An introduction to Finite Element Analysis (FEA), utilising ANSYS Workbench

Tutorial 4: Model development

Dr David Wynn

Email: d.wynn@auckland.ac.nz

Room: 401.916

This tutorial

- The importance of careful selection of loads and constraints.

- Tutorial exercise on load/constraints selection.

- Introduction to mesh refinement. Why is it necessary?

- Tutorial exercise on mesh refinement.

Reminders

- **DO NOT** save your project on the Desktop or in My Documents. **If you do this it may not solve**.

- **DO** save your project on D:Scratch. It will run, efficiently.

- For a 2D analysis, **YOU MUST** select a 2D analysis at the start of your work.

- For a 2D analysis, don't forget to set the thickness of the part. The default is 1.0m thick!
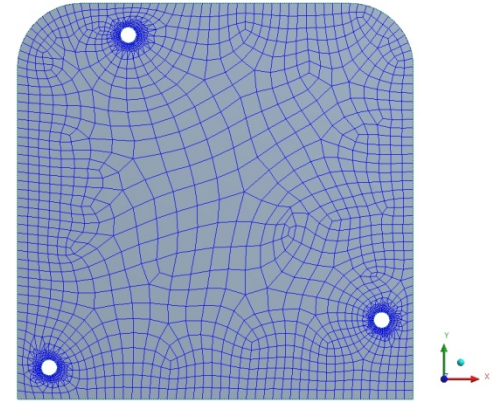
Exercise A: Loads and constraints

Recall the four basic requirements for a physical model:

a) Physics / Equations? ⇨ **Linear Elastic Structures**
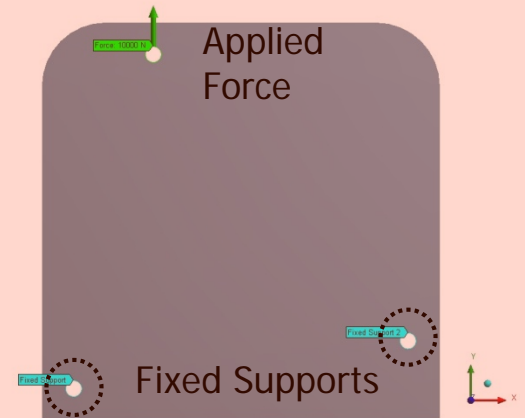
b) Geometry?

Approximated by a mesh of elements

c) Material Properties?
- Young's Modulus
- Poisson's Ratio
- Yield Stress
- Tensile Strength
- others…?

Applied Force

d) Loads / Constraints (B.C.s)

Fixed Supports

Solve…

Loads and constraints: Why you MUST be careful

Often it is selection of loads/constraints that determines whether an FEA model is:

- An appropriate approximation to reality.
- OR, a collection of worthless pretty pictures and unrealistic predictions.

Many errors are caused by poor application of loads/constraints.

We may choose to simplify a real situation in different ways:

CONSTRAINTS

Built-in?

or, Simply Supported?

LOADS

P

Point Load?

w

or, Distributed?
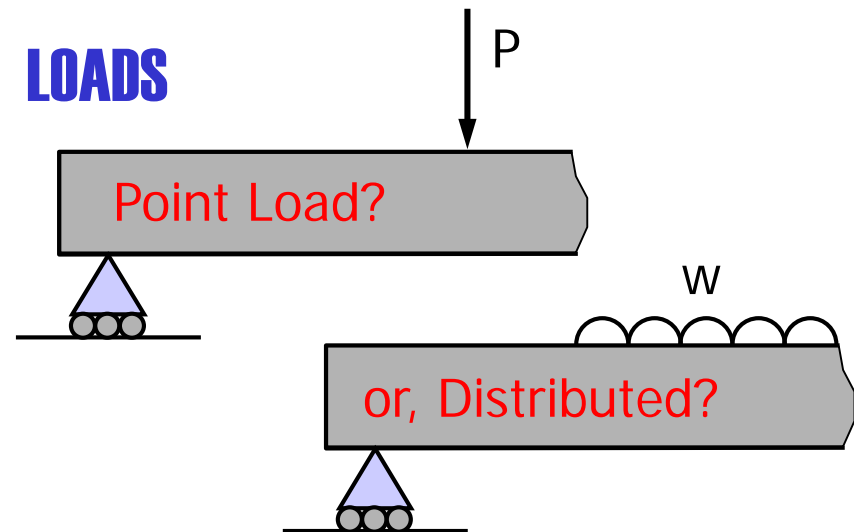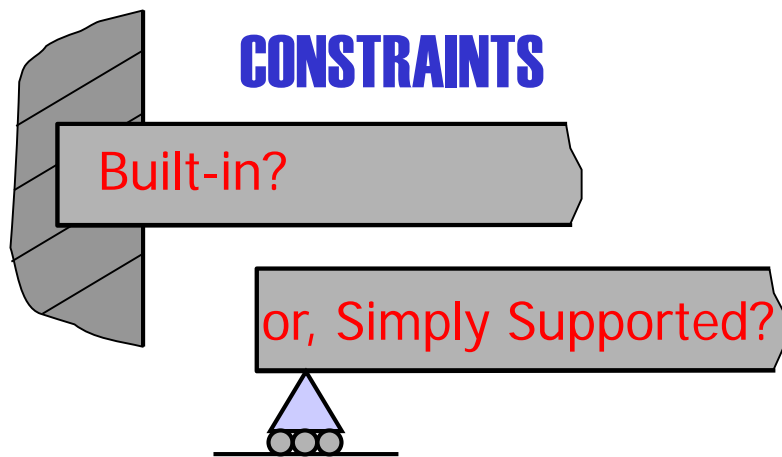
Points to note before starting work

Load/constraint selection is **not always easy**, and there **may not always be a clearly correct solution**.

- You need to think carefully about what is happening in reality. How can you **best approximate** this?

- You will need to gain experience, and learn about the various load/constraint options in Workbench.

The following exercise will **introduce you to some of the options** in Workbench. You will need to learn more...

- Go to ANSYS help...
    - Mechanical Users Guide
        - Features
            - Applying Loads
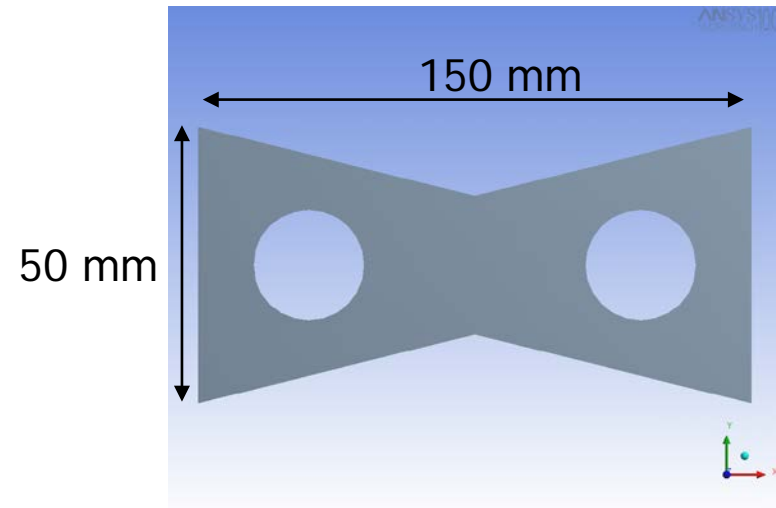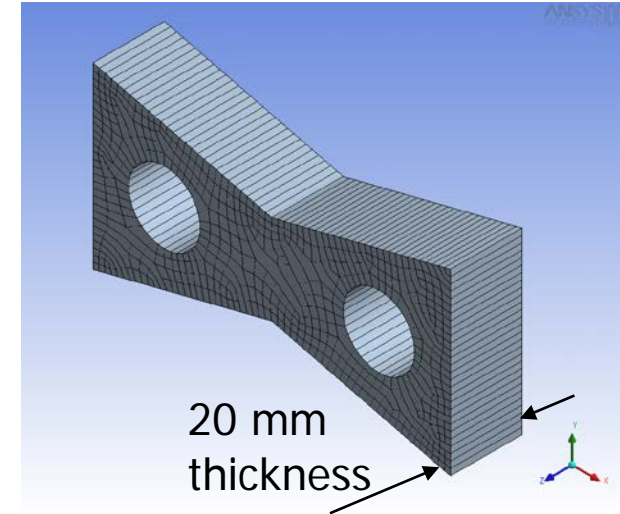   - Types of Loads   **OR**   • Types of Supports

Exercise: Loads and constraints in 2D

We'll consider a simple 2D component, which will demonstrate sensitivity to load/constraint selection.

The component is supported by two rigid rails, matching its shape. It is slid into position.
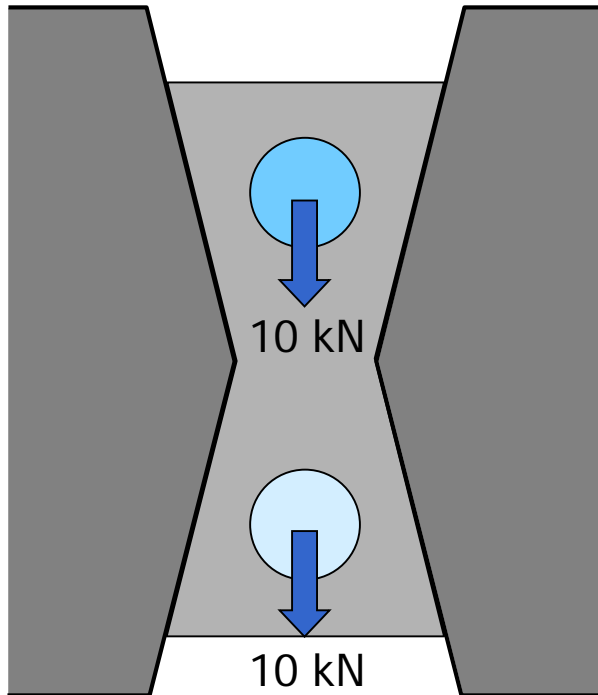

Pins apply 10 kN forces vertically to each hole.

10 kN

10 kN

20 mm thickness

150 mm

50 mm

Getting started

• Launch **"ANSYS Workbench".** Select a "Static Structural" Analysis (Drag and drop into "Project Schematic" box or double-click)

Don't forget to configure for 2D analysis:

• Right Click on step 3 of the analysis (Geometry)

• Select "Properties"

• Change analysis type to 2D

• Import the geometry…

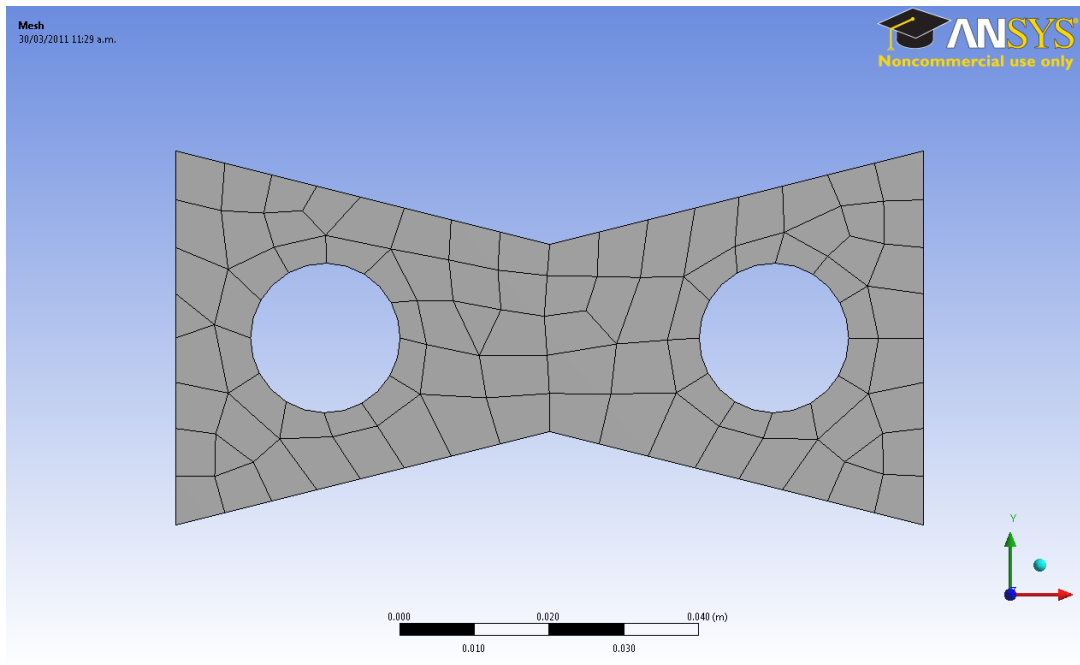• Right Click on step 3 again

• Click Import geometry…Browse…
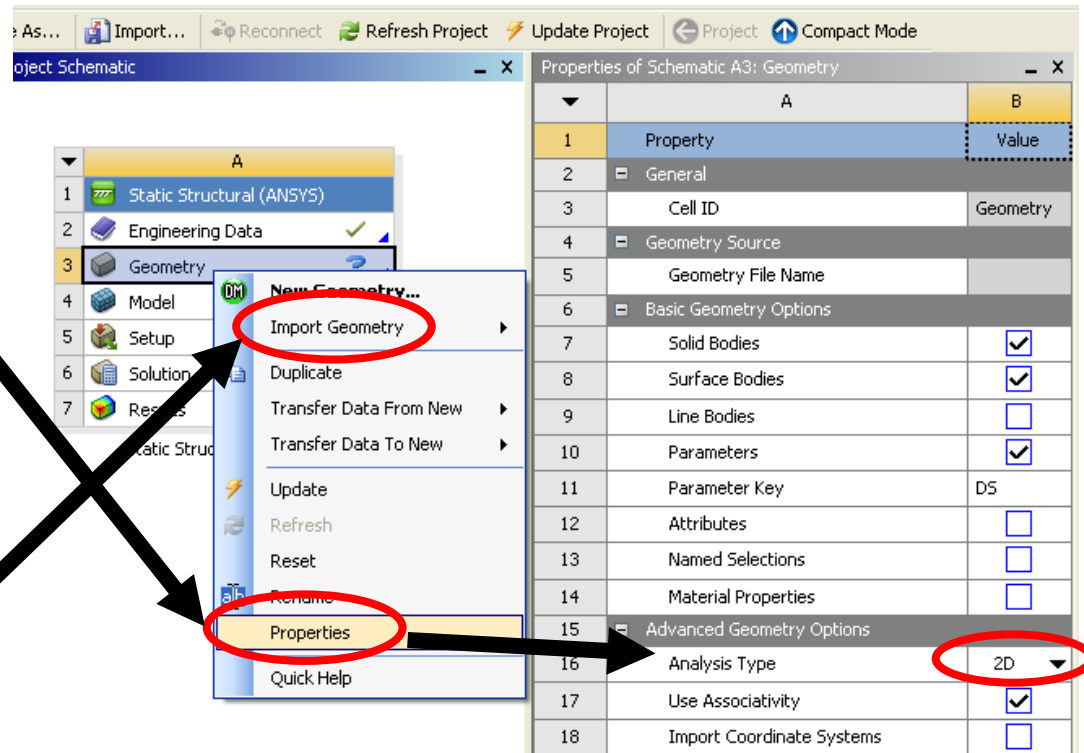
Where you have saved it   …/Tutorial3/AngledPart.agdb

In your own time

- **Special step for 2D analyses.** Specify part thickness in
    - Expand Geometry in the "Outline" box
    - Click on the Part
    - In the "Details" box, expand "Definition" and specify Thickness
- Right click on Mesh in the Outline... Generate Mesh

Don't forget to save your project locally or it may not solve!

In your own time

Application of the first set of loads/constraints

- Consider the conditions below

- Click on the Environment (Static Structural) in the Outline

- Select the Geometric Entity that you want to apply a condition to. Select the condition from the structural dropdown menu.
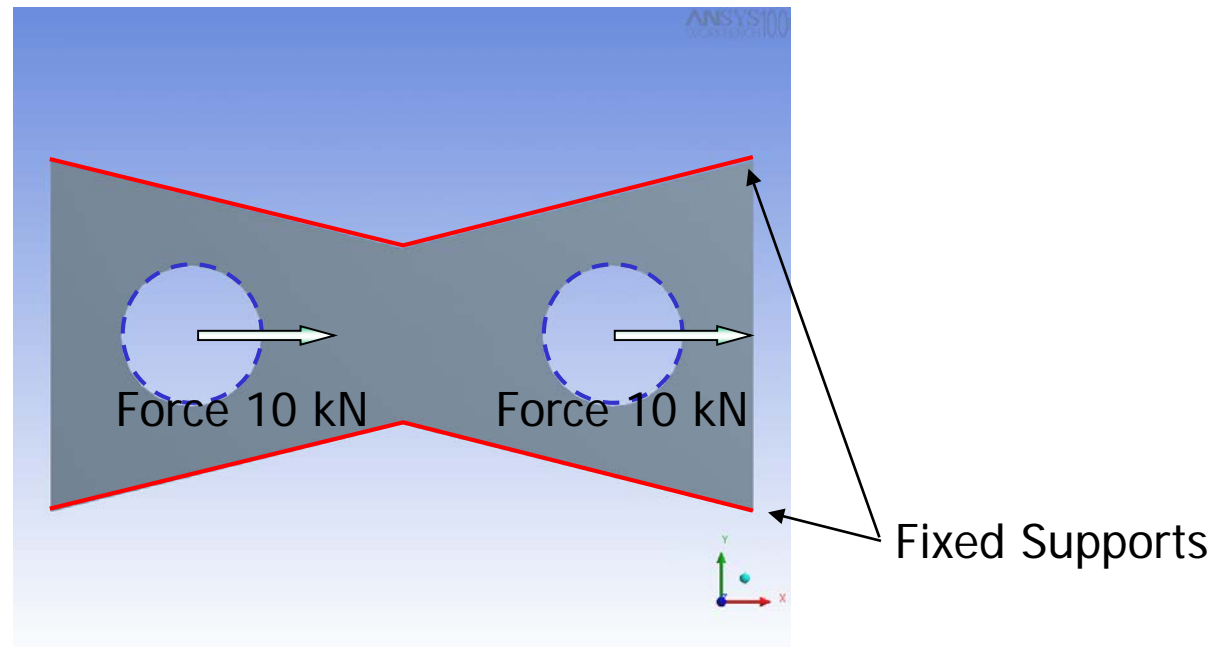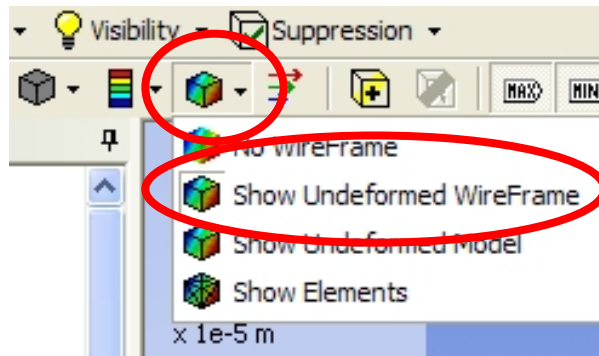

Force 10 kN
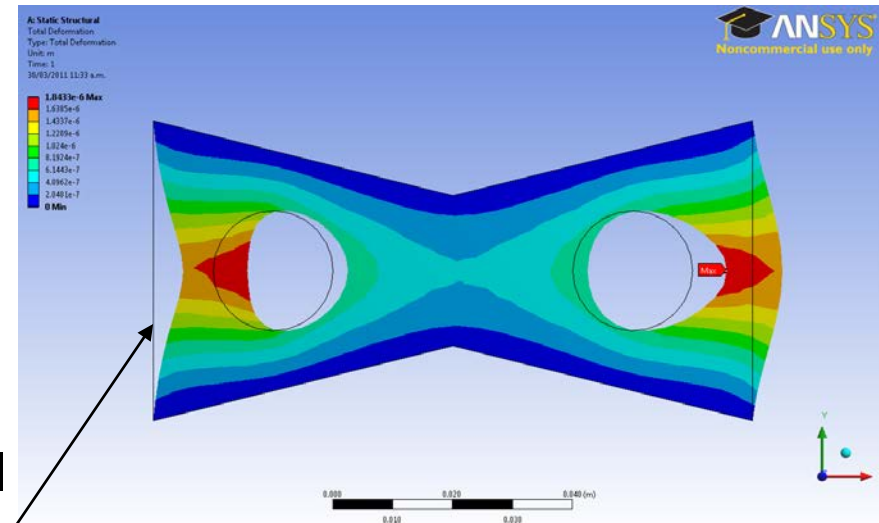
Force 10 kN

Fixed Supports

In your own time

- Establishing results. The **Solution**...

  - Click on Solution in the Outline

  - From the Solution bar, choose the following results:

    1) Total Deformation
    2) Equivalent Stress

The model is now ready to solve. Hit **solve**...

To show original shape, choose "show undeformed wireframe"

Creating Multiple Load/Constraint Cases

We'll now explore the results produced by applying a variety of **different constraints**. The best way to do this is to make several duplicates of the existing simulation.

• From the Workbench window , right click on step 5 of the simulation, "Setup", and select "Duplicate". Notice that steps 2-4 of the duplicates are linked.

• Rename the duplicate simulation "Force_Fixed".

• Create additional duplicates named "Force_Frictionless" and "Force_CompOnly".

Right Click 'Setup' to duplicate

Click here to rename
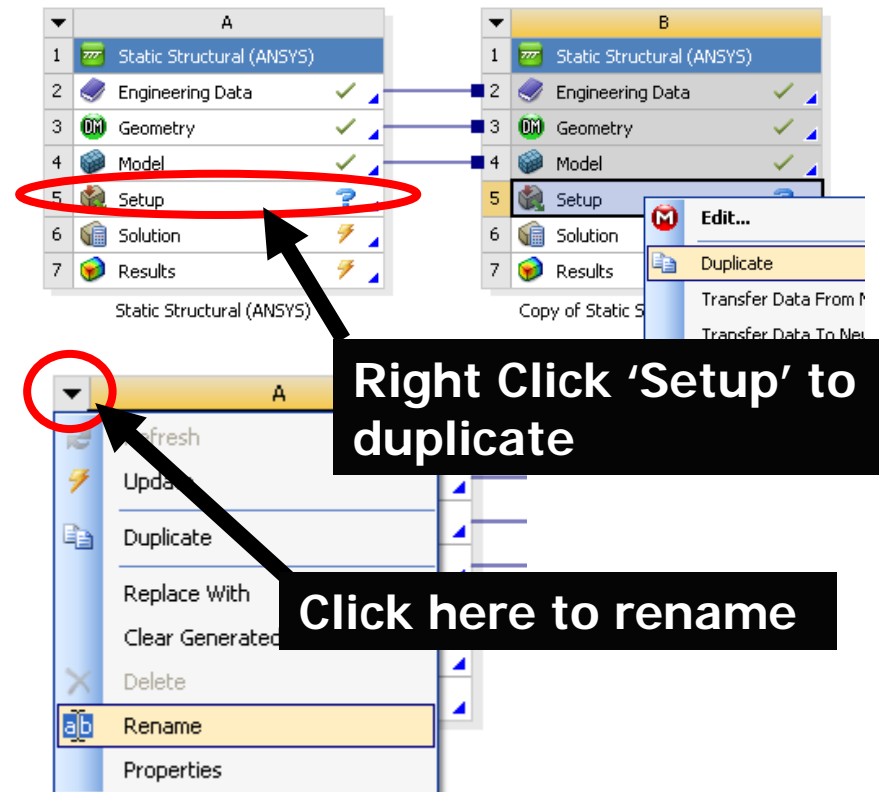
In your own time

Modify the **"Force_Frictionless"** environment in ⬤.

- Right click on the Fixed Support condition, and delete it.

- Introduce a new condition at the upper and lower surfaces, applying a Frictionless Support.

- Solve

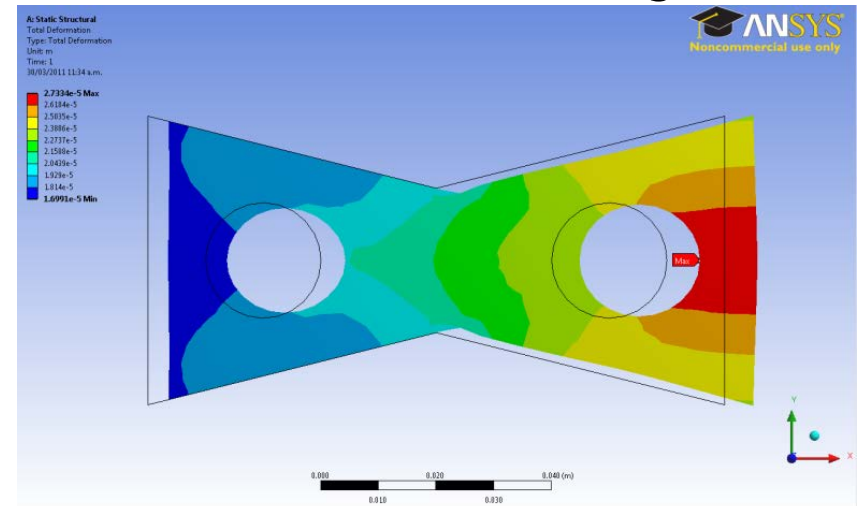Modify the **"Force_CompOnly"** environment in ⬤.

- Right click on the Fixed Support condition, and delete

- Introduce a new condition at the upper and lower surfaces, applying a Compression Only Support.

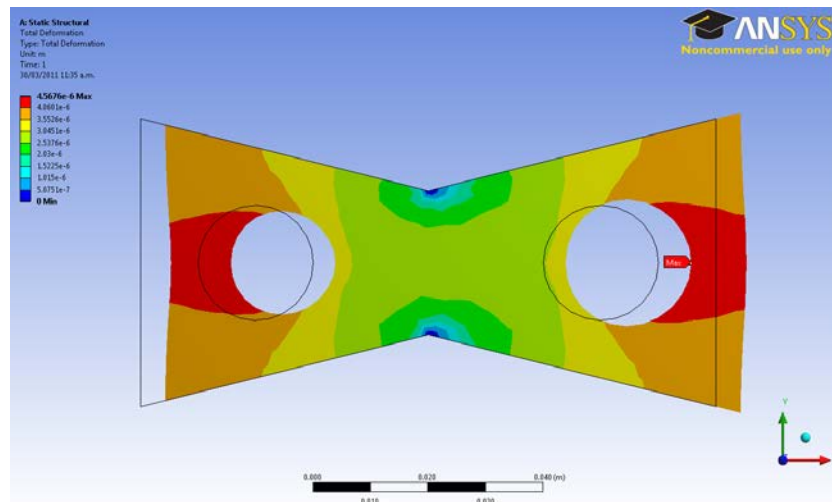- Solve

In your own time

The different deformation results should look something like...
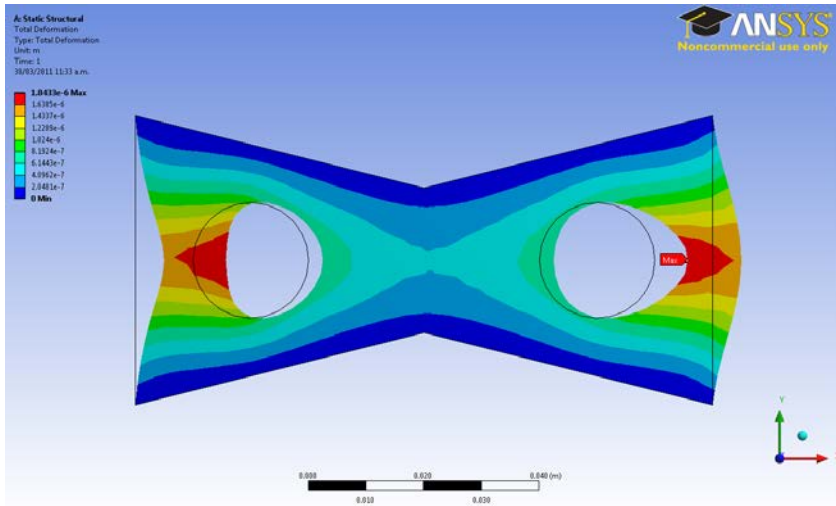
Fixed Supports

Compression Only

Frictionless Supports

Comparing Results

Take some time to compare the results.

Compare the deformations:

- Maximum values?

- How do deformations compare near the walls?

- How have the circular holes deformed in each case?

Compare the equivalent stress results:

- Maximum values? Where are the maximum values?

- How do the stress contours compare?

In your own time

We'll now explore the results produced by applying a variety of **loading conditions**. First, make more duplicates of the "Force_CompOnly" Simulation from the Workbench window.

- Create a duplicate of the "Force_CompOnly" simulation, and call this one "Bearing_CompOnly".

- Create another duplicate of the "Force_CompOnly" simulation, and call this one "Displacement_CompOnly".

In your own time

Modify the **"Bearing_CompOnly"** environment in [M].

- Right click on the 2 Force conditions, and delete

- Introduce a new condition at each hole, applying a bearing load of 10kN in the X direction.

- Solve
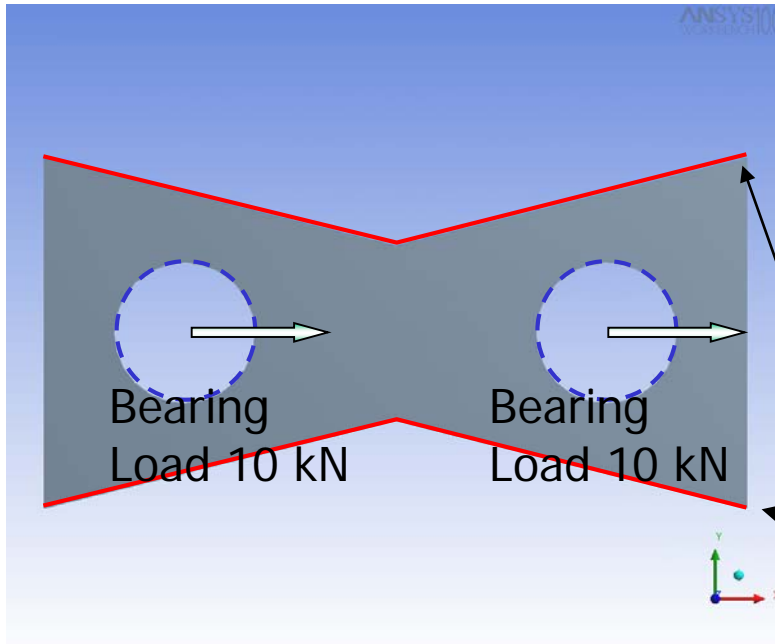
For an explanation of what a bearing load is, see ANSYS help:
- Mechanical Users Guide
  - Features
    - Applying Loads
      - Types of Loads
        - Bearing Load

Bearing
Load 10 kN

Bearing
Load 10 kN

Compression
Only Supports

In your own time

Modify the **"Displacement_CompOnly"** environment.

- Right click on the 2 Force conditions, and delete

- Introduce a new condition at each hole, applying a horizontal displacement of 3.0e-5 m, to the right.

- Solve
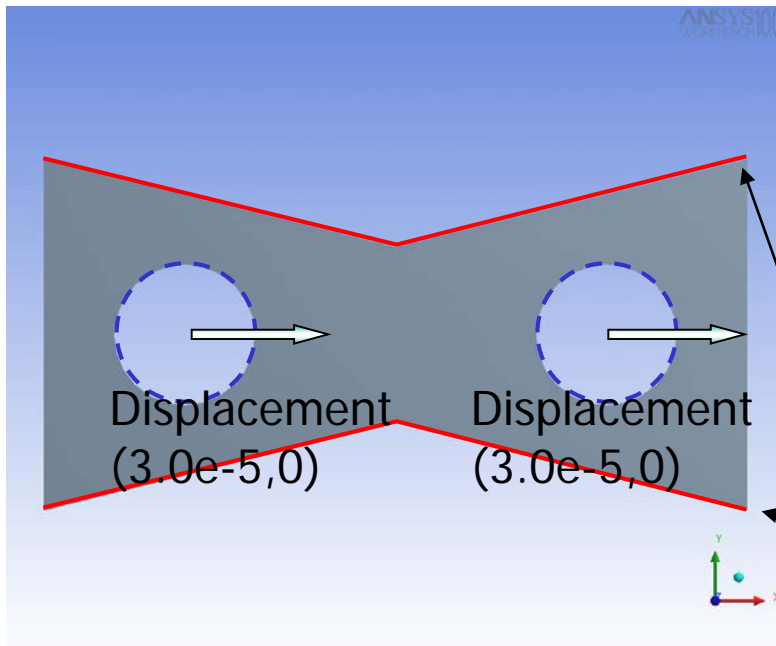
This will displace every point on the circles 3.0e-5 m to the right.

If you have time, leave the Y component of displacement as blank (not zero). What difference does it make?

Displacement (3.0e-5,0)   Displacement (3.0e-5,0)

Compression Only Supports

In your own time

The different deformation results should look something like:

Applied Forces

Applied Displacement

Bearing Loads
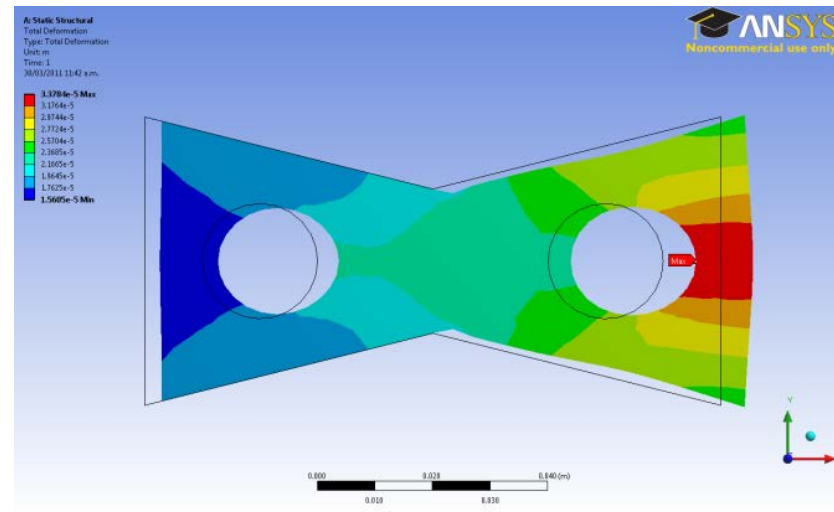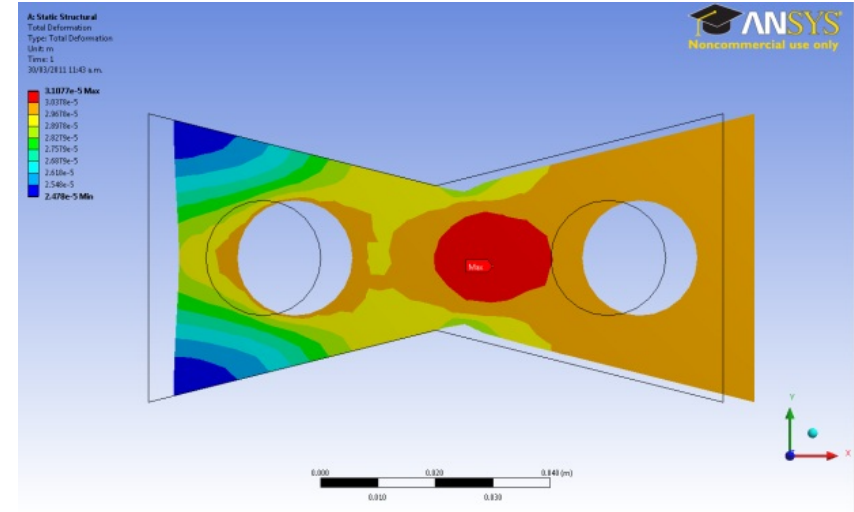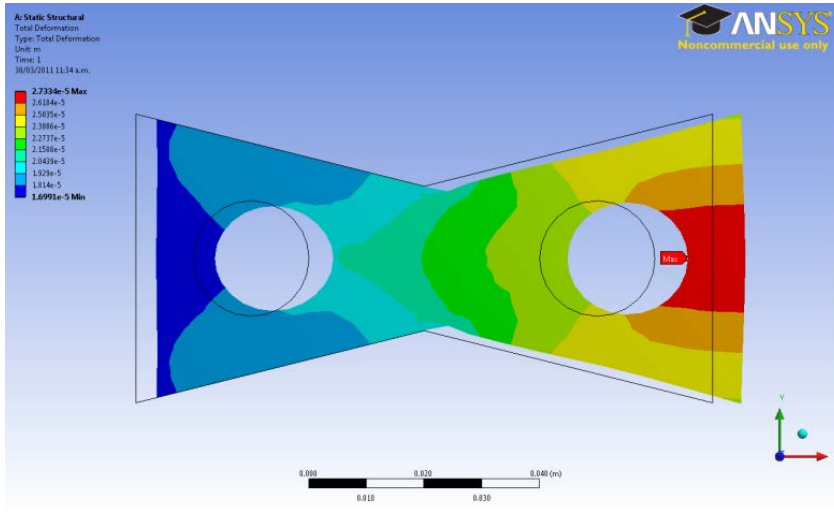
Comparing results

Take some time to compare the results.

Compare the deformations

- How do maximum values differ?

- How do deformations compare near the walls?

- How have the circular holes deformed in each case?

Compare the equivalent stress results

- What and where are the maximum values?

- How do the stress contours compare?

What was the most appropriate combination of constraint and applied loads?
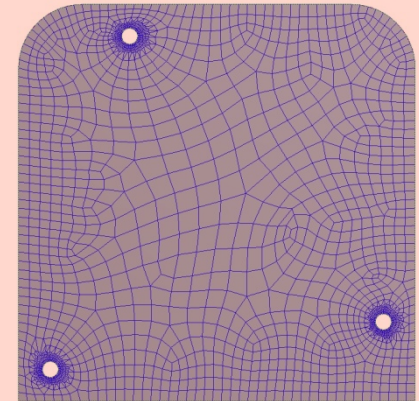
Exercise B: Meshing

Recall again the four basic requirements of a physical model:

a) Physics / Equations? ⇨ **Linear Elastic Structures**

b) Geometry?

Approximated by a mesh of elements

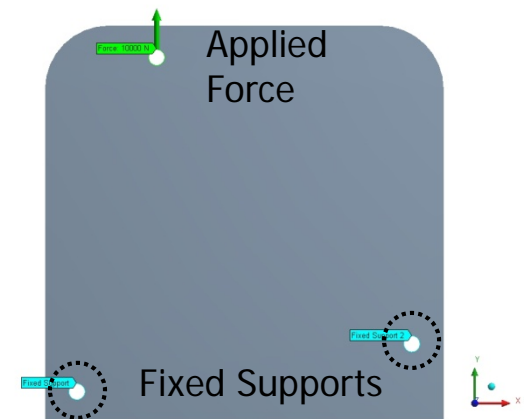c) Material Properties?
- Young's Modulus
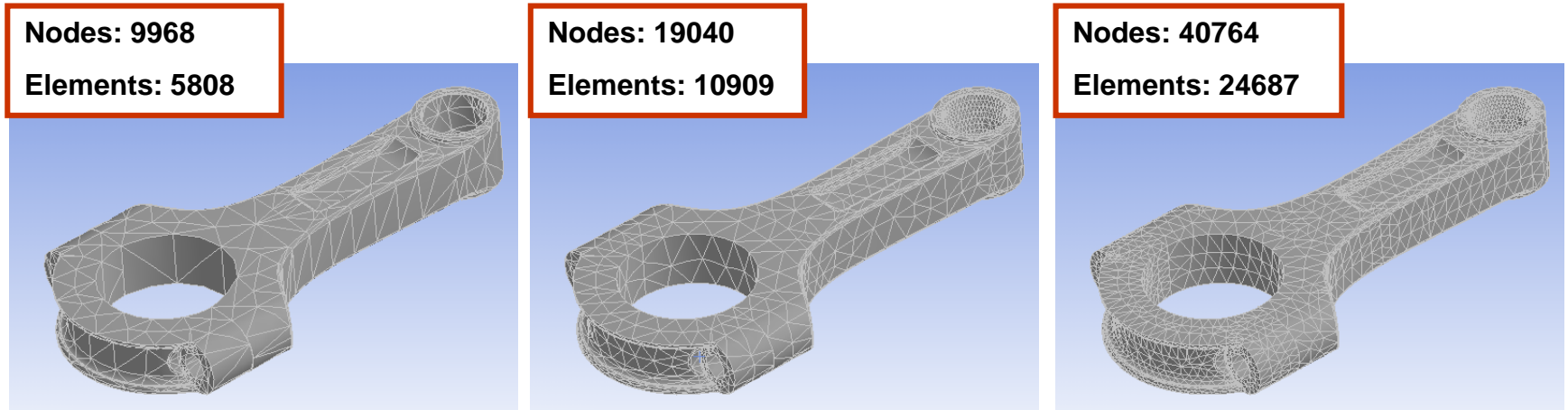- Poisson's Ratio
- Yield Stress
- Tensile Strength
- others…?

Applied Force

Fixed Supports

d) Loads / Constraints (B.C.s)

Solve…

Mesh refinement: More is NOT always better

In general, more elements will increase the accuracy of results.

Nodes: 9968

Elements: 5808

Nodes: 19040

Elements: 10909

Nodes: 40764

Elements: 24687

BUT...as discussed in lectures there is a **penalty** for using more elements!

- More equations - will take longer to solve.
- Larger data files, as we have results from more nodes.

These problems intensify as you move from 1D to 2D to 3D analyses. You need to be very careful with 3D models!

Recap from lectures: Basic meshing concepts

Solution accuracy can be improved by **increasing the number of elements** used in a model.
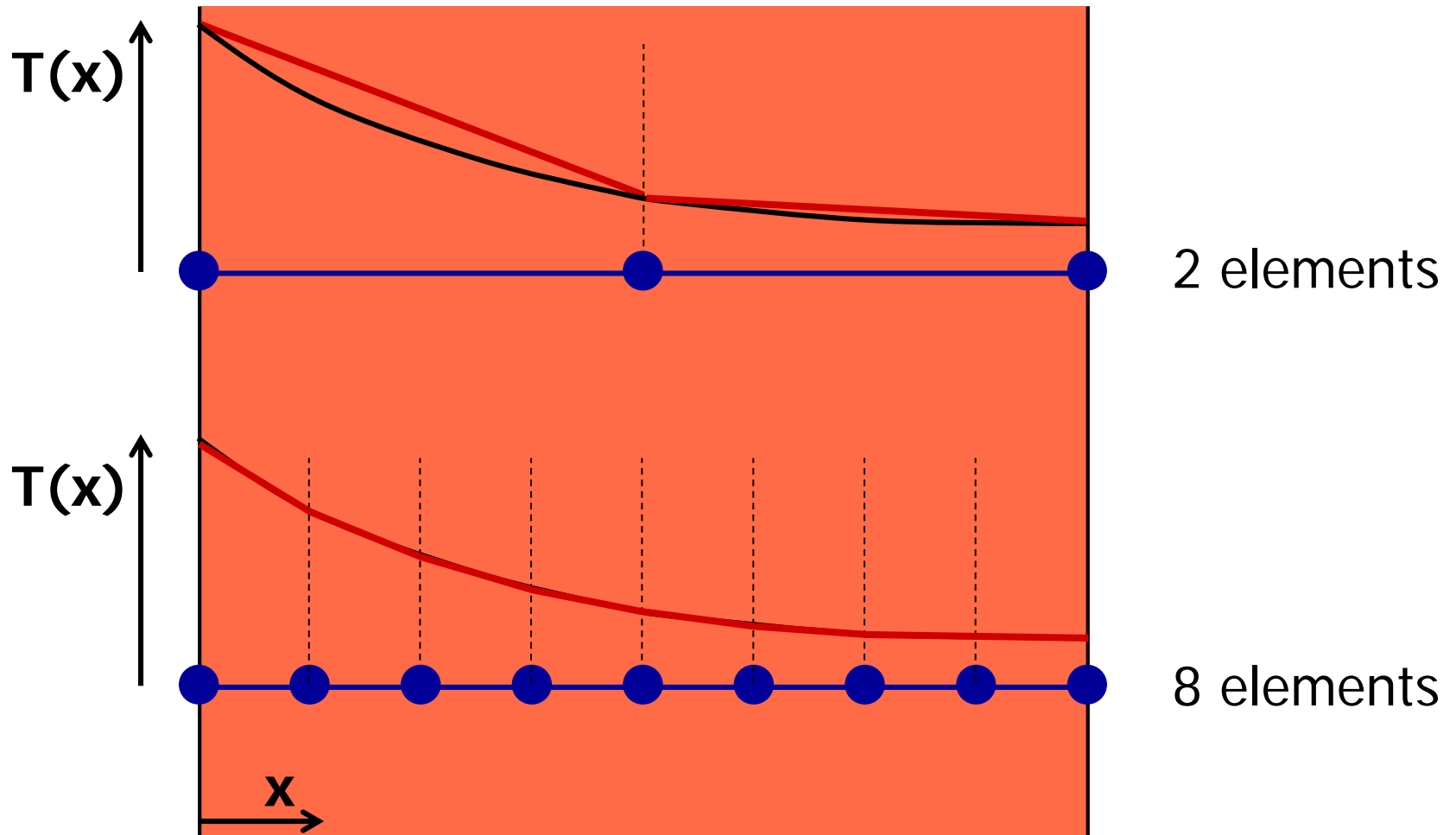
Elements **do not have to be of equal size**. We can make them smaller in areas where results change rapidly.

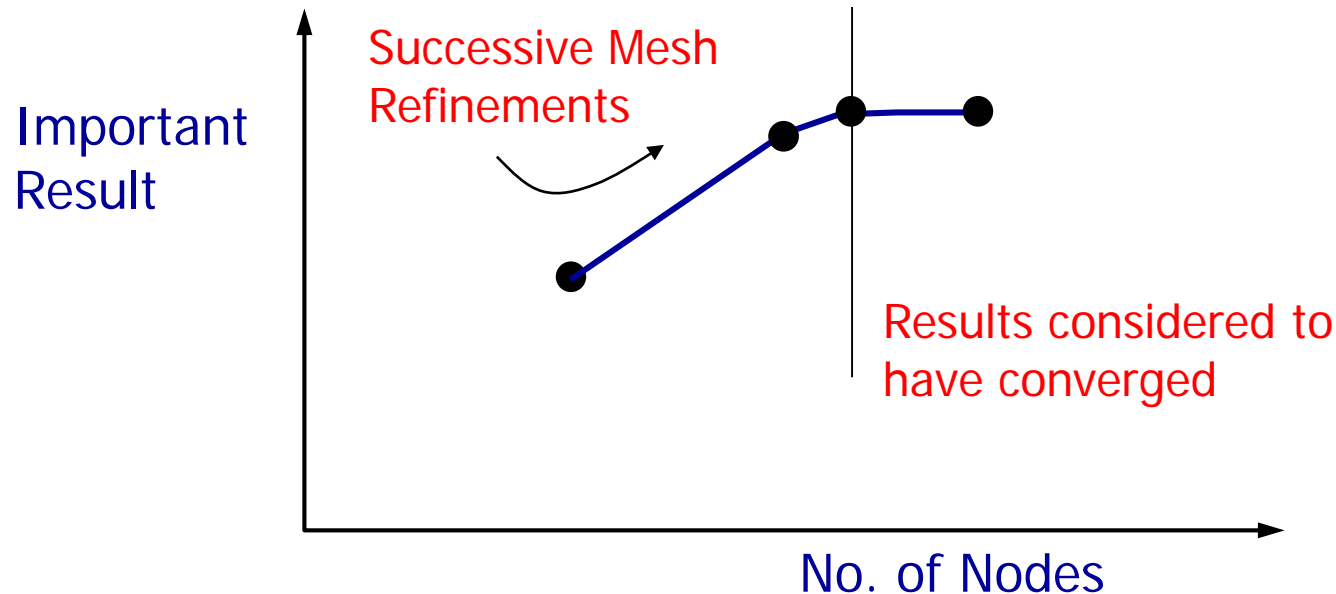

T(x)

x

4 elements

T(x)

4 elements
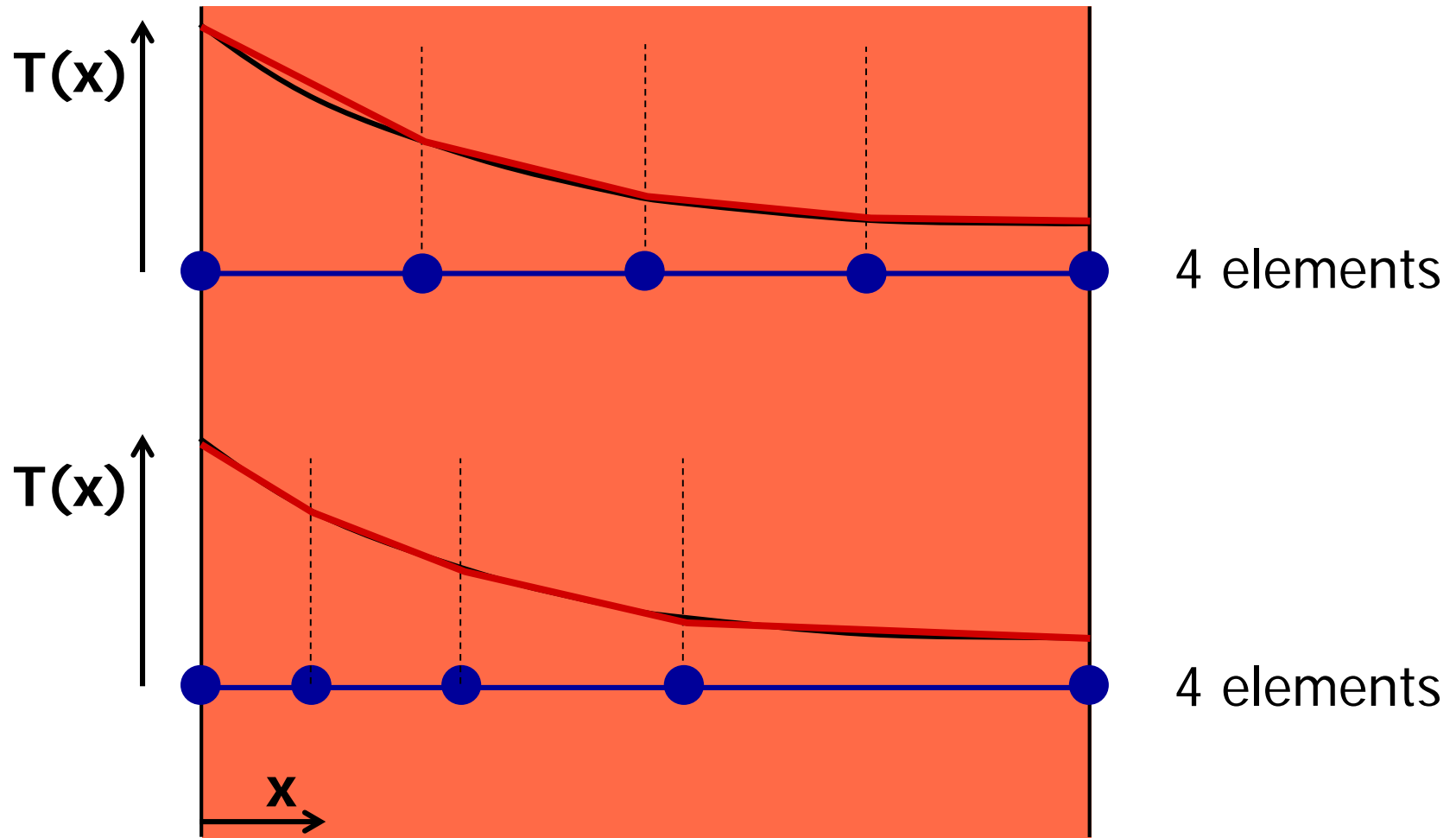
Experience is required to appreciate local mesh refinement.

A considered approach to mesh refinement

The aim of mesh refinement is for **results to converge while utilising resources efficiently**.

Important Result

Successive Mesh Refinements

Results considered to have converged

No. of Nodes

Mesh refinement can be done **globally** (over the whole model), or **locally** (where results are changing rapidly). You should **iteratively improve your mesh**, until you are convinced that the **important results have converged**.

Mesh refinement tools

Workbench provides you with an initial mesh. It provides a lot of tools to refine this mesh. These tools are roughly split to:

Global Refinement Tools

Allow you to refine the size of elements across the whole model (a surface in 2D, or a solid in 3D)
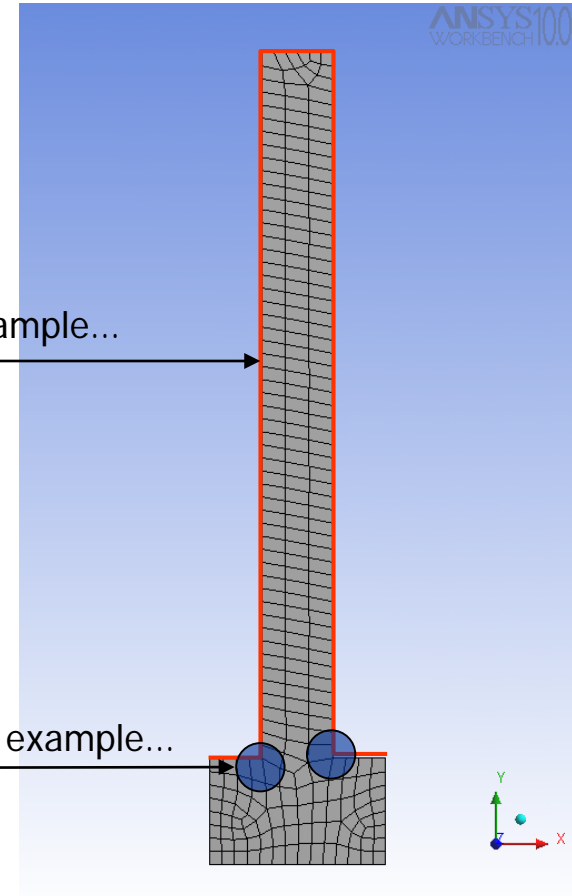
Edge Refinement Tools

Allow you to refine size of elements along model edges.

Point Refinement Tools

Allow you to refine size of elements near specific points in a model.

For example...

For example...

Final points before moving on

IMPORTANT: As a condition of the license, there are **upper limits** on the number of nodes and elements you can use.

- This is more than you will need for this tutorial, and for many of the problems we will consider.
- **Remember**, increasing the number of nodes will **increase the time to solve and increase the size of your project files**. This can be very significant in 3D.

The following exercise will introduce you to **some of the mesh refinement tools** in Workbench. You can learn more:
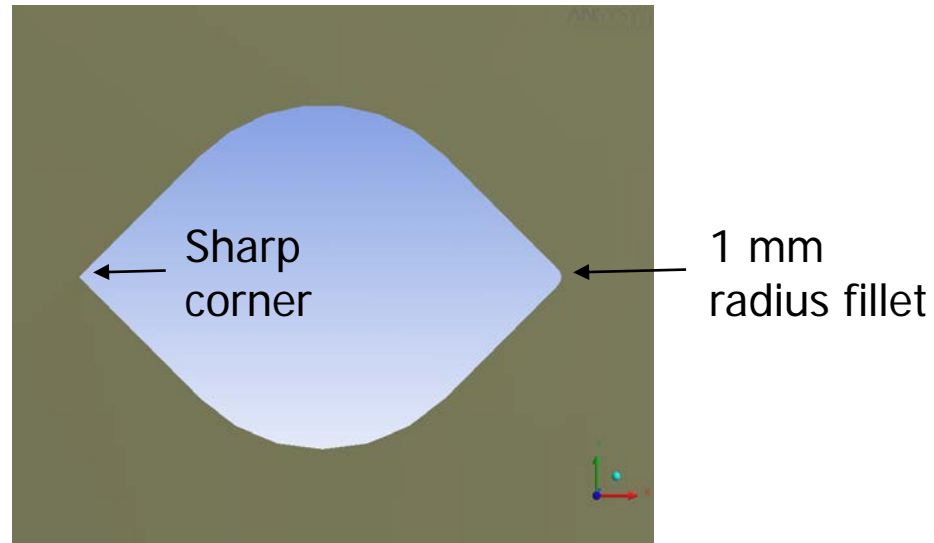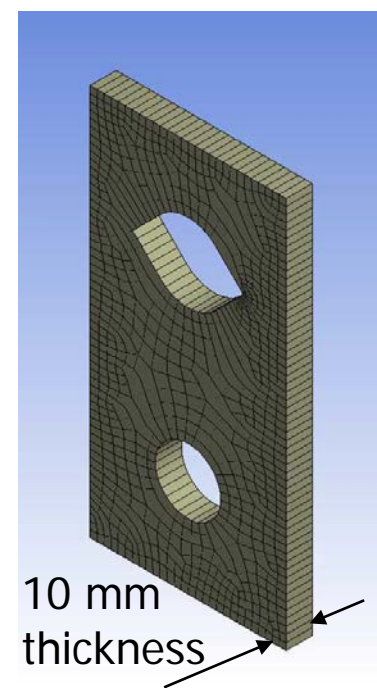
- Go to ANSYS help…

  - Meshing User's Guide
  - Mesh Controls Overview

Mesh Refinement Study in 2D

To apply a variety of mesh refinement tools, we'll again consider a simple component.

75 mm

150 mm
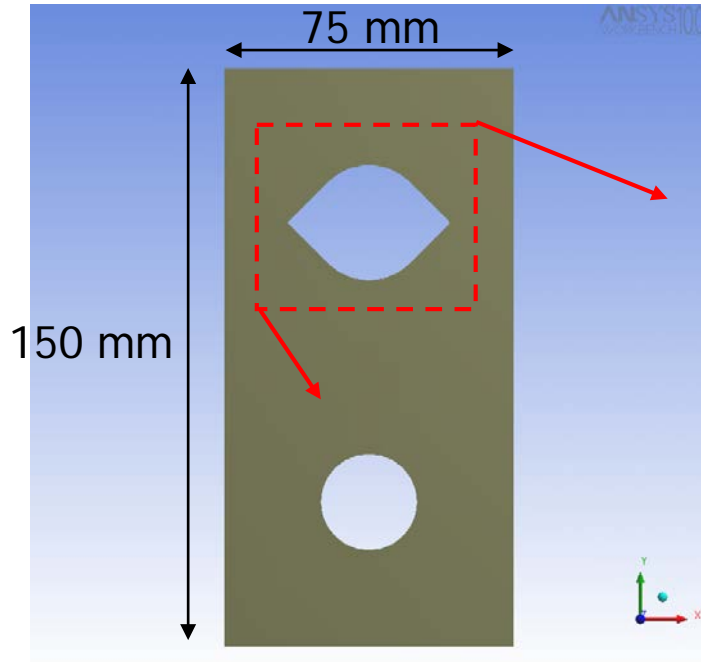
This plate has two cutouts, is fixed at the base, and has a tensile force of 10 kN applied along the top edge.

10 mm thickness

Sharp corner

1 mm radius fillet

In your own time

- Save your previous project.

- Open a new project – click "New" in the Workbench window. Choose a "Static Structural" analysis

- **Special step for 2D analyses**

  - Right Click on step 3 of the analysis (Geometry)

  - Select "Properties"

  - Change analysis type to 2D

- Import the geometry...

  Where you have saved it   .../Tutorial3/TensilePart.agdb

In your own time

- **Special step for 2D analyses.** Specify part thickness in
  - Expand Geometry in the "Outline" box
  - Click on the Part
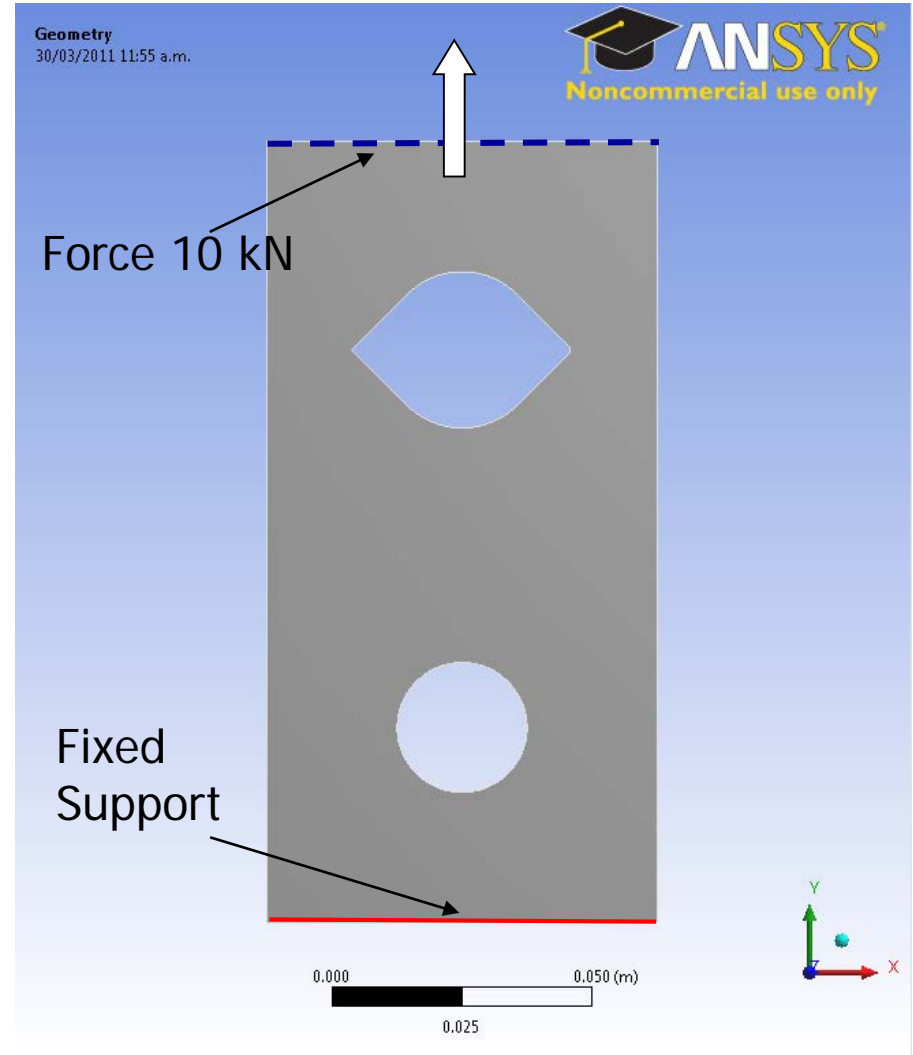 - In the "Details" box, expand "Definition" and specify Thickness
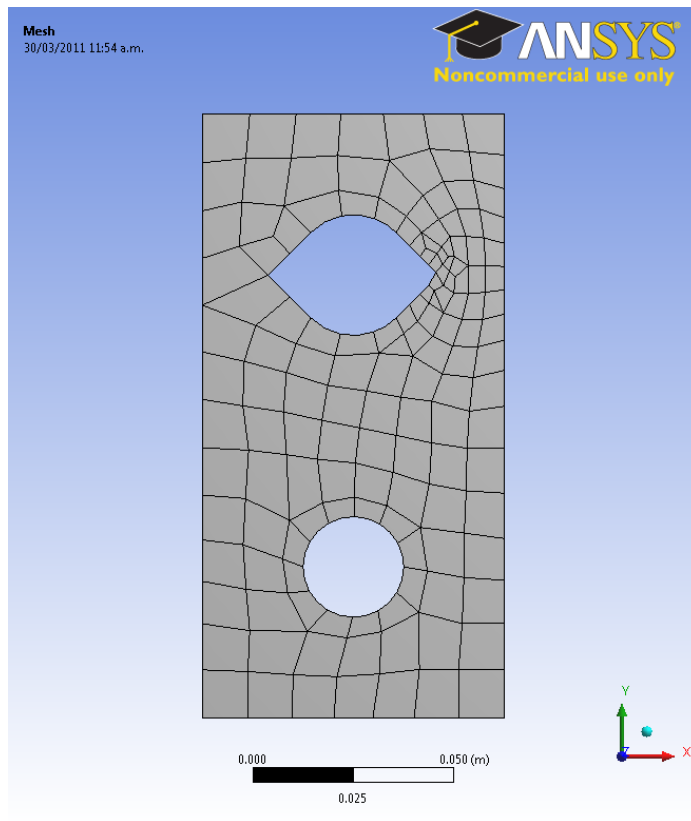
- In ,right click on Mesh in the Outline... Update Mesh

- **Remember to save the project locally or it may not solve!**

In your own time

Application of loads/constraints

• Establish the conditions as shown
•Fixed Support at the lower edge, Force applied at the upper edge.

Geometry
30/03/2011 11:55 a.m.

Force 10 kN

Fixed Support

0.000   0.050 (m)

0.025

In your own time

- Establishing results. The **Solution**…

  - Click on Solution in the Outline
  - From the Solution bar, choose the following results:

1) Total Deformation
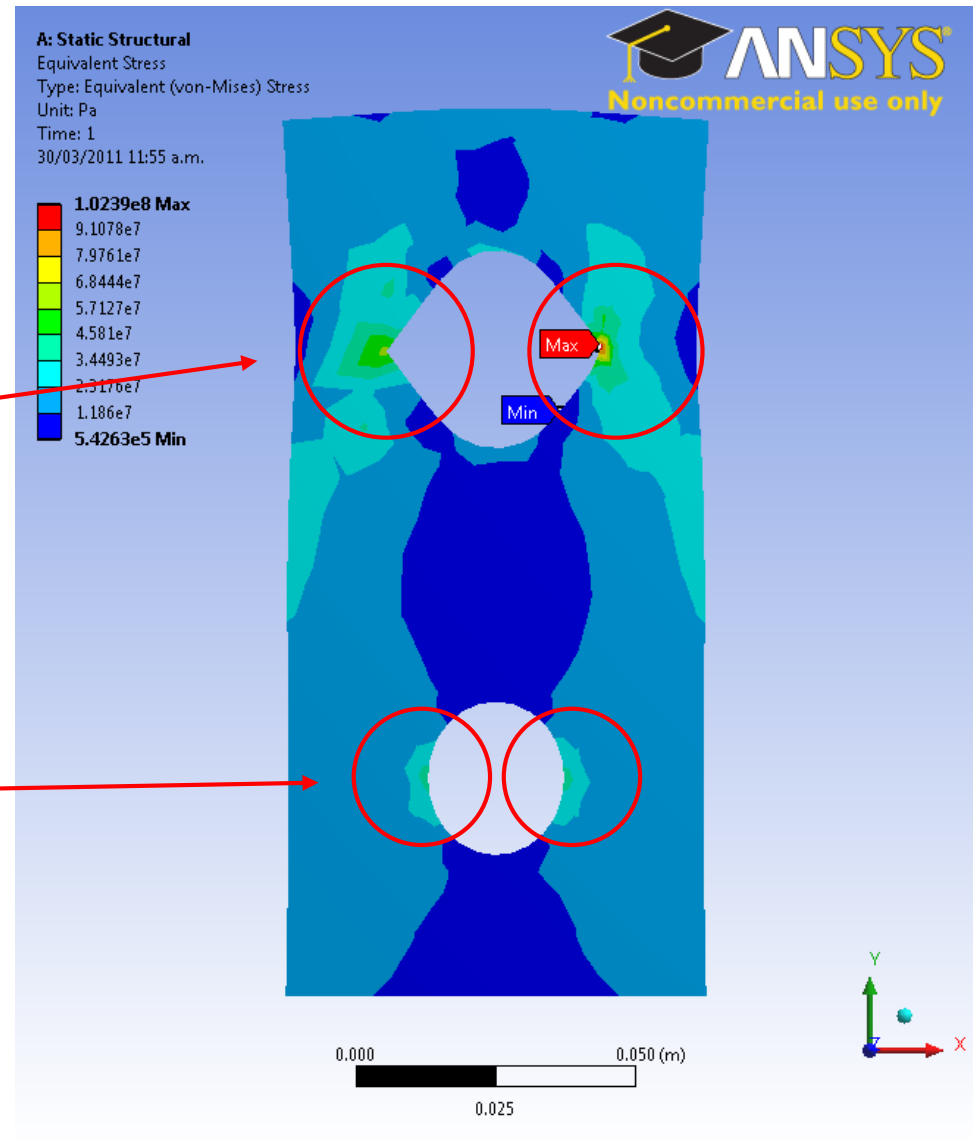2) Equivalent Stress
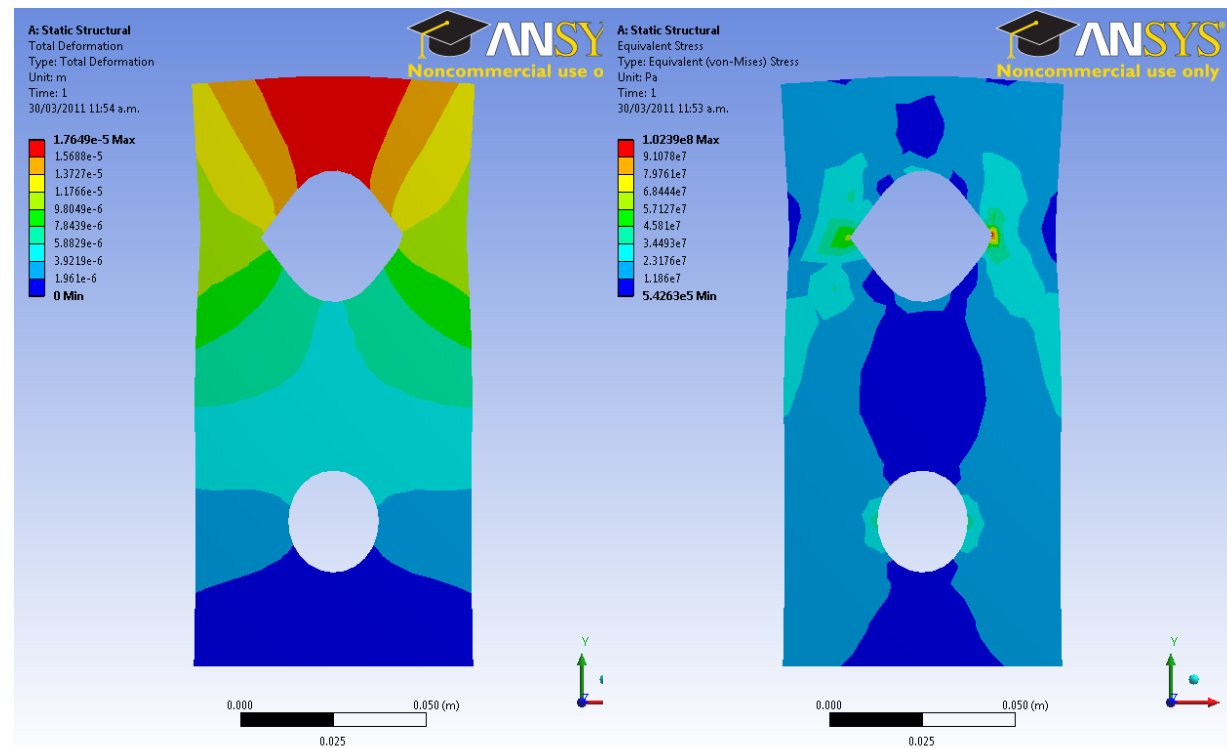
Total Deformation

Equivalent Stress

In your own time

Note the Stress Raising (or Stress Concentration) effects.

• Near the sharp corner, and 1.0 mm fillet, there are large stress raising effects.

• Near the large radius of the lower circle, there is a moderate stress raising effect.

A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: Pa
Time: 1
30/03/2011 11:55 a.m.

1.0239e8 Max
9.1078e7
7.9761e7
6.8444e7
5.7127e7
4.581e7
3.4493e7
2.3176e7
1.186e7
5.4263e5 Min
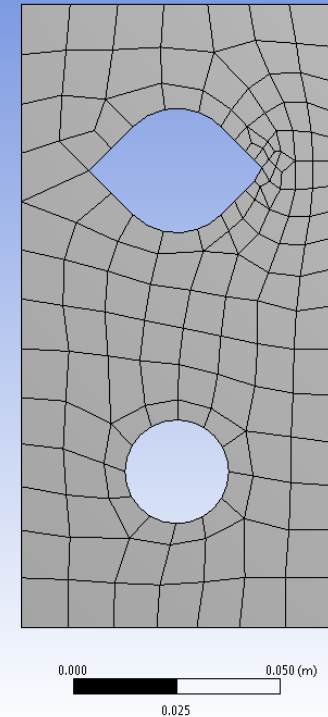
Max
Min

0.000   0.050 (m)
         0.025

Is this mesh refined enough to provide convergence? Of deformations? Of stresses?

We'll carry out a mesh refinement study on this part, considering mesh refinement at three levels:

- Global (on the whole part)
- Edges
- Points

Note, if we were doing refinement of a 3D mesh, we could add another level.

- Refinement at surfaces

Creating Multiple Models to try different meshing techniques

• First, rename the original model "BaseModel" from the 🔺 window.

• Duplicate "BaseModel" **four times at the "Model" step**. Note that step 4 is unlinked between the duplicates, so each mesh can be different

• Rename each of these new models as follows:

  • "GlobalRefinement"

  • "EdgeRefinement"

  • "PointRefinement"

  • "CombinedApproach"

Right Click 'Model' to duplicate

Click here to rename
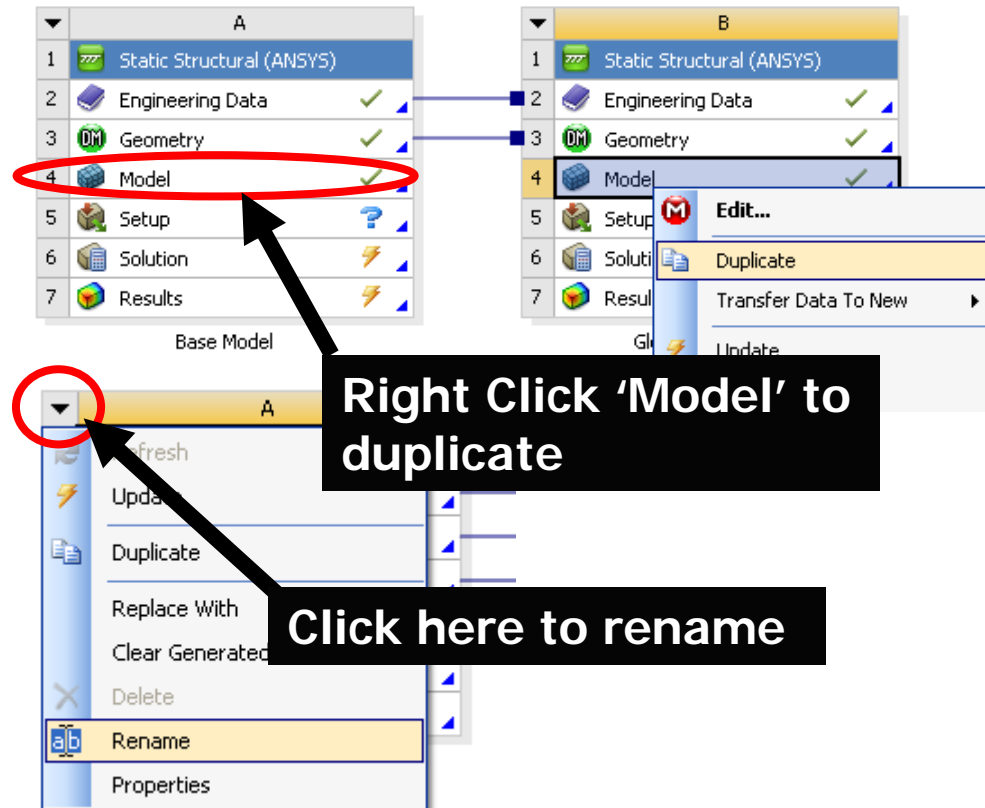
Global mesh refinement

We will try two approaches to global mesh refinement. **Global Refinement** and **Global Sizing**.

- Work in the "GlobalRefinement" Model - Right click on "Model" and click "Edit..." to open

- Select the entire part (it is a surface).

- Right click on mesh, insert Refinement. You can choose three levels of refinement 1, 2, 3 (Choose 3).

- To preview the mesh, right click on "mesh" and update. Then solve.

IMPORTANT STEP: REVIEW RESULTS

- How many more nodes and elements were used? Was it slower to solve?

- How have the results changed from the "Base Model"? Deformations? Stress values at raisers?

Now we will try **Global Sizing**.

• Under mesh, right click on the refinement you just produced, and select suppress. This turns it off, but does not delete it.

• Again, select the entire part.

• Right click on mesh, insert Sizing. You then need to enter the typical size of the element. Try putting in 0.002m. The elements will be **roughly** 2mm high and wide.

• Right click on mesh and update to preview mesh. Then solve.
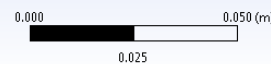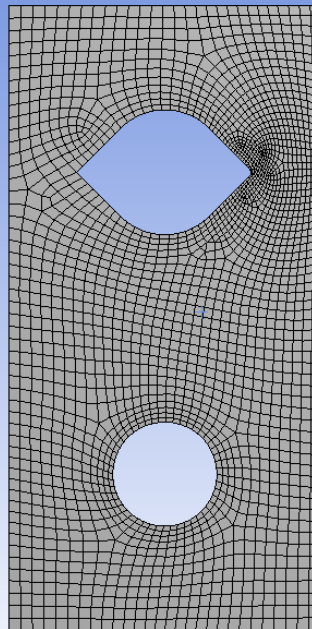
IMPORTANT STEP: REVIEW RESULTS

  • How does this mesh compare to the previous two?

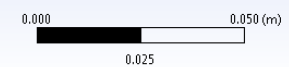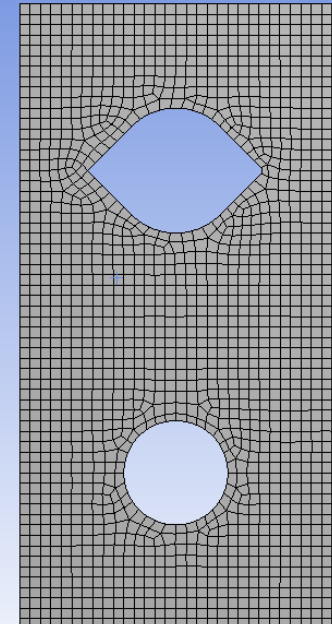 • Are the results much different from the "Base Model"?

Your meshes should have looked something like:
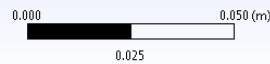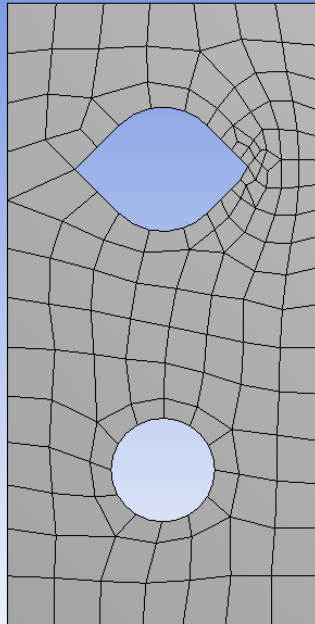
Original Mesh

Refinement, Level 3

Sizing, 0.002 mm
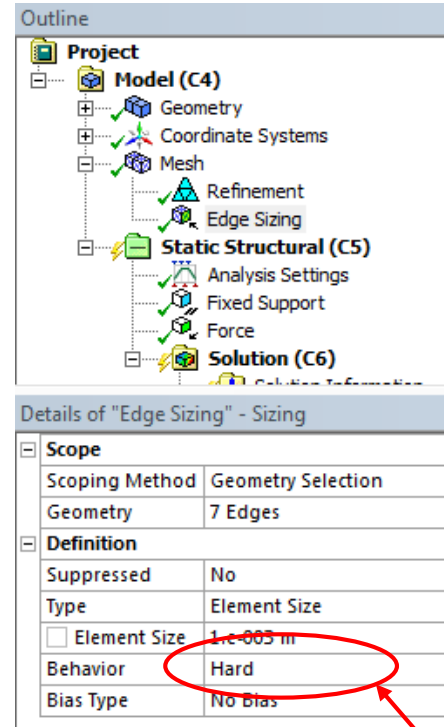
Edge mesh refinement

Now we will try two approaches to edge mesh refinement. **Refinement** and **Sizing**.

- Work in the "EdgeRefinement" Model.

- Choose the edges of the circular hole.

- Right click on mesh, insert Refinement. You can choose three levels of refinement 1, 2, 3 (Choose 3).

- Choose **all** edges of the upper cutout.

- Right click on mesh, insert Sizing. You then need to enter the typical size of the element. Try putting in 0.001m. The elements will be 1mm wide along all of the edges. **NOTE:** You will need to change "Behavior" from Soft to Hard.
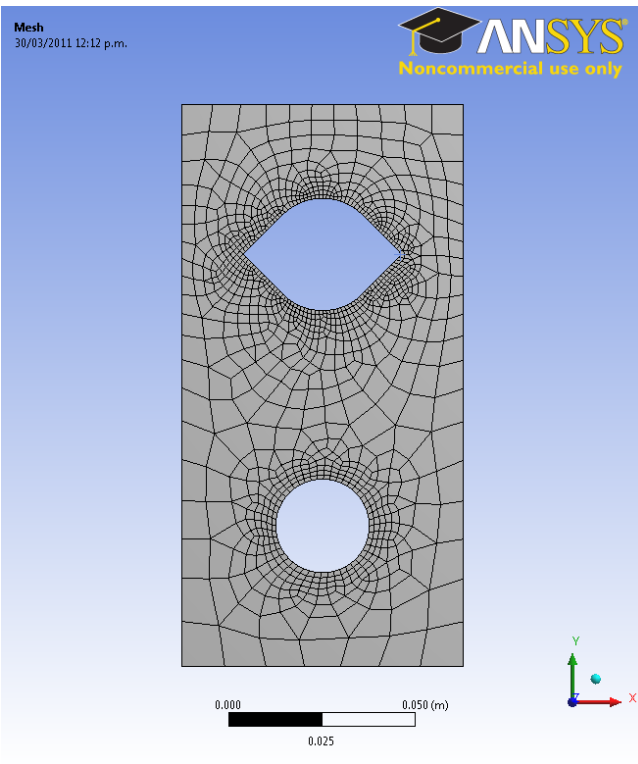
Edge mesh refinement
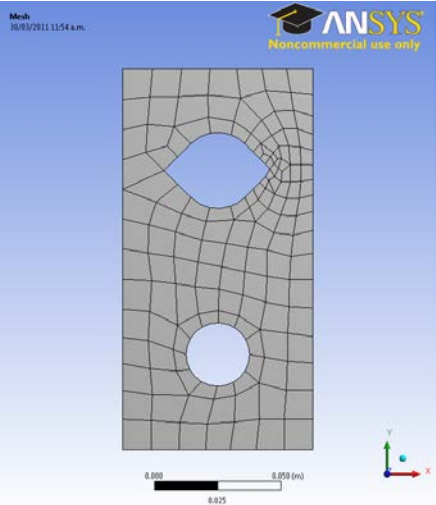
• Right click on mesh to Preview Mesh. Examine the resulting changes.
• Then, solve the model.

IMPORTANT STEP: REVIEW RESULTS

• Any difference in predicted deformation between the models?

• Any difference in the predicted equivalent stress values?

  • The maximum value?
  • Values around the circular hole?
  • Values at the sharp and 1mm radius?

Point mesh refinement

Finally we'll try two approaches to mesh refinement near a point. Again we apply Refinement and Sizing.

- Work in the "PointRefinement" Model.

- Choose the two points in the vicinity of the 1mm radius of the upper cutout.

- Right click on mesh, insert Refinement. Choose a refinement level of 3.
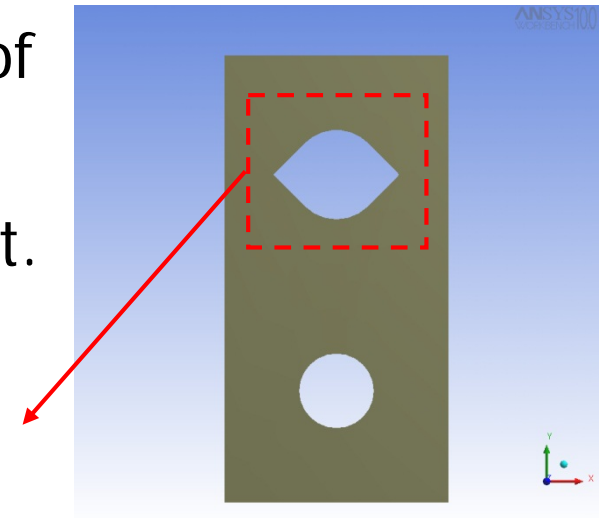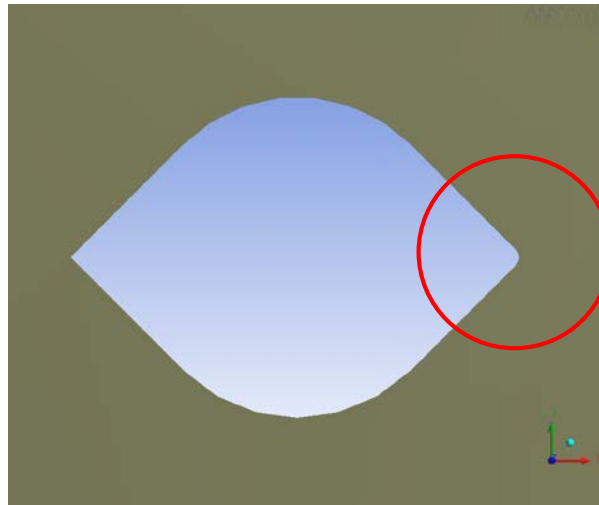
Point mesh refinement

We can also apply Sizing refinement near a point. In this case we define a volume over which the finer elements are defined. This is called a Sphere of Influence.

• Choose the point at the left side of the upper cutout.

• Right click on mesh, insert Sizing. You then need to enter the radius of the sphere of influence. Try 0.01 m.

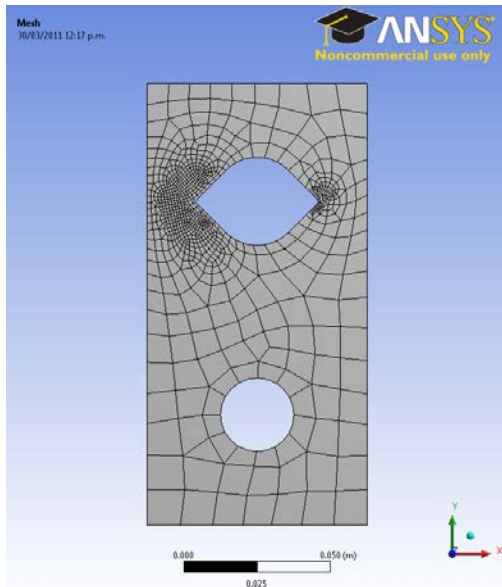• In the same window, define the typical size of these elements. Try 0.001 m.

Sphere of Influence
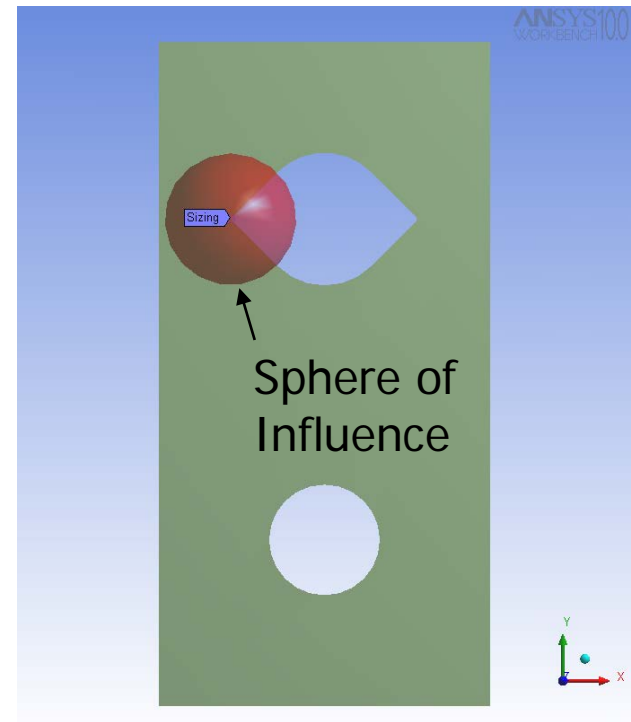
Point mesh refinement
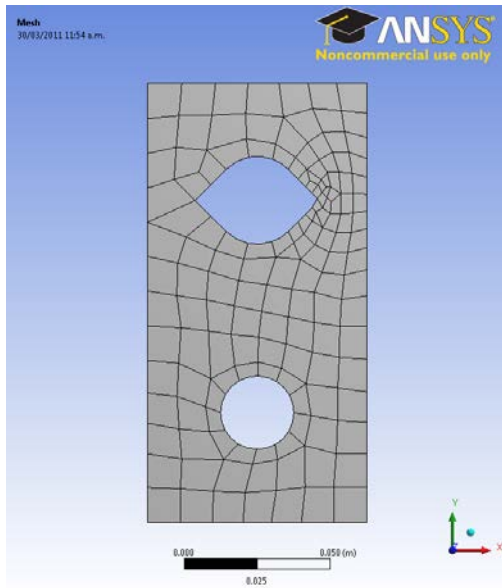
• Right click on mesh and update. Examine the resulting changes.
• Then, solve the model.

IMPORTANT STEP: REVIEW RESULTS

• Again, any difference in predicted deformation and stress results between the models?

Stress singularity

• Try refining the mesh in the sharp corner, and the 1 mm radius of the cutout. Can you get the stress to converge?

You should have success with the 1 mm radius. However, a **stress singularity** will exist at the sharp corner. As you increase the number of elements locally, the stress locally will continue to increase, and will not converge. This is a mathematical problem.

You can either:

• Accept sharp corners that cause stress singularities, and remain skeptical about the maximum stress in these areas.

• OR, include a small radius fillet in the model.

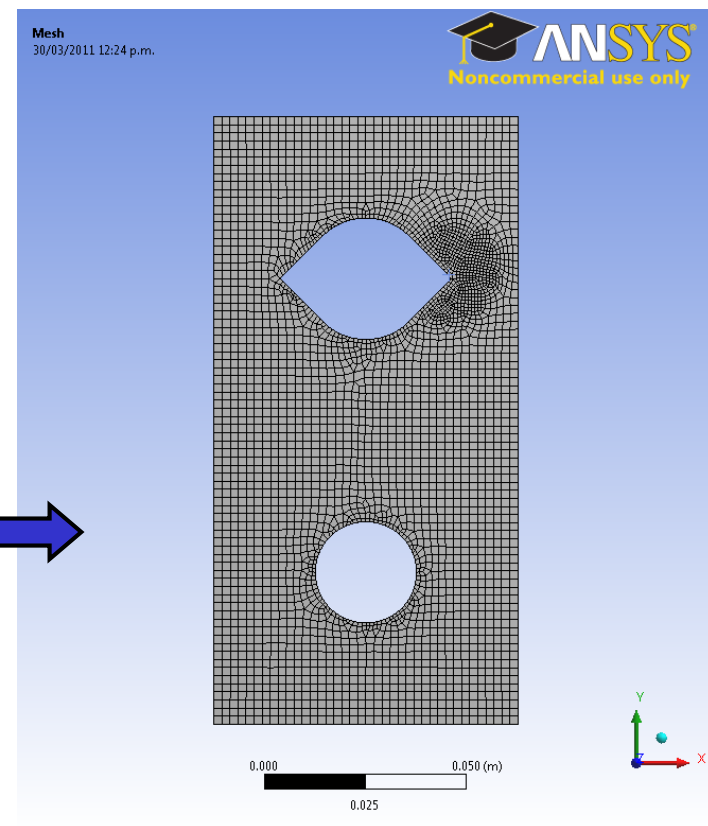In reality a "sharp corner" must have a small radius…

Combined approach

A complete mesh refinement approach may use global, surface, edge, and point refinement tools.

Working in the "CombinedApproach" model, try applying a mixture of the mesh refinement tools you've tried so far.
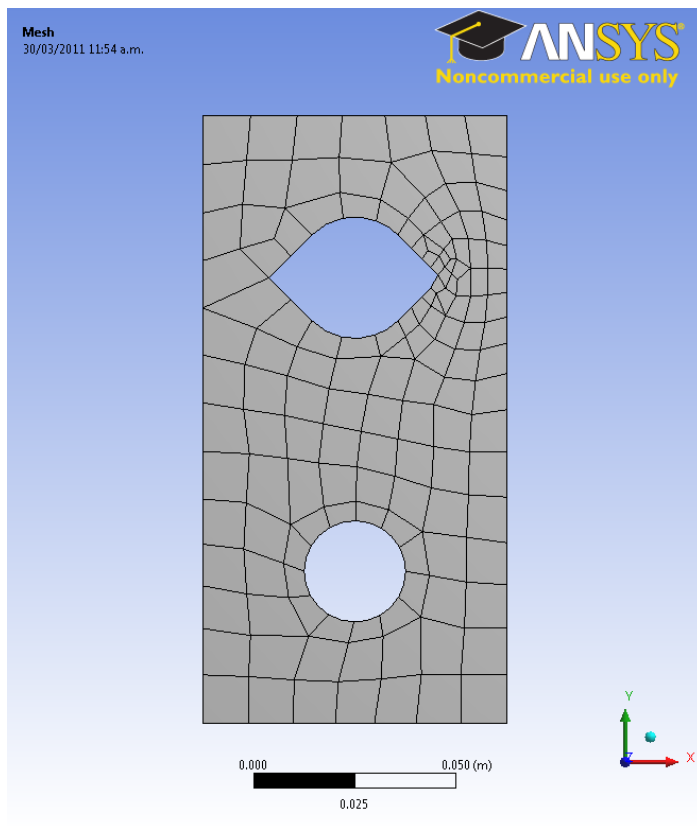
Here is one possible set of refinements...

All done!
Now move on to Tutorial 4.