
8.8255  
2.9365  
-2.9524  
-8.8414  
-14.73  
-20.619  
-26.508 Min

Details of "Directional Deformation"

<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	1 Body
<b>Definition</b>	
Type	Directional Deformation
Orientation	X Axis
By	Time
Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Identifier	
<b>Results</b>	
<input type="checkbox"/> Minimum	-26.508 mm
<input type="checkbox"/> Maximum	26.492 mm
<b>Information</b>	

Geometry Worksheet Print Preview Report Preview

Graph

Tabular Data Graph

Press F1 for Help 3 Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Start frictionless "no separatio... Autodesk Inventor Profe... Unsaved Project - Workb... StressTestAssembly-Nor... B: Static Structural (...)

EN 12:25