FEA in the design process: An introduction to Finite Element Analysis (FEA) with ANSYS Workbench

Tutorial 1: Introduction to ANSYS Workbench

Dr David Wynn

Email: d.wynn@auckland.ac.nz

Room: 401.916

Content of Tutorial 1

- An introduction to ANSYS Workbench.

- Navigating around Workbench: How is the file system organised? What are the features of the interface?

- Application of the four basic requirements of physical models within FEA:

a) Physics / Equations

c) Material Properties

b) Geometry

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

- You will gain experience as you work through two examples with our help: A Heat Conduction analysis, and an Elastic Structural analysis.

Caution!

Modern FEA software is easy to use.
DANGEROUSLY easy.

**ANSYS Workbench** is the commercial FEA package we will use during this course. **Why?**

- It's easy to use! Dangerously easy…
- Seriously, it is a very intuitive interface, allows you to adjust your models quickly, and interfaces with Creo.
- Is an excellent introduction to the full commercial package **ANSYS Classic**.
- Workbench will supercede Classic soon.

Unfortunately packages like Workbench are being used by people who've never done Mechanics of Materials etc…

Commercial FEA Packages

Multiple commercial FEA packages are used in industry. The software is expensive, so a company may use one of them.

These packages have a variety of strengths and weaknesses. The major packages include:
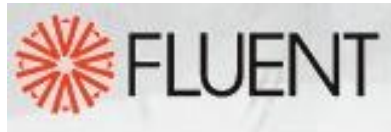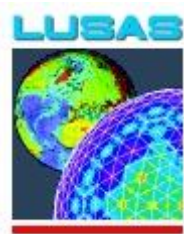
And lots more...

It takes some time to learn the use of each package, **BUT** the underlying concepts covered here will apply to all.

Capabilities of ANSYS Workbench

Workbench can address various physics:
- Structural analysis, isotropic materials
- Thermal conduction analysis, isotropic materials
- Vibrations analysis (modal frequencies)
- Electromagnetic analysis
- Combinations of these
- Steady State AND Transient analysis

Workbench can be interfaced with Creo, Solidworks, and a variety of other CAD packages.

Workbench includes its own solid modelling package, so you can develop FEA models independent of other software.
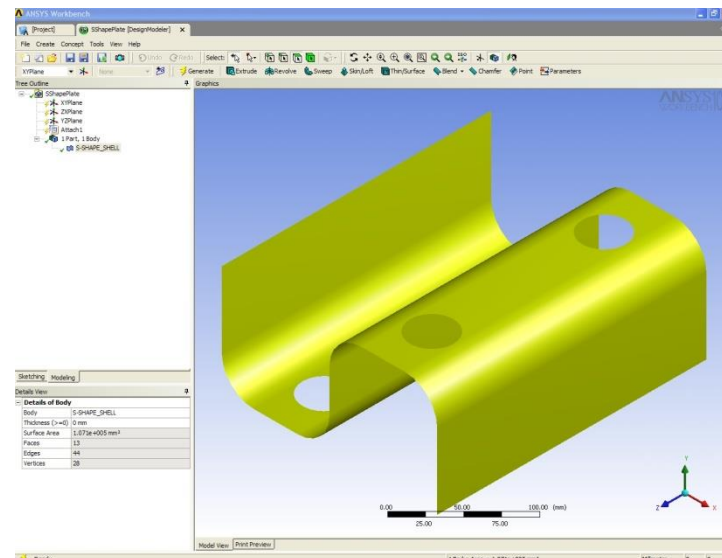
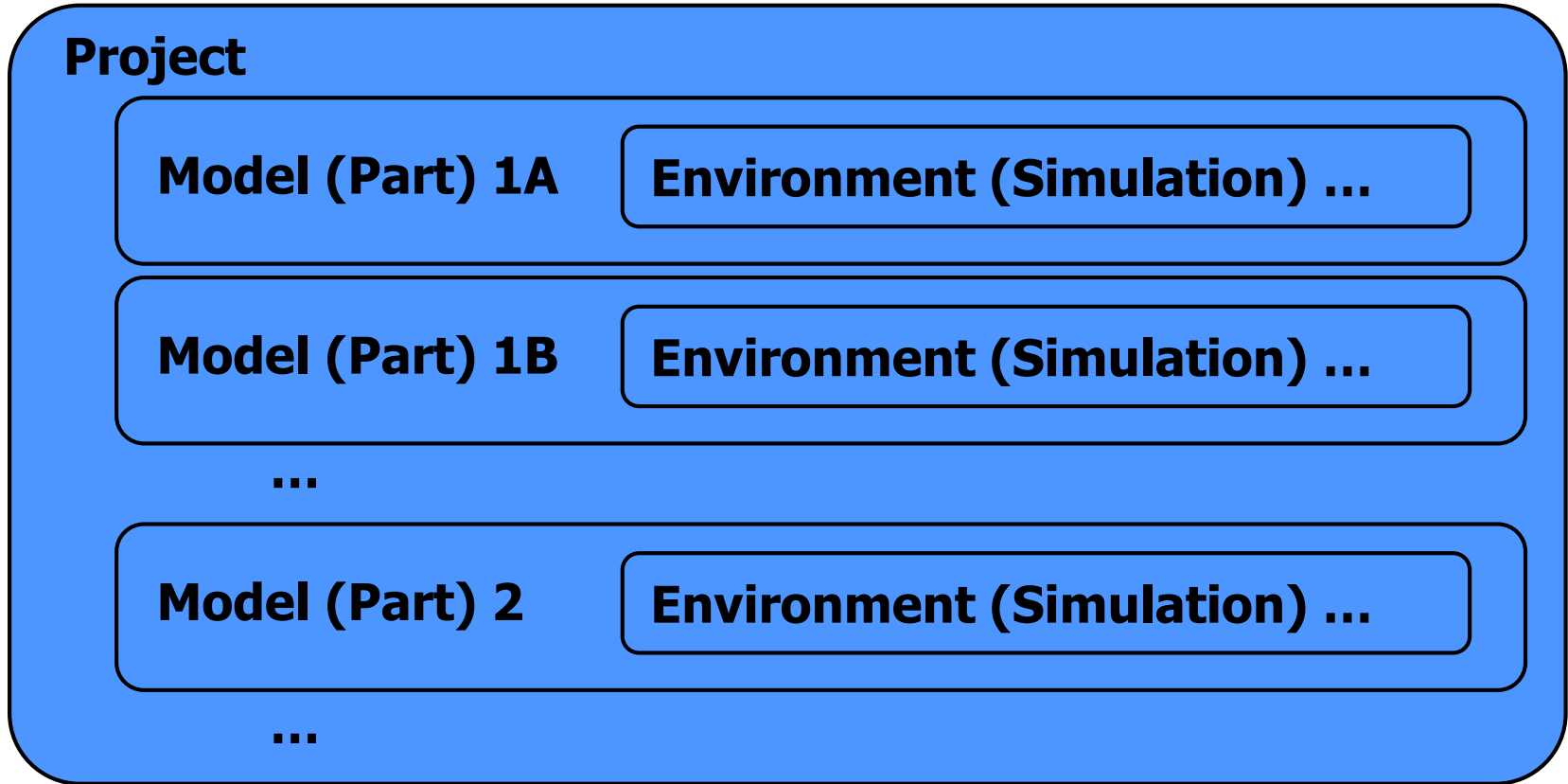Recall that there are four components to any physical model:

a) Physics / Equations

b) Geometry

c) Material Properties

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

Workbench File System

Workbench organises your work into **projects**...

Project

| Model (Part) 1A | Environment (Simulation) ... |

| Model (Part) 1B | Environment (Simulation) ... |

...

| Model (Part) 2 | Environment (Simulation) ... |

...

• You can work on several **Models** (or Parts) within a project.

• To carry out multiple simulations with the same geometry, you need to create multiple models of that geometry.

Getting Started

- Launch **"ANSYS Workbench"** from application launcher. You are faced with the **Workbench Interface**, which allows you to:

- Open an existing project.

- Navigate between various Models, launching the modeller.
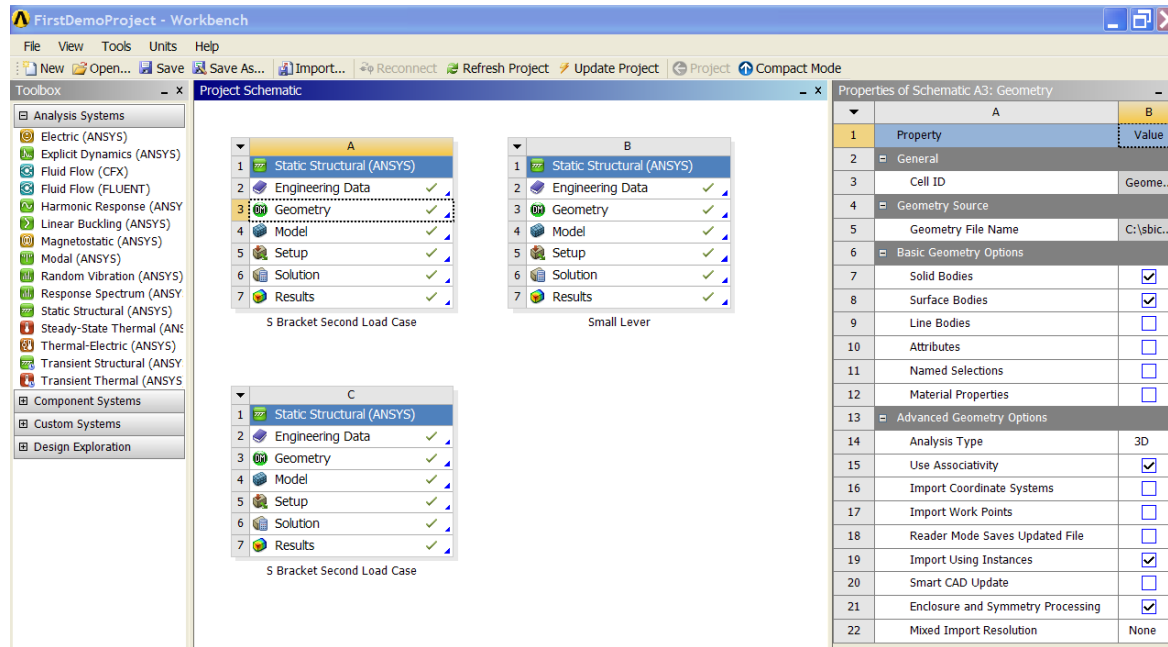
This window, the "ANSYS Workbench Interface" is particularly important.

From here we include material properties, and launch geometry and modelling screens.
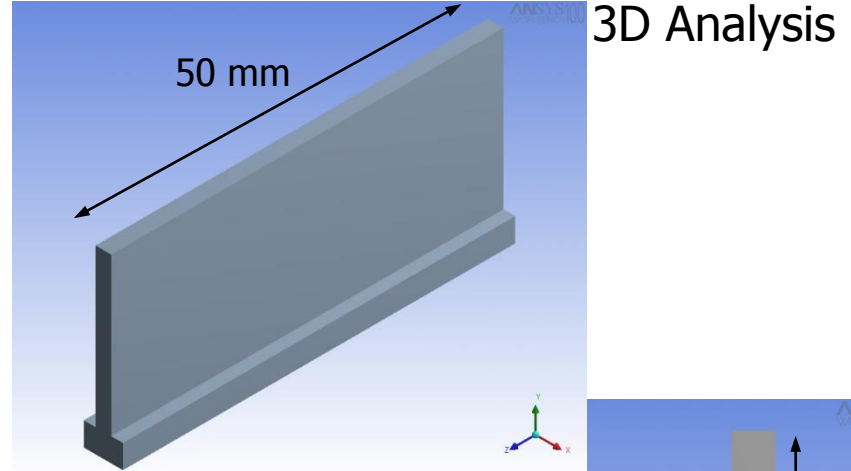
- Navigate between Geometry, Mesh, Environment, Results.
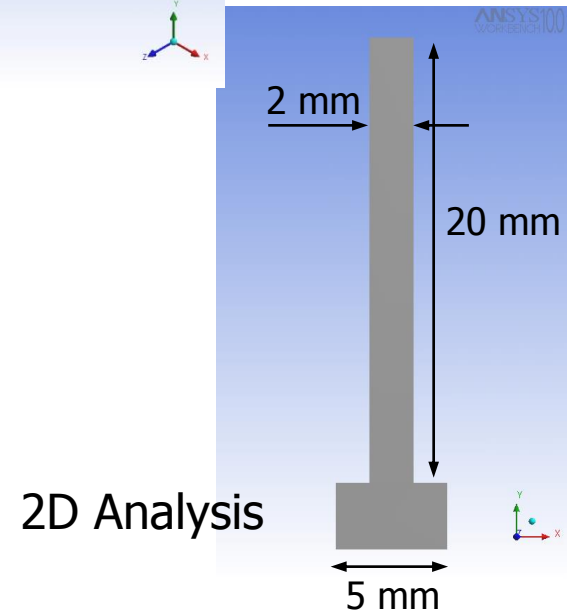
Heat Conduction Analysis

We'll look at a heat sink, using it as an introduction to carrying out a heat conduction analysis.

3D Analysis

50 mm
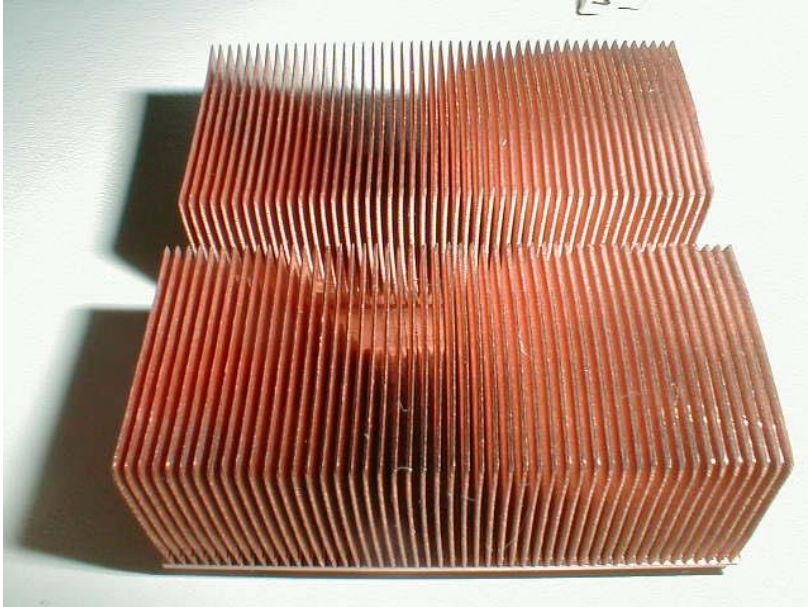
For simplicity, we'll go with a **2D analysis**.

2 mm

20 mm

2D Analysis

5 mm

Step 1

- Launch **"ANSYS Workbench"** from start menu (if you can't find it in the menu, search for the word *Workbench*)

- Drag "Steady State Thermal" from the Toolbox into the Project Schematic window to create an Analysis System **(or** double click on "Steady State Thermal")

- **Special step for 2D analyses:**

  - In the Analysis System you just created, right click **Geometry**, and select properties

 - Under Advanced Geometry Options, set Analysis Type to 2D

Steady-State Thermal (ANSYS)
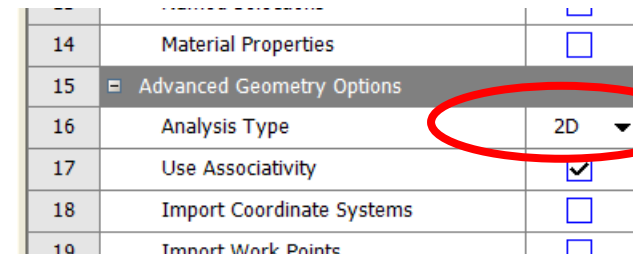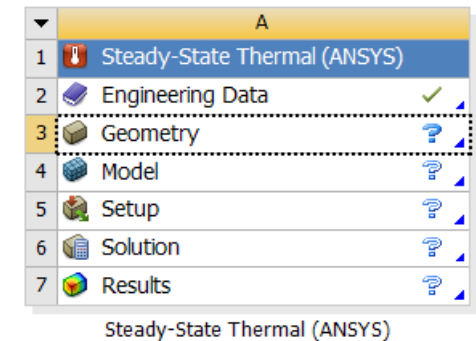
Step 2

- Import geometry to your Analysis System

Steady-State Thermal (ANSYS)

 - Right click Geometry.
  - Select "Import Geometry" > browse

Where you have saved it   …/Tutorial1/2DFin_m.agdb

- Right-click Model, and select Edit (This will open ANSYS Mechanical, from where you can work with the mesh, boundary conditions and results)
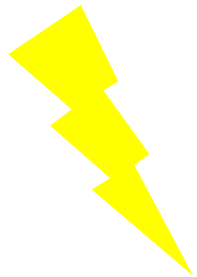
Note: This symbol indicates that something needs your attention! In this case, you'll see it in Mechanical because a mesh has not yet been generated…
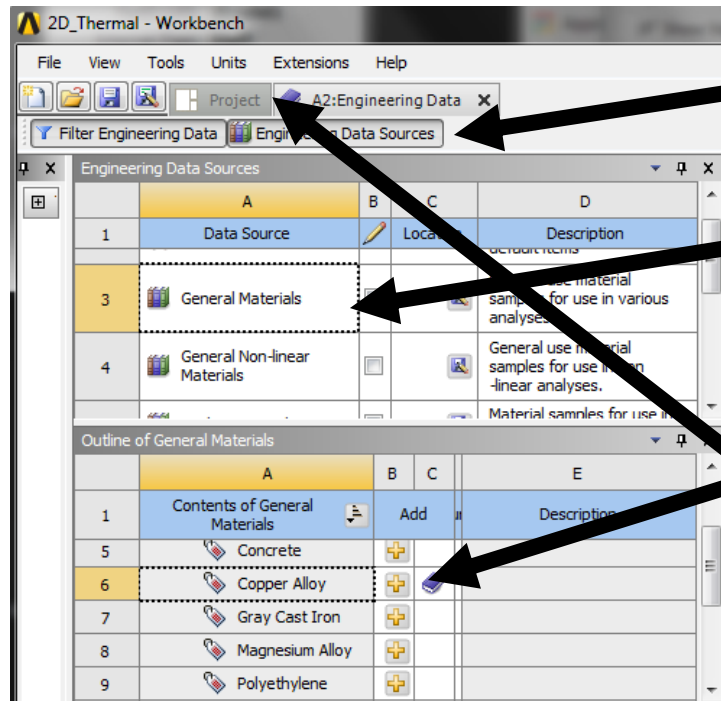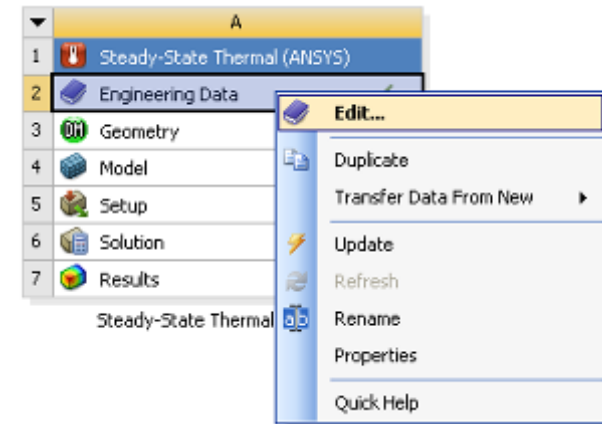

- In the Mechanical window, right click on Mesh in the Outline -> Generate Mesh
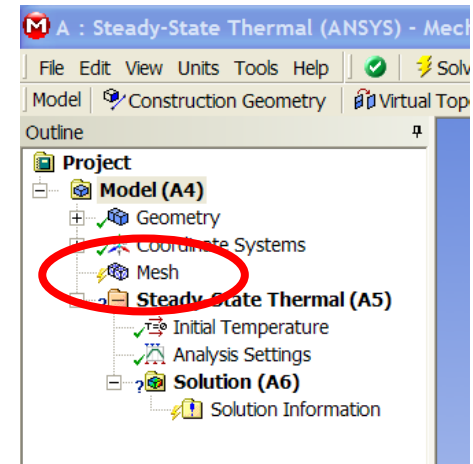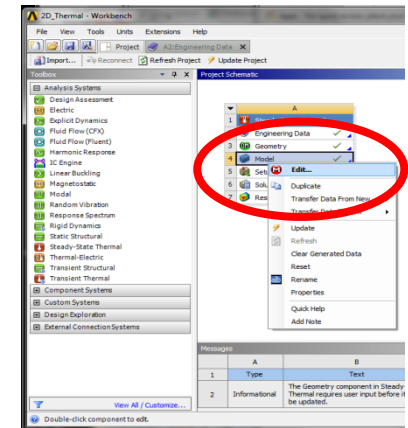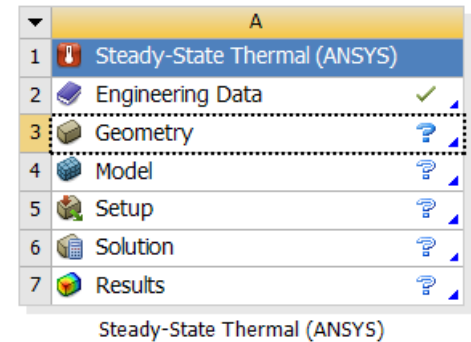
Step 3

Importing and selecting a Material

• Back in the Workbench Interface, right-click Engineering Data, and select Edit

• This opens the Materials Interface

• Find Copper Alloy and add it to your Analysis System

**1. Toggle the data library ON**

2. Click a source of materials

3. Click "+" to add the material to the simulation library

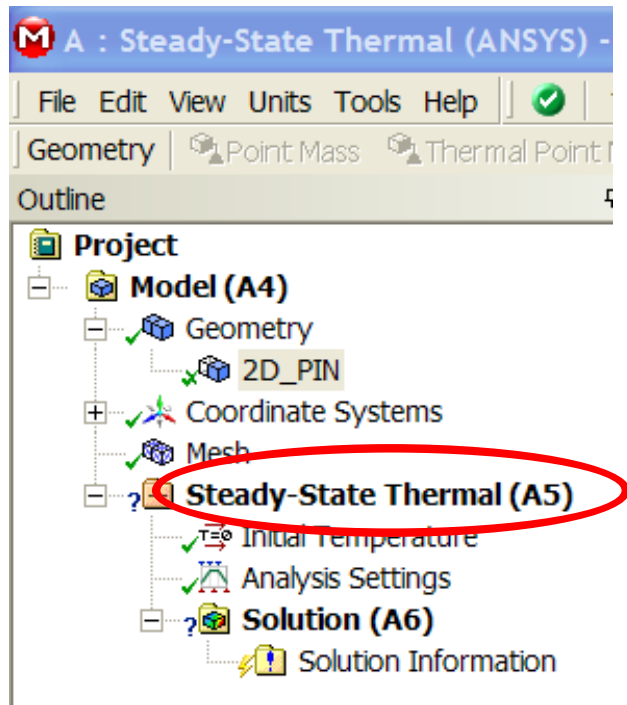**4. Click on the project tab and then update project**

Step 4

- Return to the Mechanical window.

- Expand the "Geometry" tab and select the part

- Under Details, change the material to copper alloy

- **Special step for 2D analyses:** Specify part thickness.

- In the Details window, enter the part thickness.

- We will use **0.05 m (50 mm)** in this example.
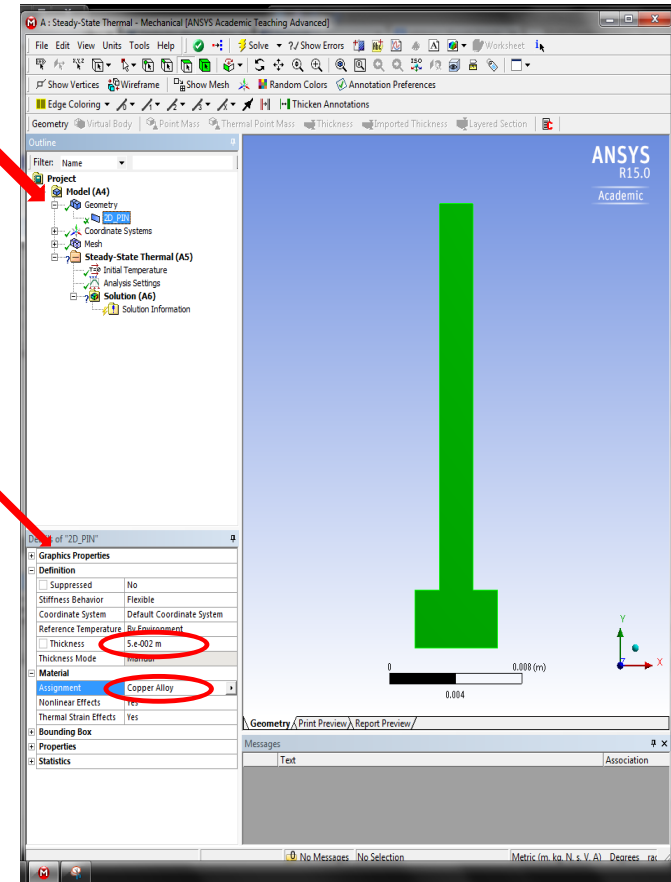
Step 5

Establish the **boundary conditions**

• Click on the Environment (Steady-State Thermal) in the Outline

• Some boundary conditions will be specified in the following slides.

•However, you could specify any boundary conditions you like, according to the physical situation you want to examine!

Step 6

• From the Environment bar we can create different types of load/constraint.

• These can typically be applied to different types of Geometric Entity: vertices, edges, faces, or bodies.

In this case, we will apply BCs to *edges* of the pin - so be sure the Edges tool is selected!

• On the next page you will apply the following BCs to the edges of the pin:
- Temperature (to 1 edge)
- Convection (to 5 edges)
- Heat flow (to 2 edges)
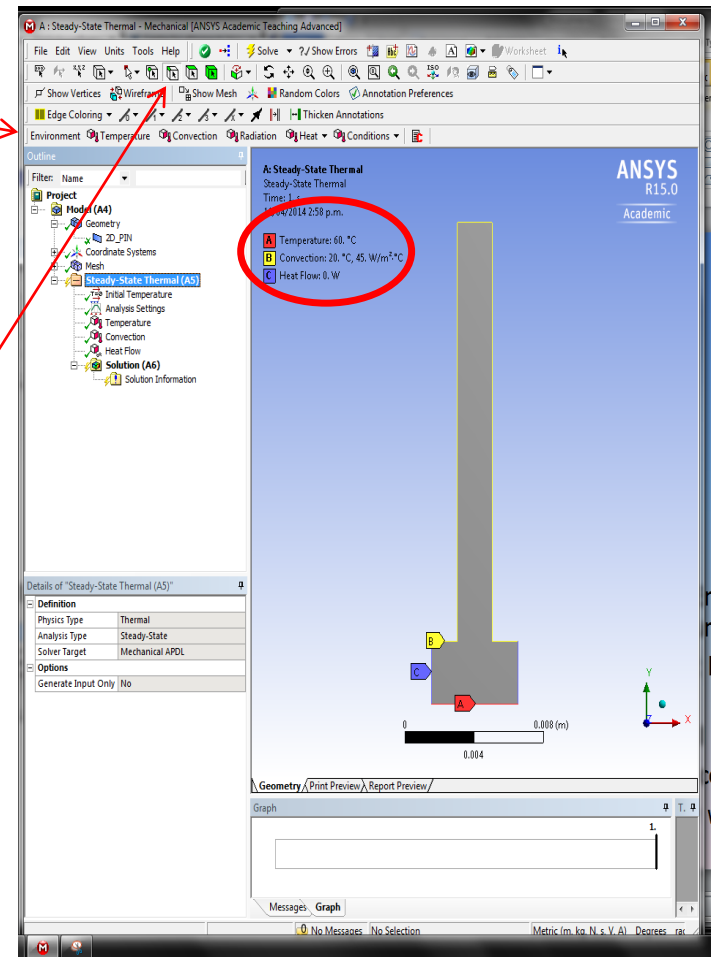
Step 7

Applying a Temperature BC:

- Click "Temperature" on the Environment bar. This will create a new BC, which will appear in the Outline.

...now, we need to configure the new BC...

- Ensure the new BC is selected in the Outline, then:
  - Ensure the Edge tool is selected and pick the bottom edge of the fin.
  - In the "Details" window:
    - find the "Geometry" item and click "apply" (This associates the selected edge with the BC)
    - Set the magnitude to be 60 degrees.
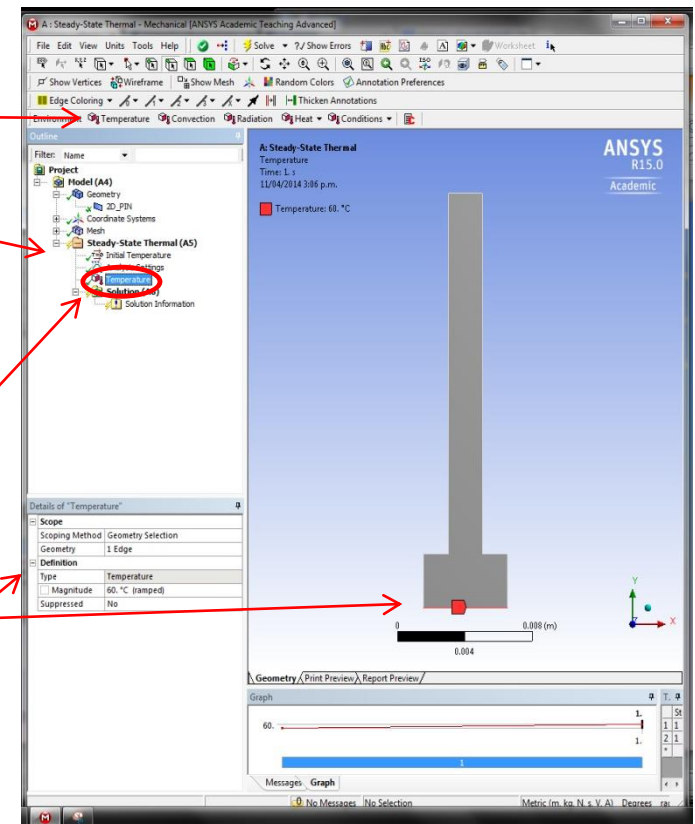
Step 8

• Apply Convection and Heat flow constraints in a similar way.
- • Heat flow
  - • Define as perfectly insulated
  - • Apply to the 2 sides of the fin base
- • Convection
  - • Film coefficient 45 W/m^2
  - • Ambient Temperature 20 degrees
 - • Apply to the remaining 5 edges of the fin.
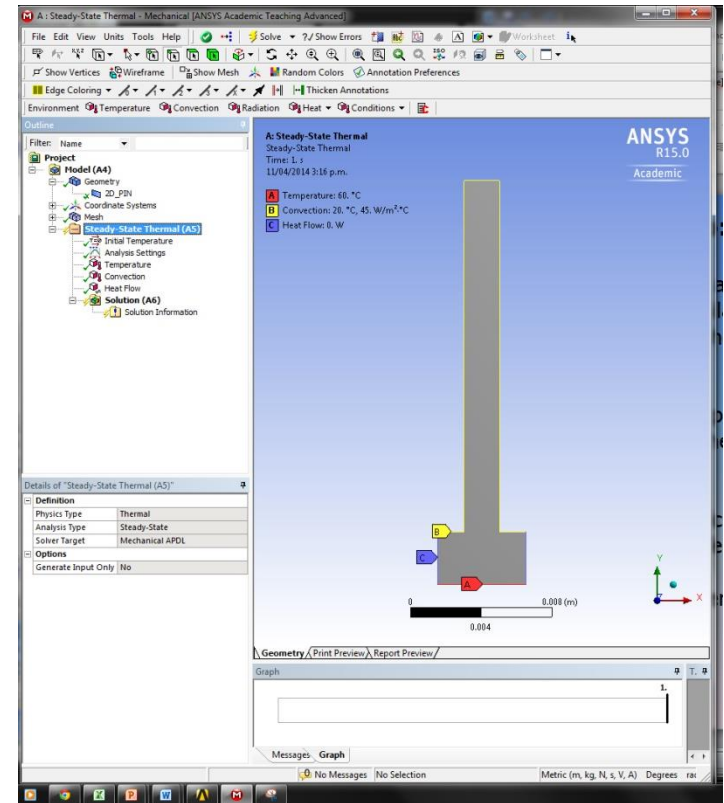
Hint: Hold "Ctrl" while using the edge tool to select multiple edges!

Step 9

Finally, you must specify the results you want to be created and stored by ANSYS Mechanical.

- Establishing results. The **Solution**…

    - Right-click Solution in the Outline
    - Add **Temperature** and then **Total Heat Flux**
    - You will see these results appear in the Outline (although they will not be populated with data until the model is successfully solved)
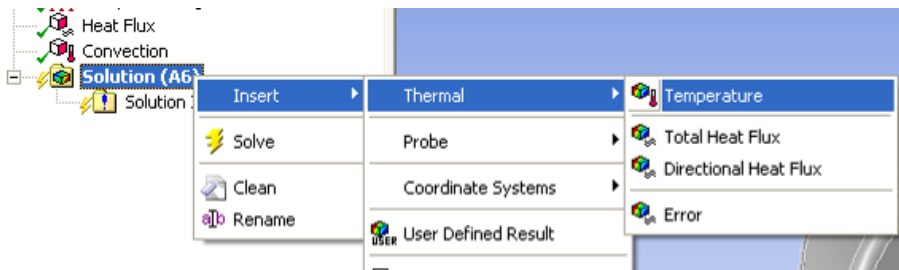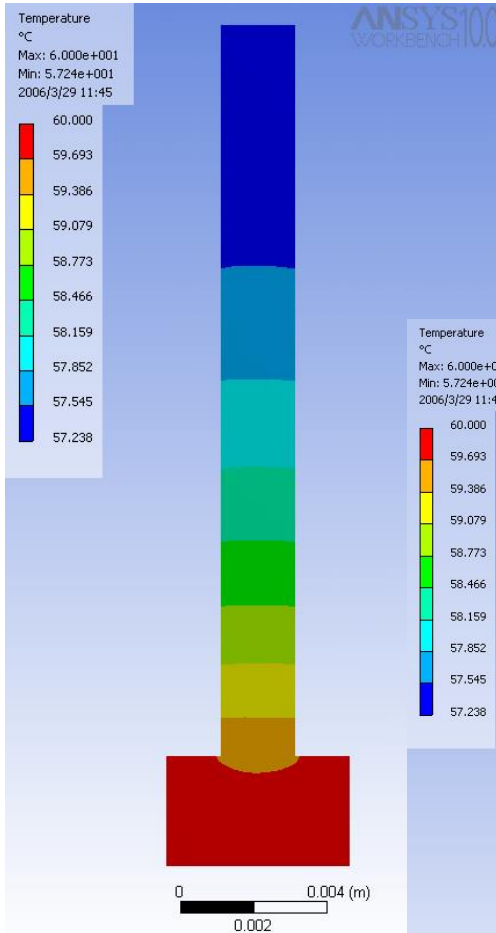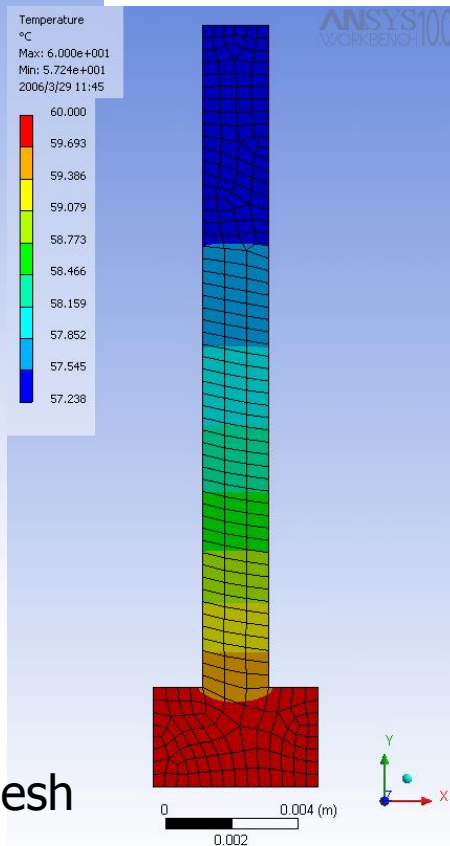

The model is now ready to solve.
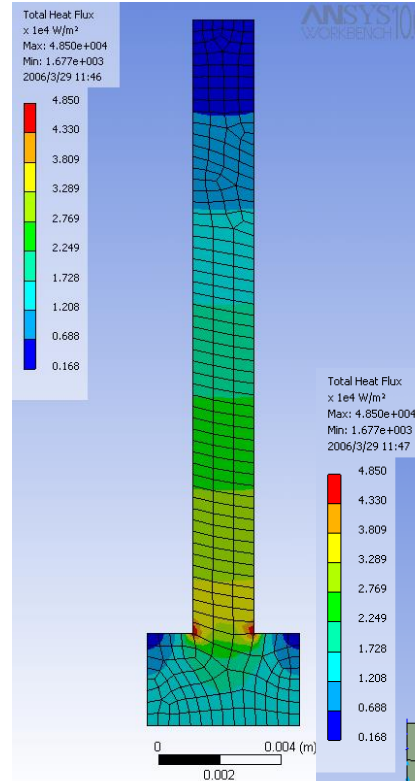
- Hit the **magic solve button**…
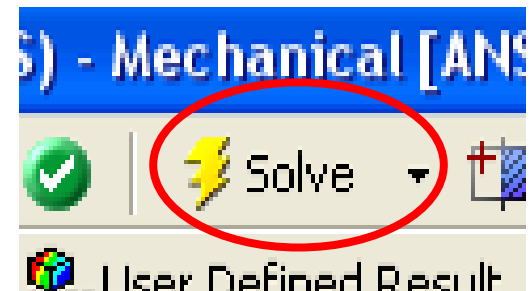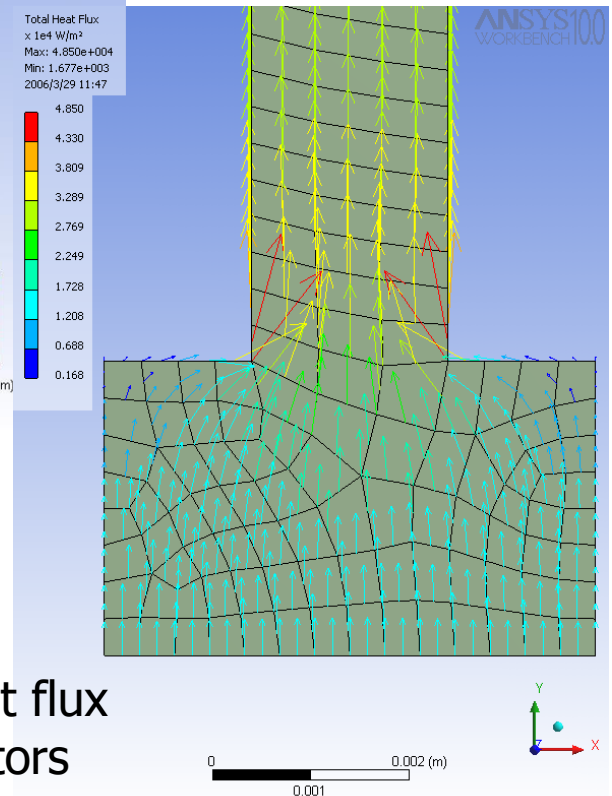
Some example results for the 2D fin

Temperature contour

Temperature contour, with mesh

Heat flux magnitude contour

Heat flux vectors

With Workbench it is quite easy to make changes to your model and see the effects. E.g. you can…

- Change the material

- Delete old BCs, introduce new BCs.
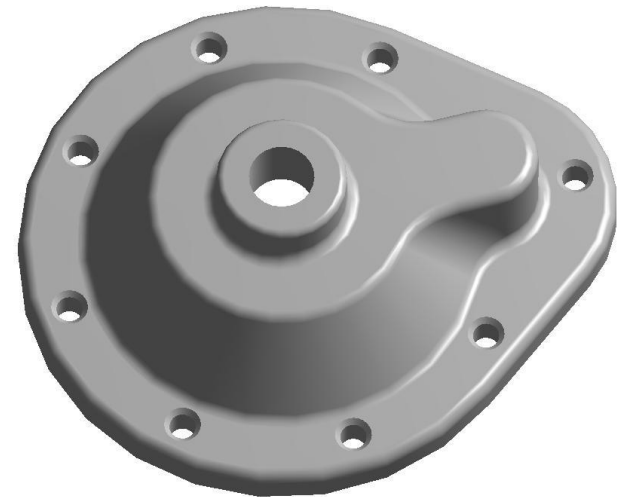
- Introduce different types of results to view.

Once you are happy with your changes, press Solve again.

Elastic Structural Analysis

For an example of structural analysis, we'll look at one part from the pump assembly below.
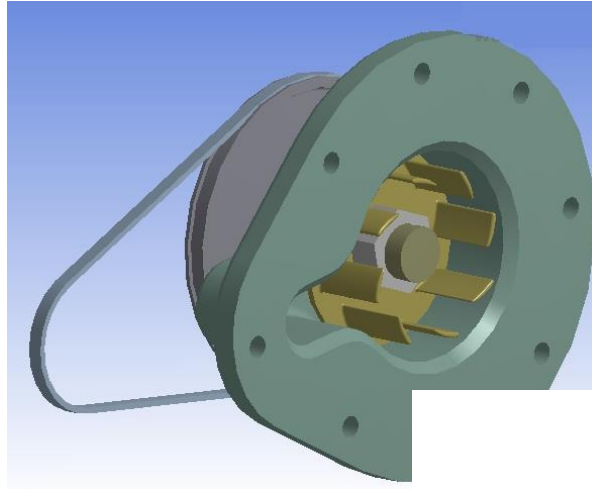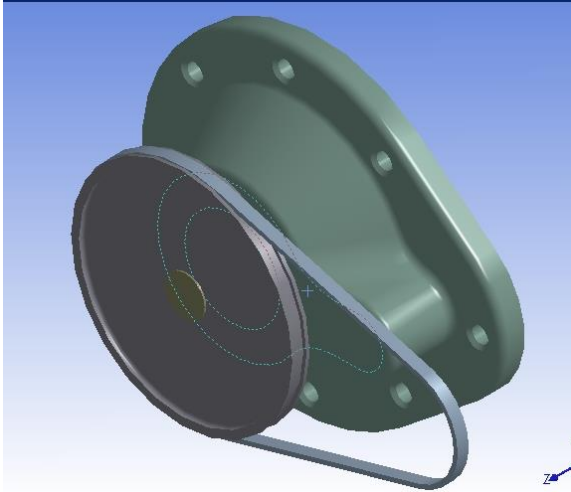
This time we'll carry out a 3D analysis.

Step 1

- Launch **"ANSYS Workbench"** from start menu. Again, you are faced with the Workbench Interface.

- **OR**, while still in the Workbench Interface, choose new from the file menu.

- Drag the "Static Structural" item from the Toolbox into the Project Schematic to create an Analysis System.

- Import the geometry…

Static Structural (ANSYS)

- Right click on Geometry drop down. Select "Import Geometry" > browse
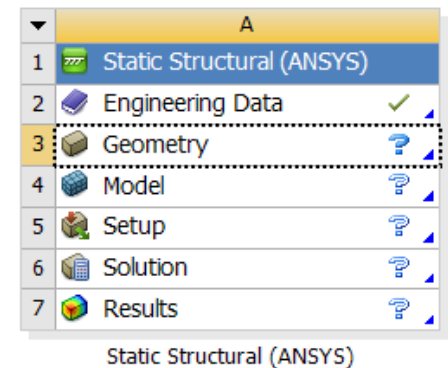
Where you have saved it   .../Tutorial1/Pumphousing.agdb

Step 2

- Selecting a Material

    - Right click on Engineering Data, and select Edit

    - Find the desired material in the libraries, and click on the add button.
        - As previously, except select **aluminum alloy**.

    - To actually add this material to the project, click on Update Project. Then click Return to Project.

- Open the Mechanical window, to start working with the mesh, boundary conditions, results. Right-click on Model, and select Edit.
    - In the Outline, open the Geometry node, then select the part.
    - In the Details window, change the default Structural Steel to the new material.

Step 3

• In the Mechanical Window, right click on Mesh in the Outline -> Generate Mesh

Note: The mesh we have at this stage is what ANSYS is recommending. Often we will want to adjust this. We'll come to that later...

• Application of loads/constraints. The **Environment**...

> • Click on the Environment (Static Structural) in the Outline
>
> • Select the Geometric Entity that you want to apply a condition to. (Is it a volume, surface, edge, or point?)
>
> • From the Environment Bar, choose the type of load/constraint.
>
> • Make sure you fill in all the data required
>
> • Have you got enough constraints?

Step 4

- Application of loads and constraints
  - We will apply a Pressure Load and a Fixed Constraint.

Pressure load…

- Select "Static Structural" in the outline window
- Select the faces shown
  - There should be 19!
- In the Environment bar select "Loads" >> "Pressure"
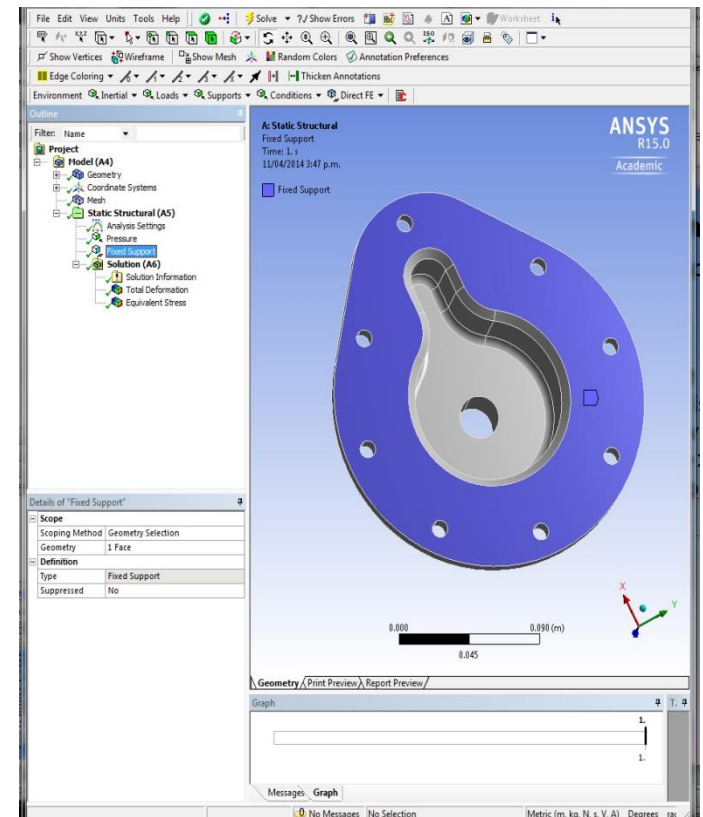- Click "apply" in the Details tab
- Assign a pressure load of 10 kPa
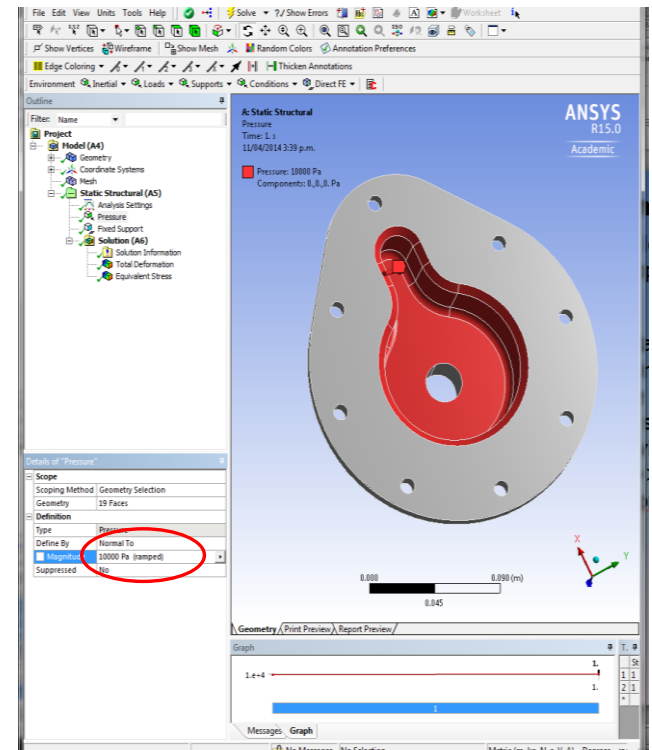
Step 5

- Now we will apply a fixed support in a similar way.

Fixed support…
- Select the face shown
  - There is only 1!
- In the Environment bar select "Supports" >> "Fixed support"
- Click apply

Step 6

Note: Some conditions, such as forces and moments, are vector quantities. You have to specify:

- A magnitude and direction
- **OR**, components (F_x, F_y, F_z)

Step 7

Before solving the model, you need to specify the type of results to plot. Specify some now, and add others later.

- Establishing results. The **Solution**…

    - Right click on Solution in the Outline

    - To start with, I'd suggest you select a **Total Deformation** and an **Equivalent Stress** result.

    - Note, in a 3D example, you have even more options. We will have to think carefully about what is needed in each analysis.
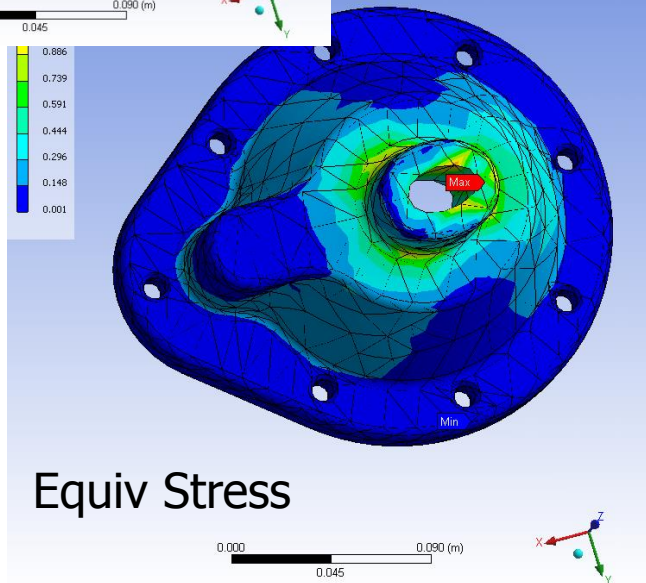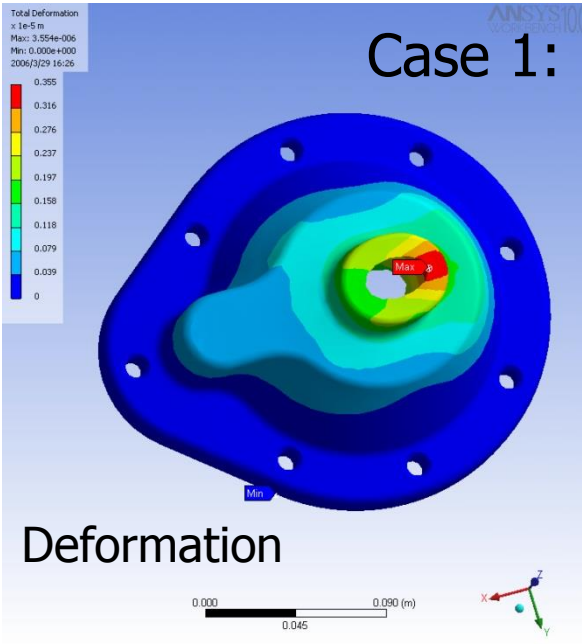
The model should now be ready to solve.
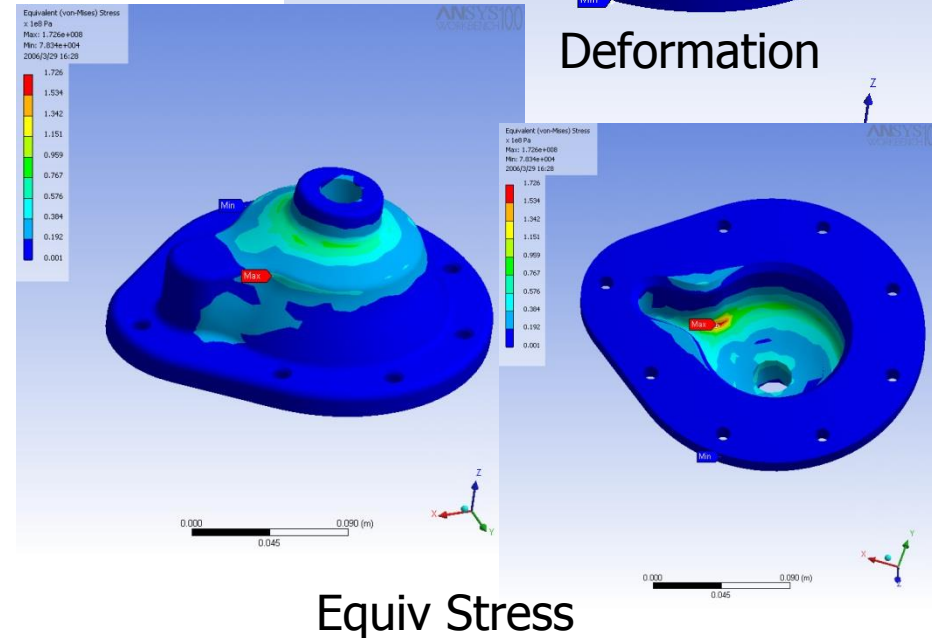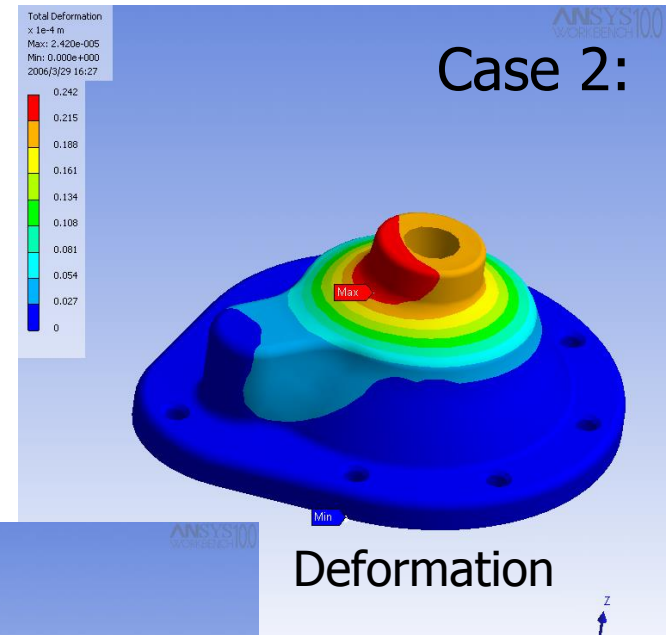
- Hit the **magic solve button**…
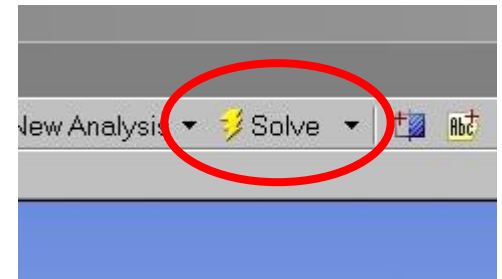
Some example results for the 3D pump housing

Case 1:

Case 2:

Deformation

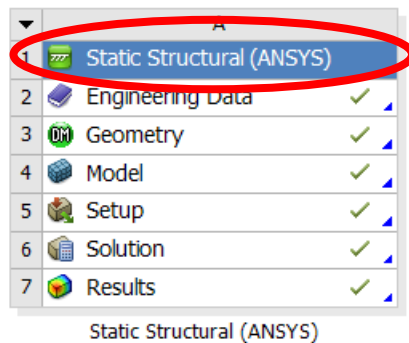Deformation

Equiv Stress

Equiv Stress

Step 8

Note: Perhaps you want to look at a different set of conditions for the same part. To do this, we must return to the Workbench Interface, and create a duplicate of our existing model.
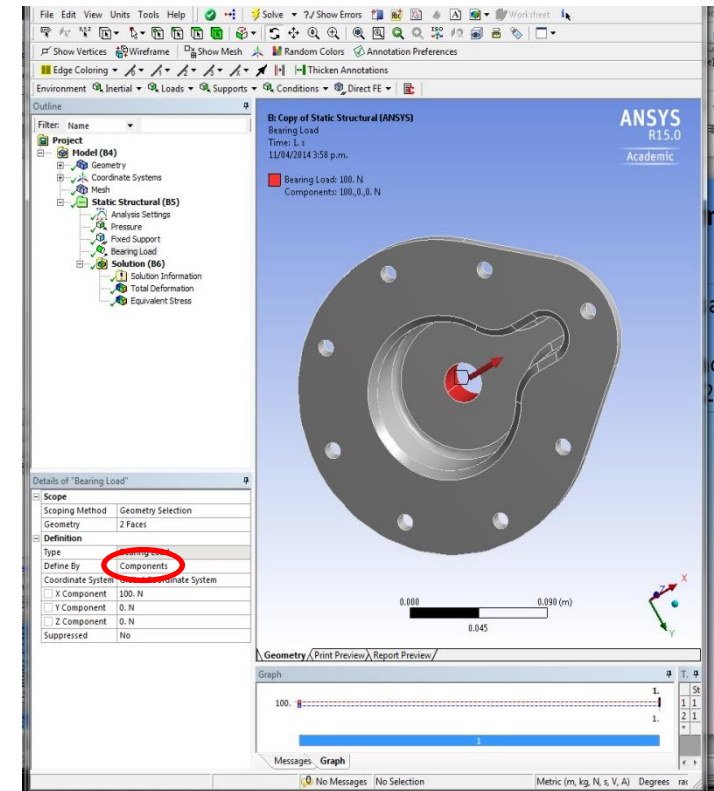

Static Structural (ANSYS)

• In the Interface, right click on the existing model (as shown at left), and choose duplicate.

• To work with this model, right click on model, and select Edit.

• Two load cases are now created within your project.

Static Structural (ANSYS)

Copy of Static Structural (ANSYS)
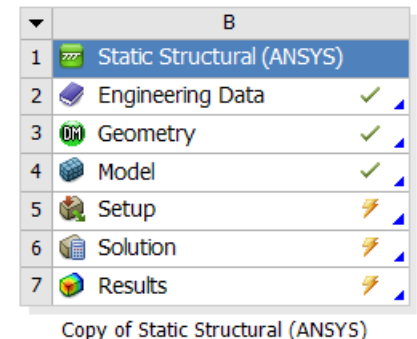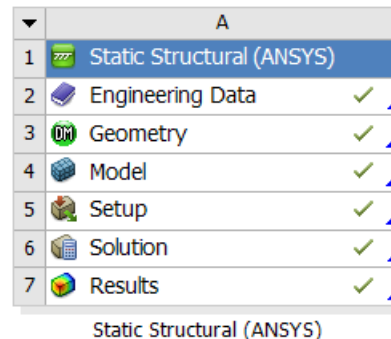
Step 9

• In the second load case try adding a bearing load.

- • Select the faces shown
  - • There are 2 this time!
- • Select "Loads" >> "Bearing" in the Environment bar.
- • Click apply in the Details tab.
- • Under definition, define the load by components.
- • Specify a X component of 100 N.

- • Hit Solve and view the results!

Summary...

This tutorial has:

• Introduced the basic steps of using ANSYS Workbench.
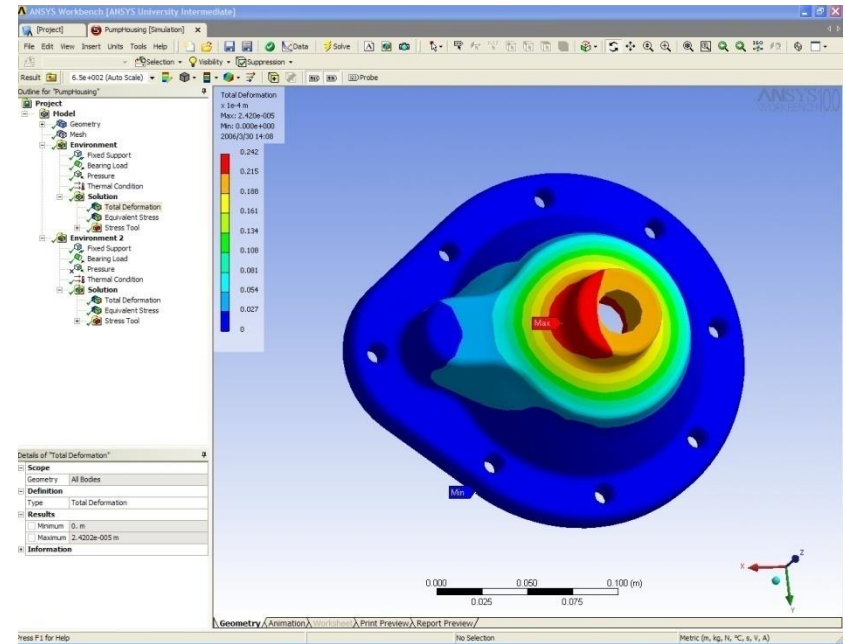
• Walked you through examples of:
- • Heat Conduction Analysis
- • Elastic Structural Analysis

• Demonstrated the vast array of results that you can collect from FEA.

All done! Now move on to Tutorial 2…