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

Tutorial 2: 3D heat conduction / elastic structural analyses
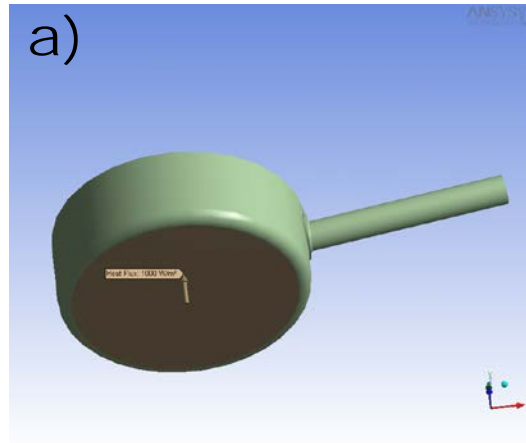
Dr David Wynn
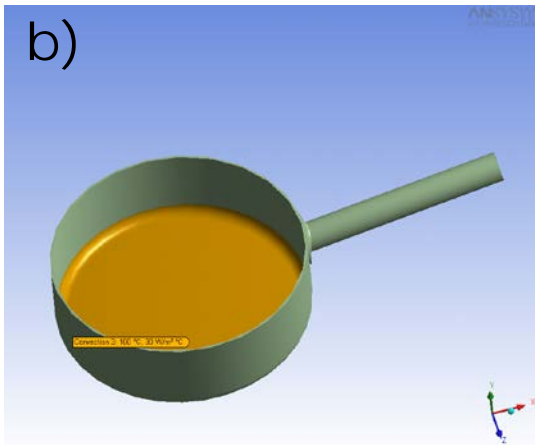
Email: d.wynn@auckland.ac.nz

Room: 401.916

Example Heat Conduction Analysis, 3D

We'll perform a simple analysis of an aluminium saucepan.
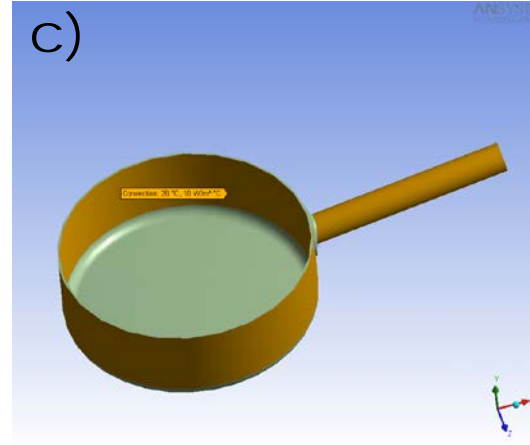For the specified conditions, would you burn your hand?

What conditions will we apply?

a)

Heat flux at the base, due to the heating element.

b)

Convection due to a small amount of boiling water.

c)

Convective cooling at the majority of other surfaces.

Getting Started

• Download Tutorial 2 files from Canvas. Save to local disk.

• Launch ANSYS Workbench from the Start Menu.
Click All Programs >> FOE Applications and Documentation

• Double-click the "Steady-State Thermal" system **or** drag it into the Project Schematic box.

• Import the geometry...

  • Right click on "Geometry"

  • Select "Import Geometry"

  • Browse for the following file...
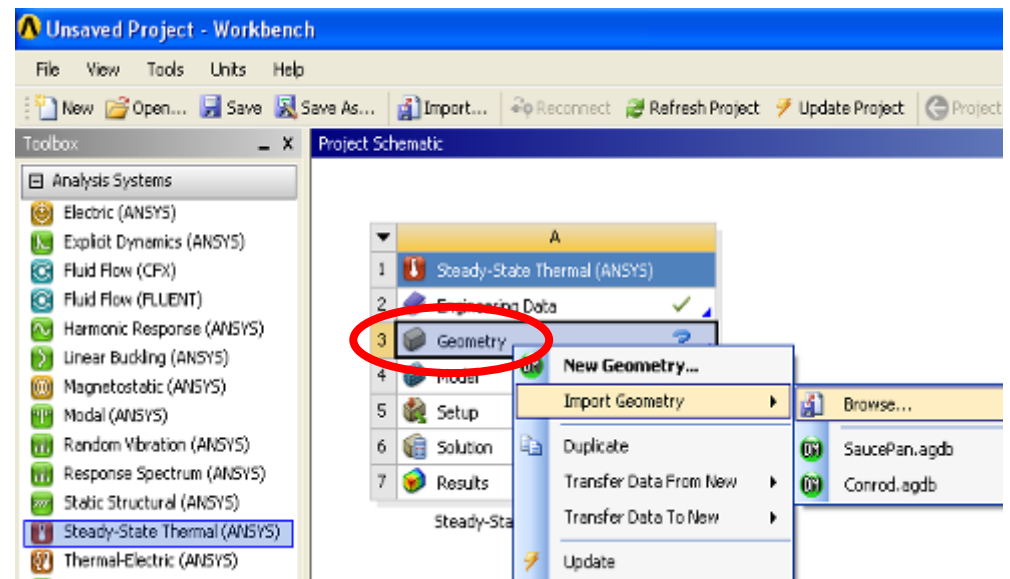
Where you have saved it   .../Tutorial2/Saucepan.agdb

NOTE: *.agdb files are Workbench geometry files.

IMPORTANT! Location of files for working.

It is **vital** that you now save the project in a temporary folder, on your desktop hard drive. This will...

- Provide the best performance when you solve models.
- Avoid clogging the network as calculations are performed.

- Choose "Save as" and save the project file to a folder on the local harddrive.

- You might want to create a folder for yourself under D:/Scratch
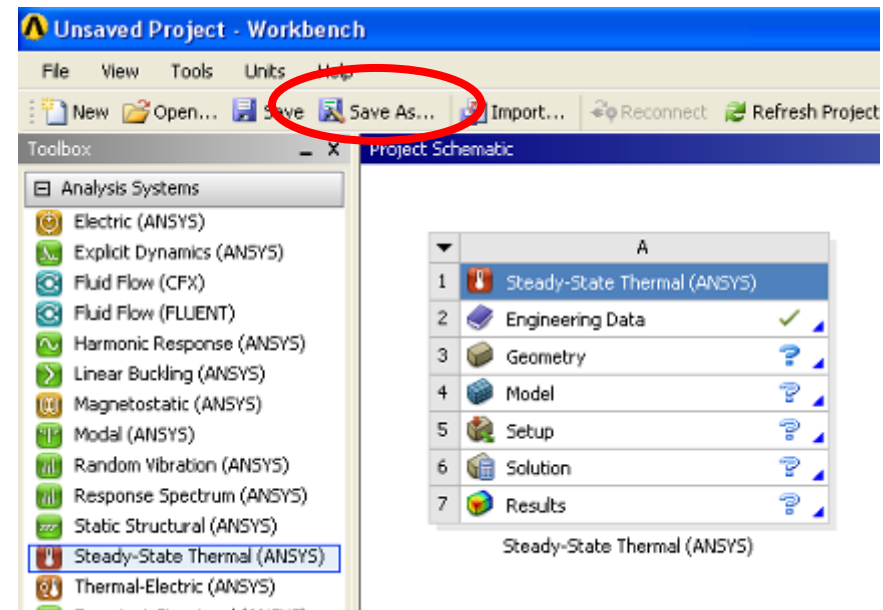
Hint: At the **end of today's session, BEFORE YOU LOGOUT**, copy your folder to your H drive or a USB stick if you want to keep your work.
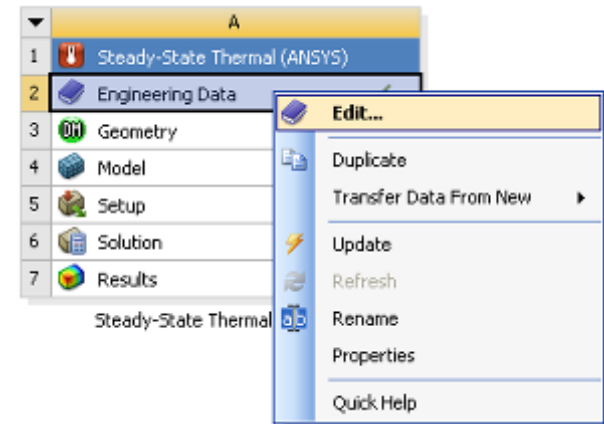
ANSYS Mechanical application

- You can edit your simulation using ANSYS Mechanical
  - Right clicking on any of steps 4 – 7 opens a dropdown with the M icon.
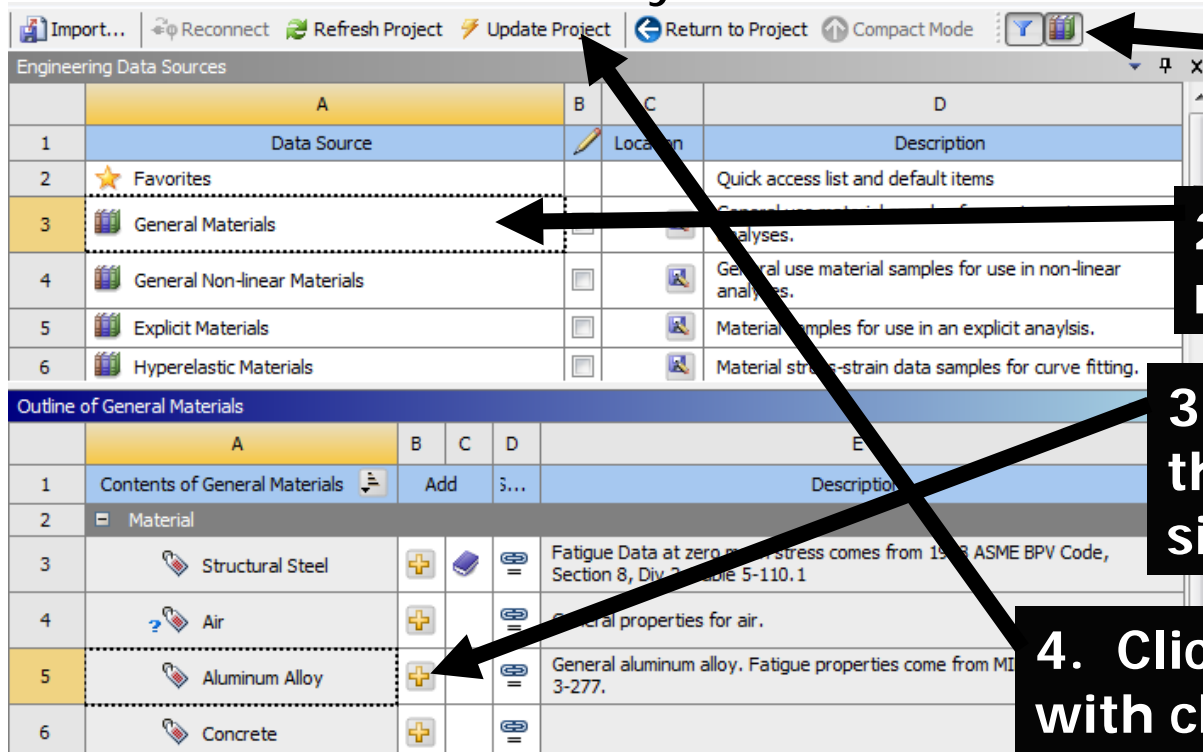 - Click "Edit..." to open the ANSYS Mechanical application
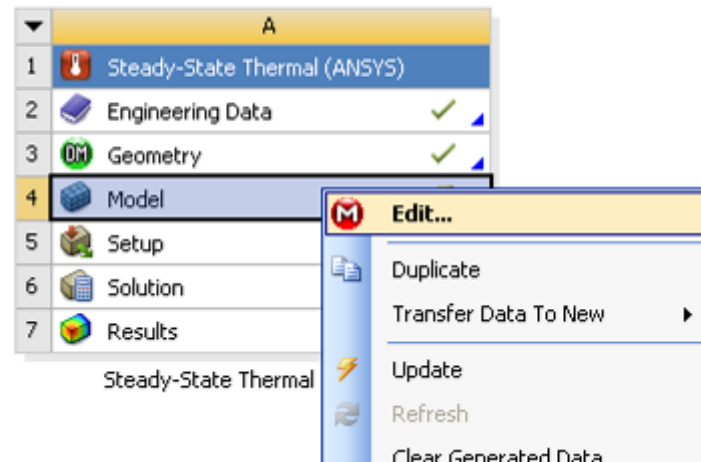
Editing Material Data

- To configure the materials accessible to ANSYS Mechanical
  - Right click [M] step 2 – "Engineering Data"
  - Click "Edit..."

- Add Aluminium alloy to the materials accessible in [M]...

1. Toggle the data library ON
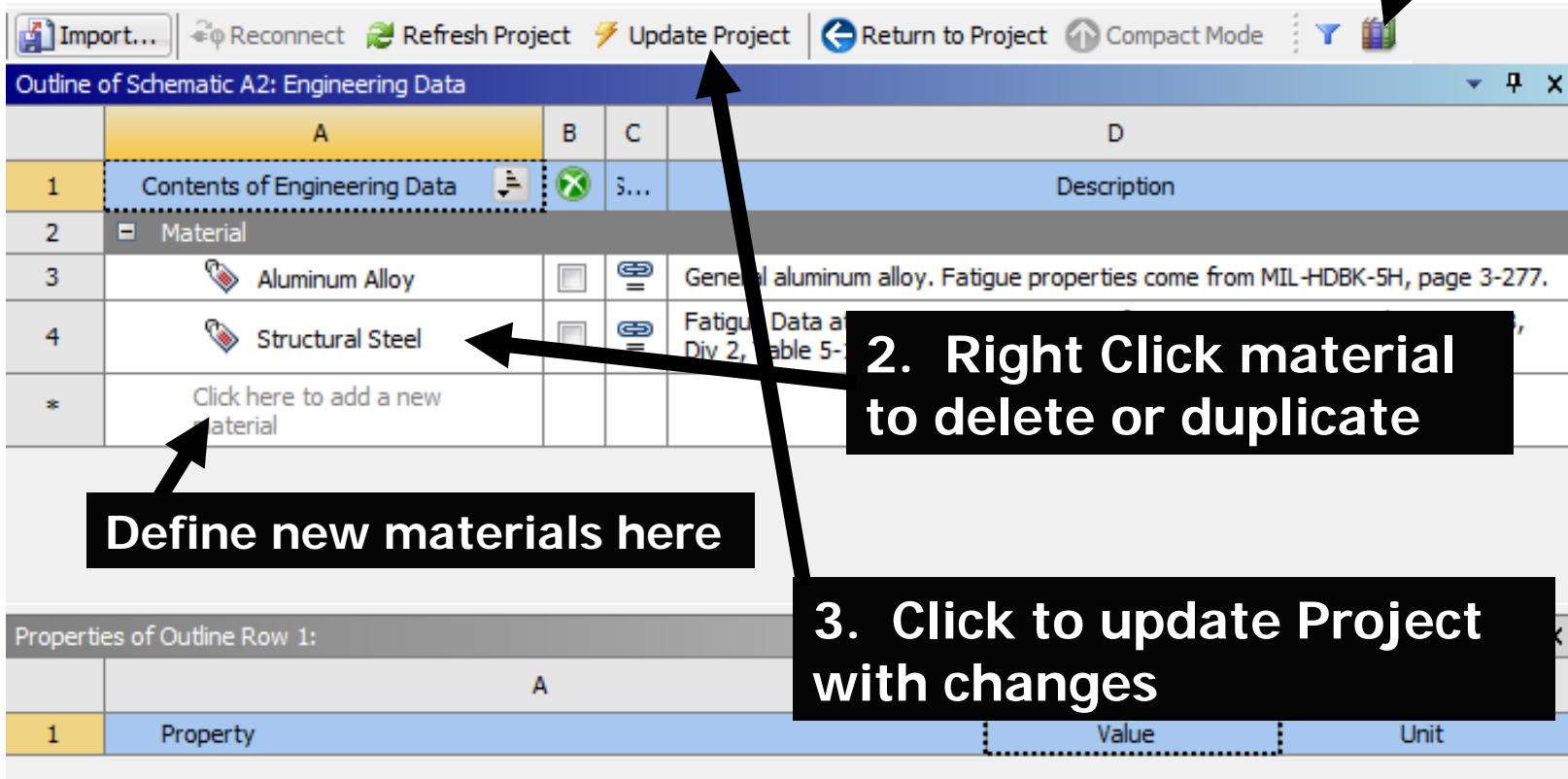
2. Click a source of materials

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

4. Click to update Project with changes

Editing Material Data

• Deleting materials accessible from and defining new materials

1. Toggle the data library OFF

2. Right Click material to delete or duplicate

Define new materials here

3. Click to update Project with changes
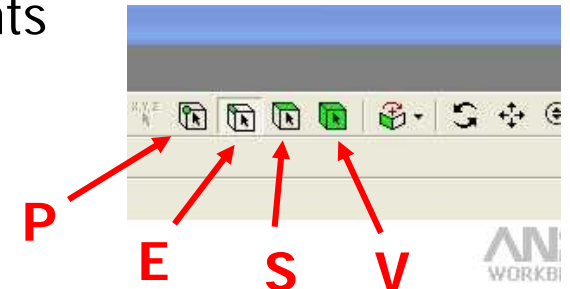
Within ![M] , in your own time...

- Explore how to **manipulate the model**.
  - Rotation, zoom, etc. similar to Creo
  - Use tools in top bar...
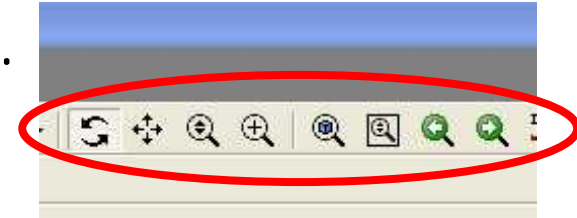
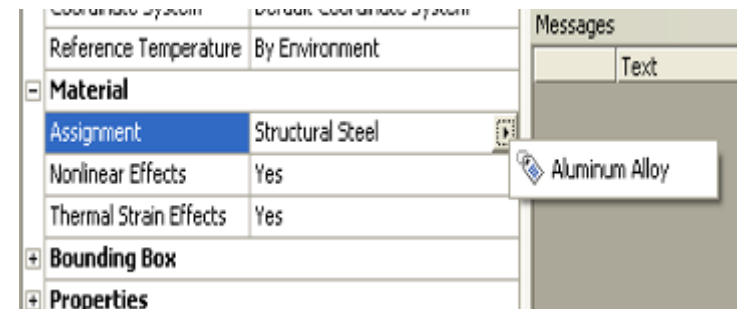- Explore how to **select different parts of the model**.
  - Select; Volumes, Surfaces, Edges, and Points
  - Use these selection tools...

  P   E   S   V

- Applying **Material Properties**

  - Expand Geometry in the Outline
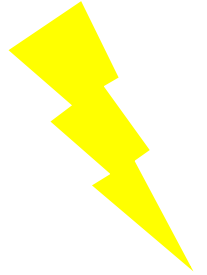 - Click on the **Part** in the Outline (it should be highlighted)

  - In the Details menu, click on the current material (structural steel). Change the material to **Aluminium Alloy**

In your own time...

Remember: The "lightning" symbol indicates that something needs your attention!

How does the mesh appear?

- Right click on Mesh in the Outline... hit "Generate Mesh"

- Examine the mesh that ANSYS Workbench has suggested.

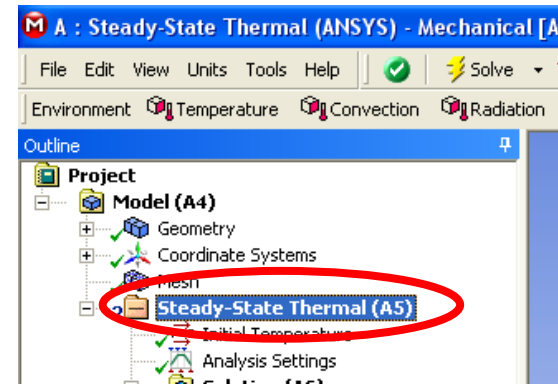- We'll use this for now, and consider mesh refinement later.

NOTE: You could carry on with the analysis without even considering the mesh.

Not a good idea...

In your own time…

- From the outset, we chose a steady state thermal analysis
- Application of loads/constraints. The **Environment**…

- In , click on "Steady-State Thermal" in the "Outline" box.

- Select the Geometric Entity that you want to apply a condition to. (In this case we'll deal with surfaces).
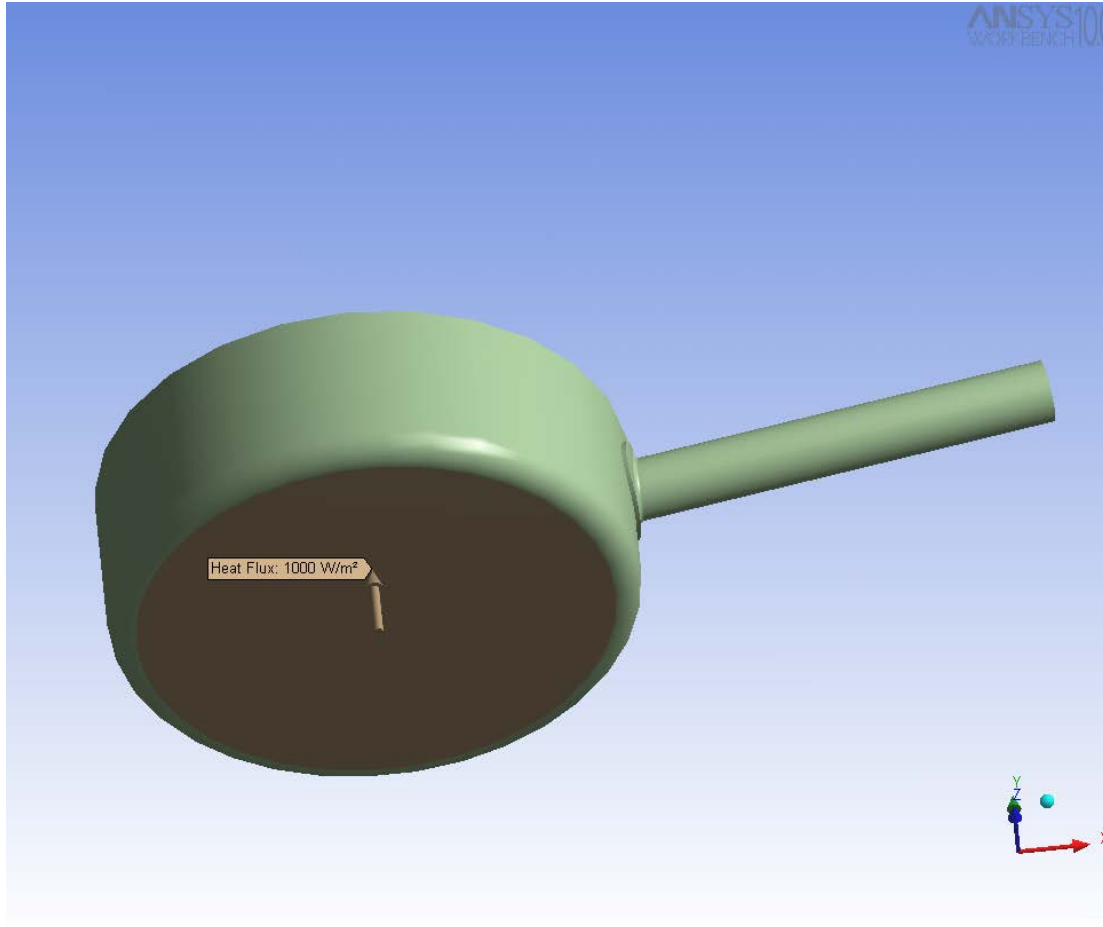
Reminder: If you are applying the same condition to multiple surfaces, hold the CTRL key to pick all of them!

- From the Environment bar, choose the type of load/constraint…
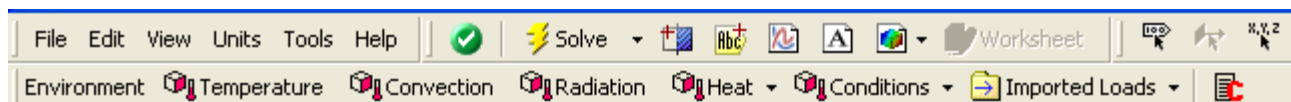
In your own time…

- Modelling heat transfer from the stove top into the pan

Heat Flux: 1000 W/m²

- Pick the exterior bottom surface of the pan.
- Select heat flux from the "Heat" drop down menu in the Environment bar.
- Enter 1000 W/m² in the details area.

In your own time…

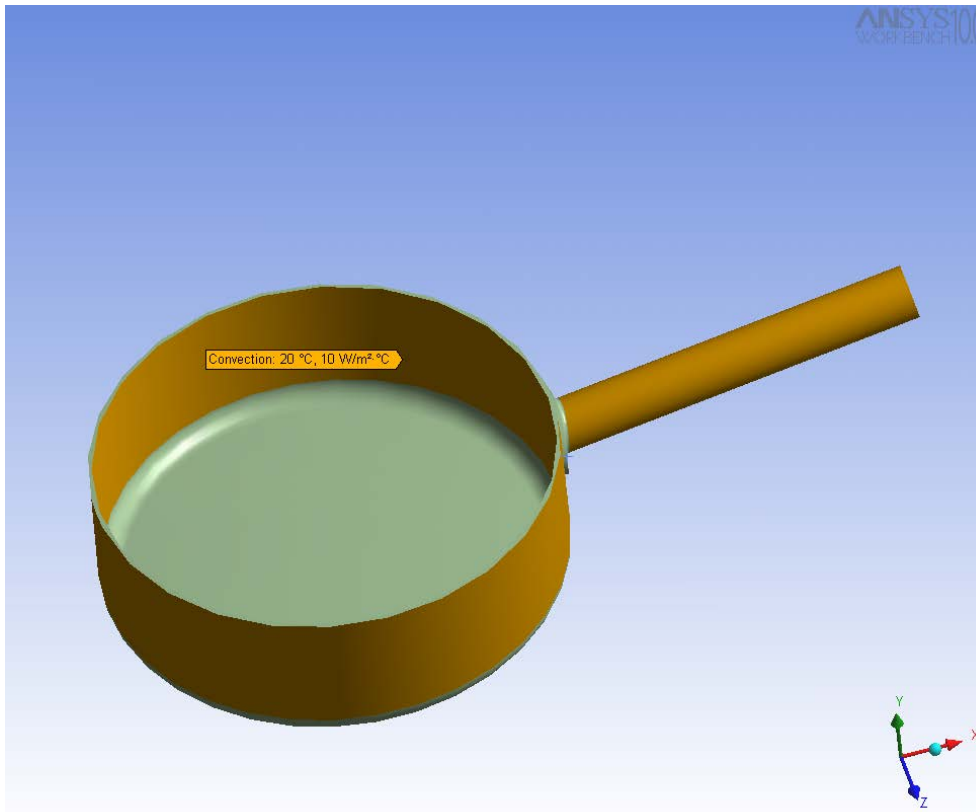• Modelling heat transfer from pan into the boiling water

Convection 3: 100 °C, 30 W/m².°C

• Pick the surfaces shown.

• Select convection from the Environment bar.

• Enter a film coefficient of 30 W/m²°C, and an ambient temperature of 100°C.

In your own time…

- Modelling heat transfer from the pan into surrounding air

- Pick the majority of the remaining surfaces.

- Select convection from the Environment bar.

- Enter a film coefficient of 10 W/m^2°C, and an ambient temperature of 20°C.

- Click on "Steady-State Thermal" in the "Outline" box to review the load/constraint environment you have set.

In your own time…

Before solving the model, you need to specify the type of results to plot. Specify some now, and add others later.
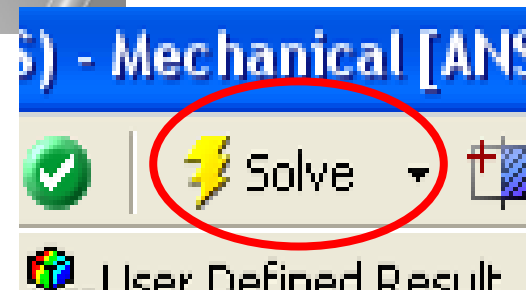
- Establishing results. The **Solution**…

  - Right-click on Solution in the Outline

  - Choose the following results to present:

1) Temperature
2) Total Heat Flux
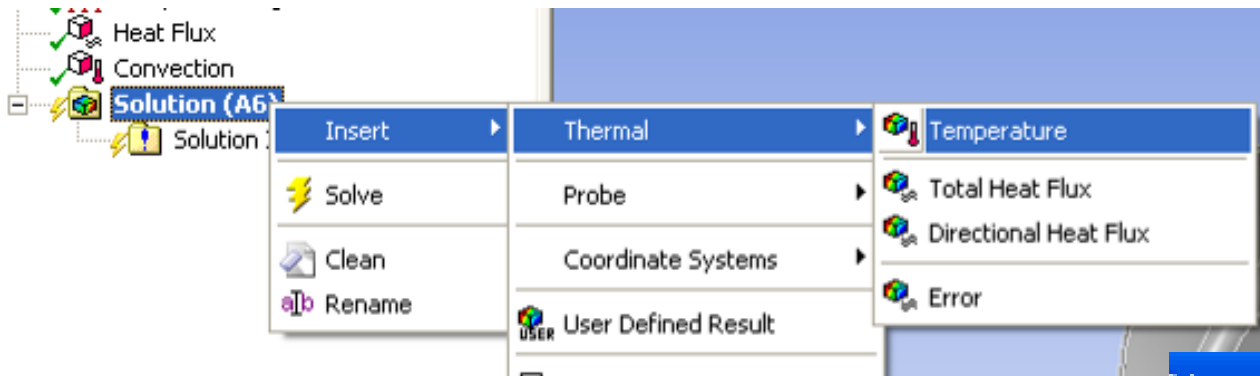
The model is now ready to solve.

- Hit the **solve button**…

In your own time…

The results can now be examined.

  • Expand the **Solution** folder within the **Outline** box.

  • Click on each result type under Solution, and explore them. Zoom, rotate. Where are temperature and heat flux **maximum** or **minimum**? Look at **heat flux vectors**.

FINISHED EARLY? Before we move on, try the following:

  • Change the material. How does it affect the results?
  • Try modifying the boundary conditions. How are the results changed?
  • Duplicate your simulation from the Workbench window. In this 2nd simulation, delete your old boundary conditions, and try some new ones.
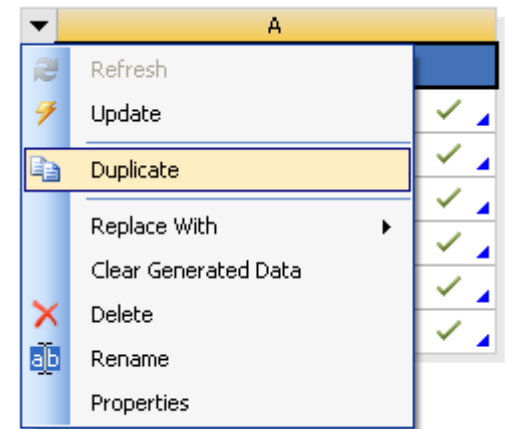
Once you change anything, you will need to solve again!

Example Elastic Structural Analysis

For 3D elastic structural analysis, we'll look at the conrod from the slider/crank mechanism below.

Conrod

5000N Force

Fixed Supports:
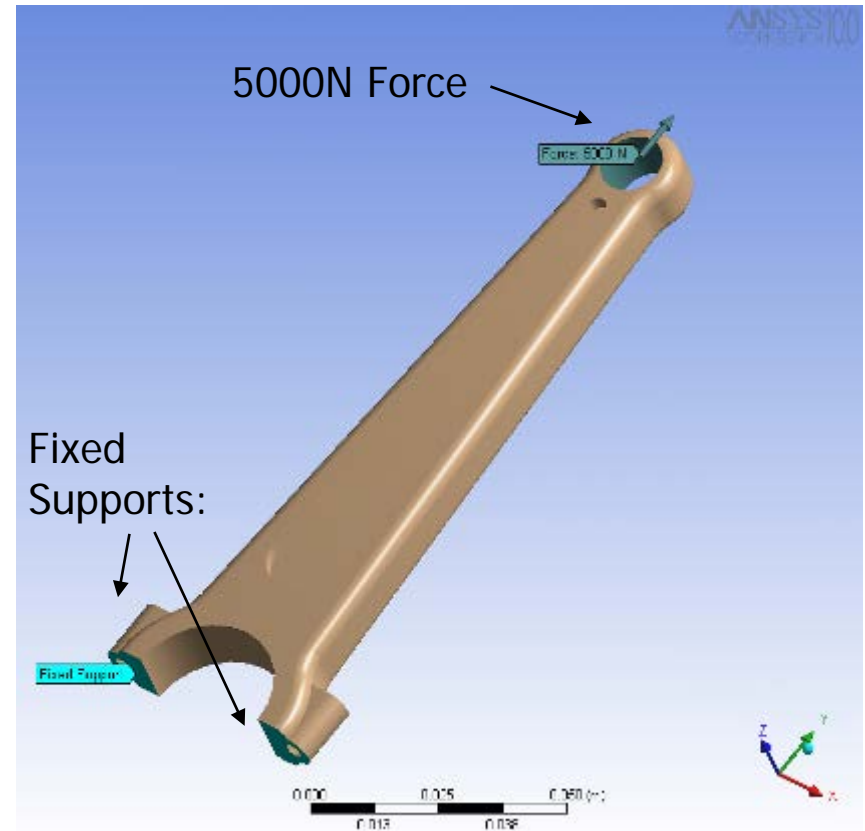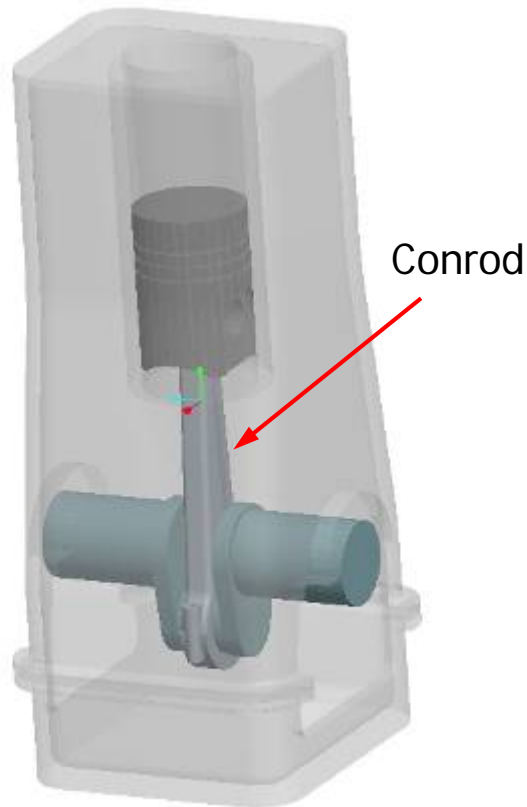
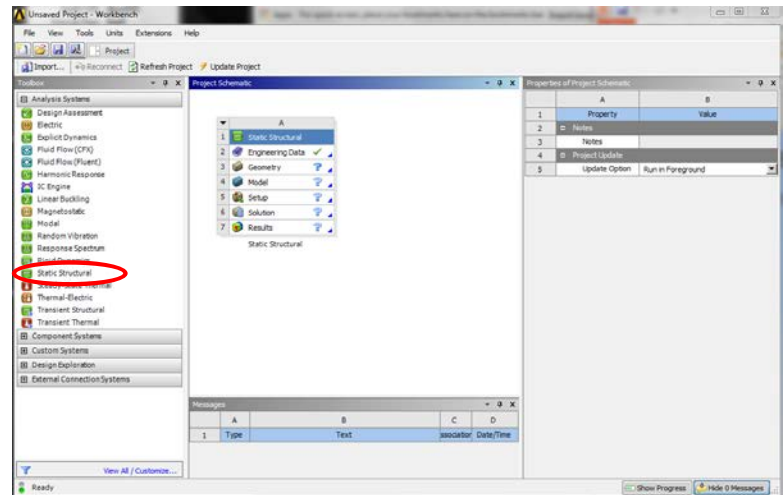Initially, apply the conditions shown above.

In your own time… (Step by step instructions follow)

• Save the previous project. File>Save within the ▲ or Ⓜ windows

• Start a new project. Within Workbench ▲ , click "New"

• Create a new analysis, this time **Static Structural**

• Import the geometry…

• Apply Material Properties… maintain structural steel

• Generate mesh

• Set the loads/constraints…

• Establish the result types…

       1)   Total deformation
       2)   Equivalent stress
       3)   Normal stresses, σ_x, σ_y, σ_z

In your own time…

• Launch Ansys Workbench or if already running, create a new model.

• Double-click the "Static Structural" system **or** drag it into the Project Schematic box.

In your own time…

- Import the geometry...

  - Right click on "Geometry" and select "Import Geometry"

  - Browse for and select the file that you have downloaded

Where you have saved it   …/Tutorial2/Conrod.agdb

**NOTE:** *.agdb files are Workbench geometry files.
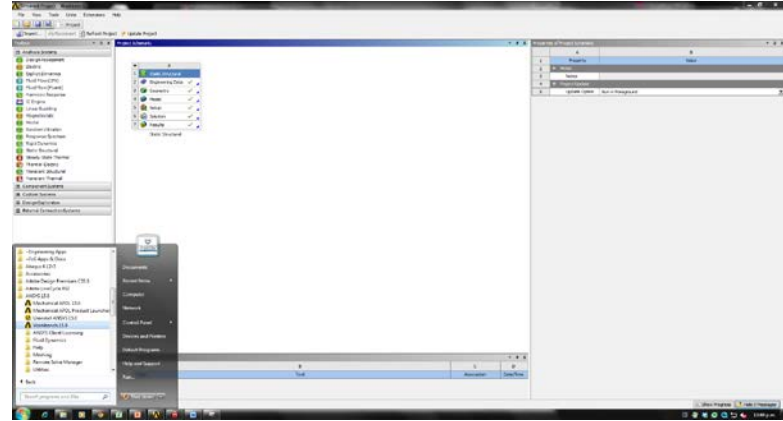
IMPORTANT! Location of files for working.

REMINDER: It is **vital** that you now save the project in a temporary folder, on your desktops harddrive. This will...

- Provide the best performance when you solve models.
- Avoid clogging the network as calculations are performed.

- Choose "Save as" and save the project file to a folder on the local harddrive.

- You might want to create a folder for yourself under D:/Scratch
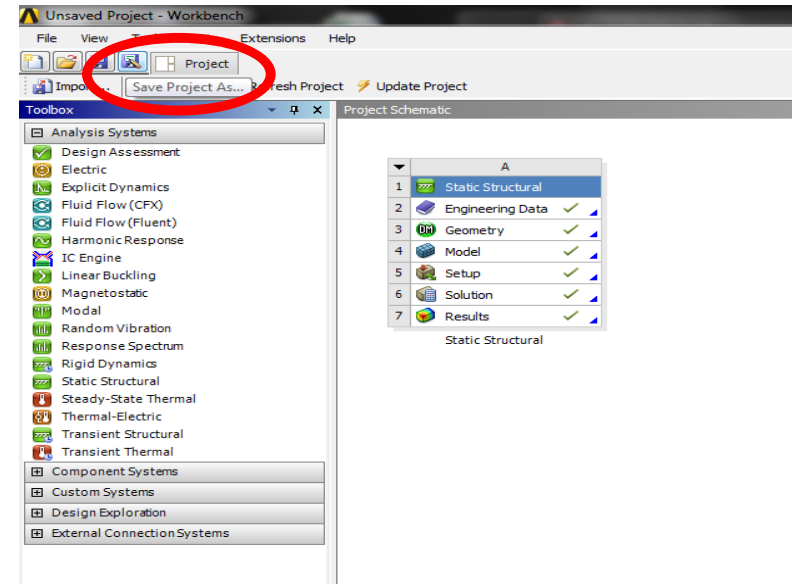
REMINDER: At the **end of todays session, BEFORE YOU LOGOUT**, copy your folder to your H drive or a USB stick if you want to keep your work.
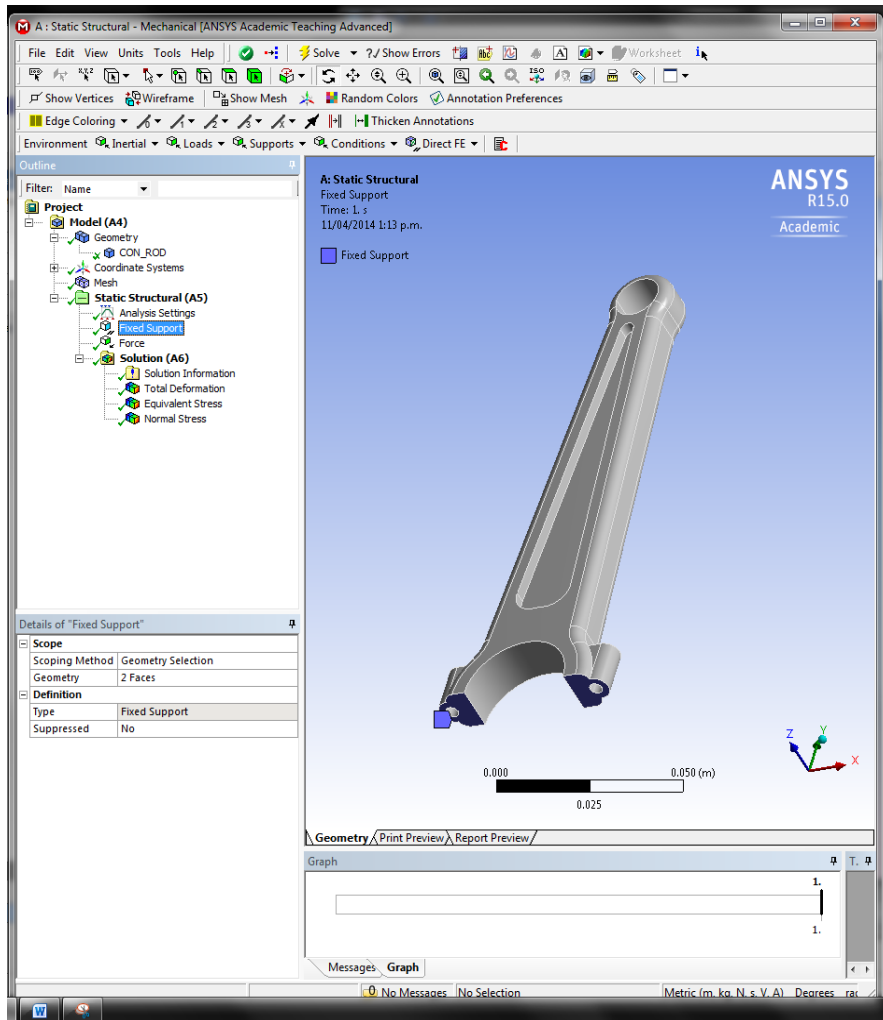
In your own time…

• Right click on model and select edit.

    • This will open the "Mechanical" window

    • (You can right click on any of the steps 4-7, they all open this window)

•Expand the "Geometry" tab and select the part.

    • Check the material is Structural Steel.

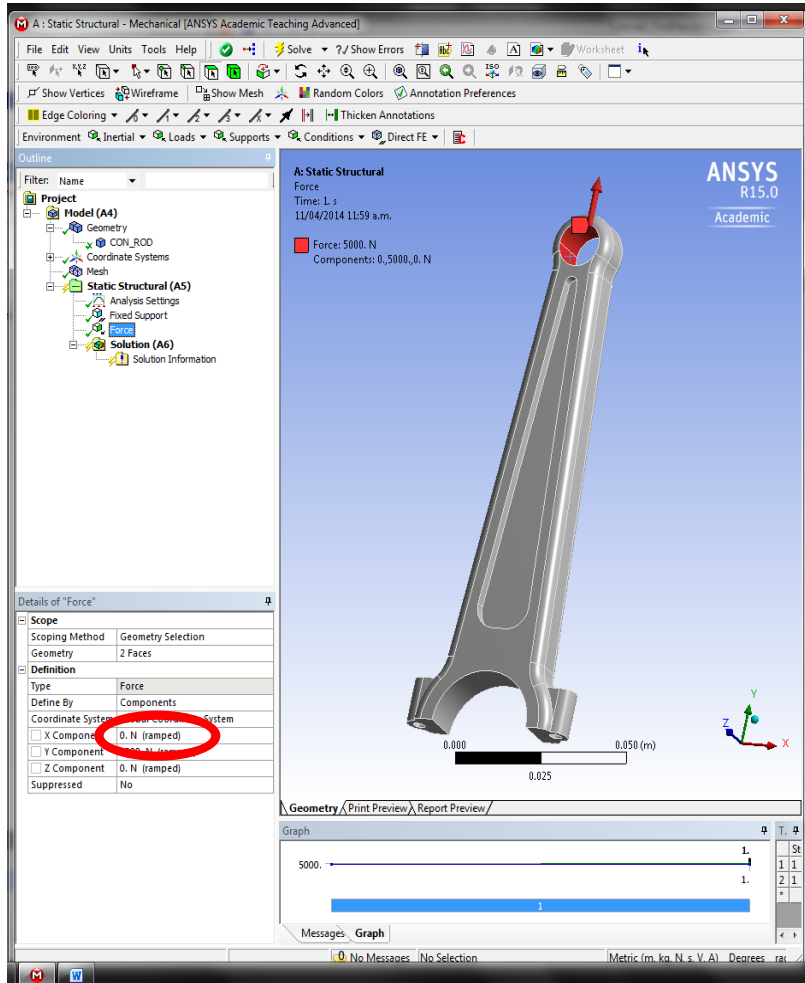•Right click on "Mesh" and generate the mesh by selecting "Generate mesh"
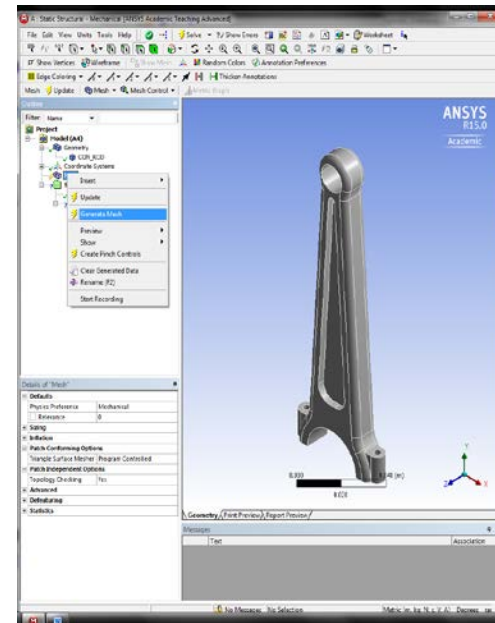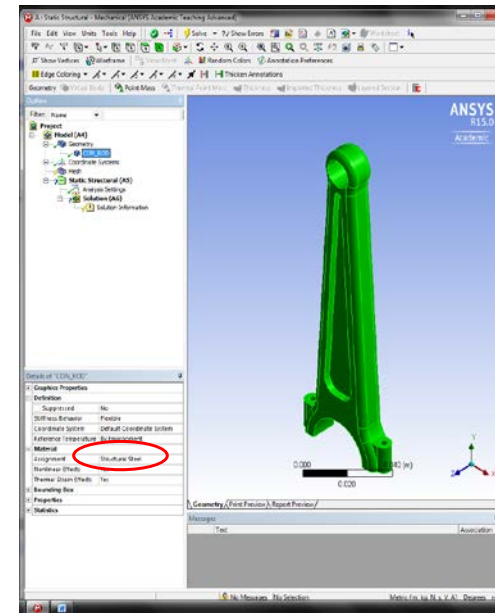
In your own time…

- Fixed Support condition…

• Click on "Static structural" in the "Outline" window to apply the BCs and Loads.

• Select the surfaces shown

• From the Environment bar select "Fixed Support".

In your own time…

- Force application…

- Select the surfaces shown

- From the Environment bar select Force

- Under the Definition tab define the force by components

- Apply a 5000 N force in the Y direction

In your own time...

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

  - Right-click on Solution in the Outline

  - Choose the following results to present:

    1) Total deformation
    2) Equivalent Stress
    3) Normal Stress

  - Hit solve! View the results.
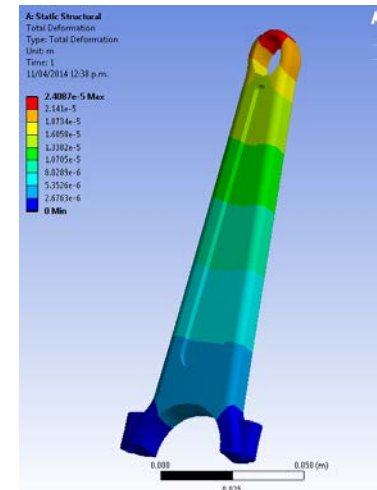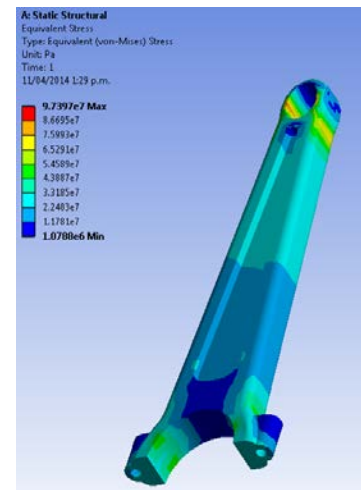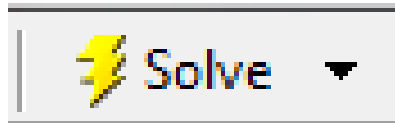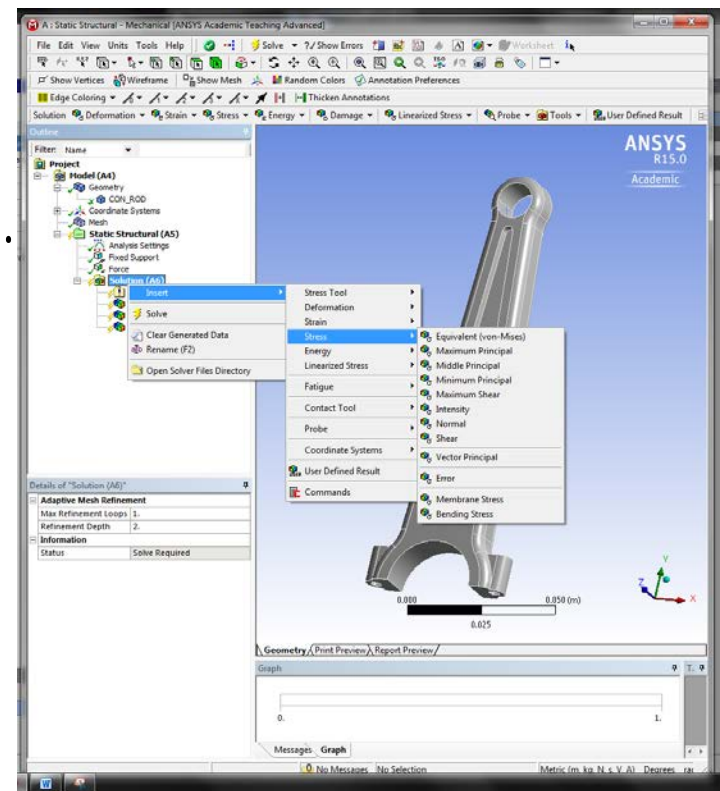
In your own time…

The results can now be examined.

• Expand the **Solution** folder within the **Outline** box.

• Click on each result type under Solution, and explore them. How does deformation look? How smooth are stress contours? Where are stresses maximum? Can you animate the deformation?

FINISHED EARLY?

• Try modifying the boundary conditions. Apply a displacement rather than a force. How do results change?
• Try different forces and moments/torques.
• Try applying an acceleration.
• Look at magnitudes of stress. Do they look too high for steel?

Once you change anything, you will need to solve again!
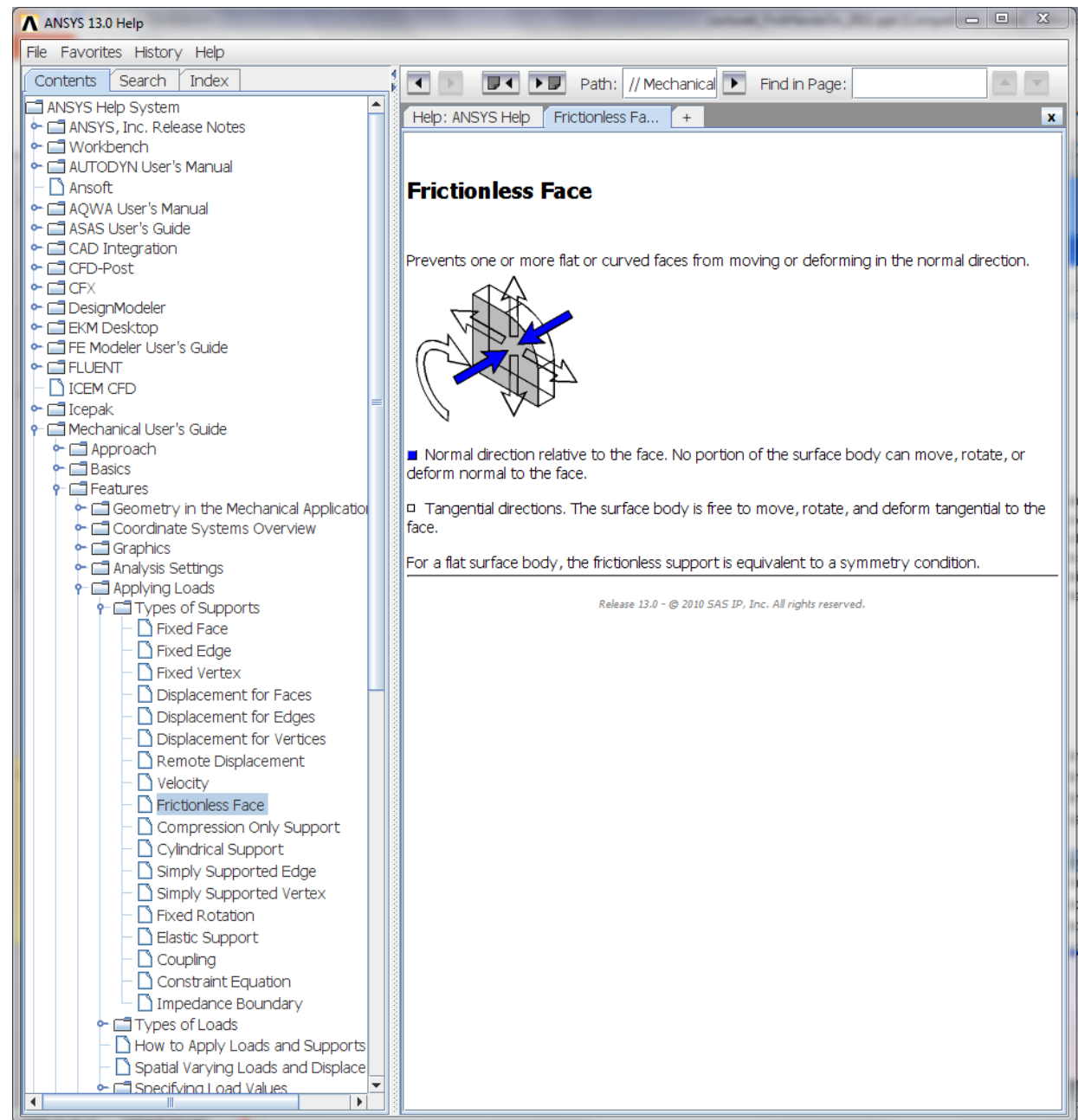
ANSYS Workbench Help
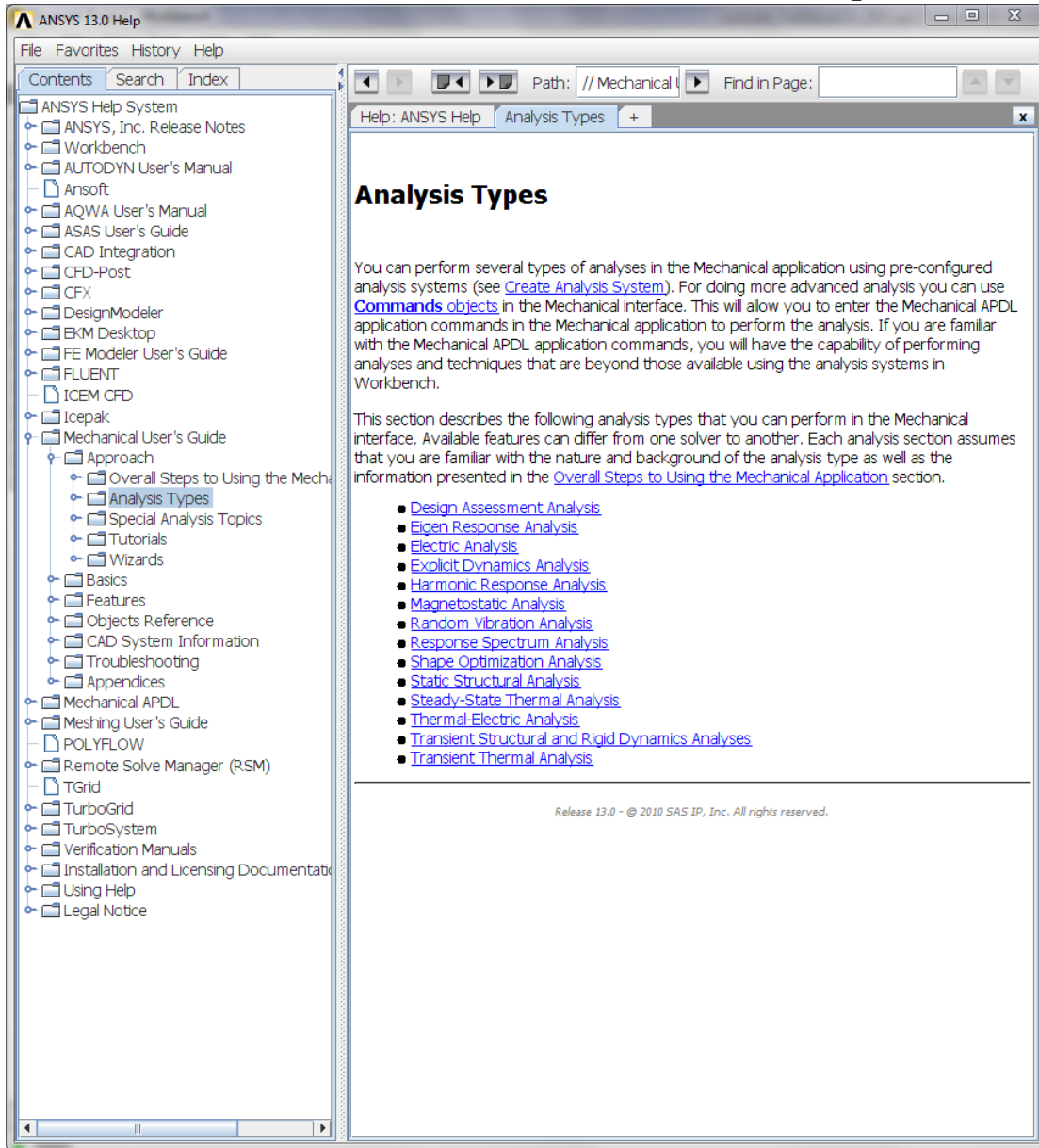
You will find Help to be a very useful resource.

Launch it, and note the topics listed under "Mechanical User's Guide".

Use Help to read more about the types of **loads** and **supports** that can be specified in ANSYS.

For example, this page explains the "Frictionless Face" condition.

All done for this week!

You should now be familiar with the Workbench interface.

Next time, you'll learn to analyse your own geometry created in Creo.

You'll learn everything you need to start designing your Brake Lever, and some people will start on that during the tutorial session. So, be sure you sit next to your project partner at the next tutorial session!