

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

Tutorial 4: FEA and the design process

Dr David Wynn

Email: d.wynn@auckland.ac.nz

Room: 401.916

So far we have covered...

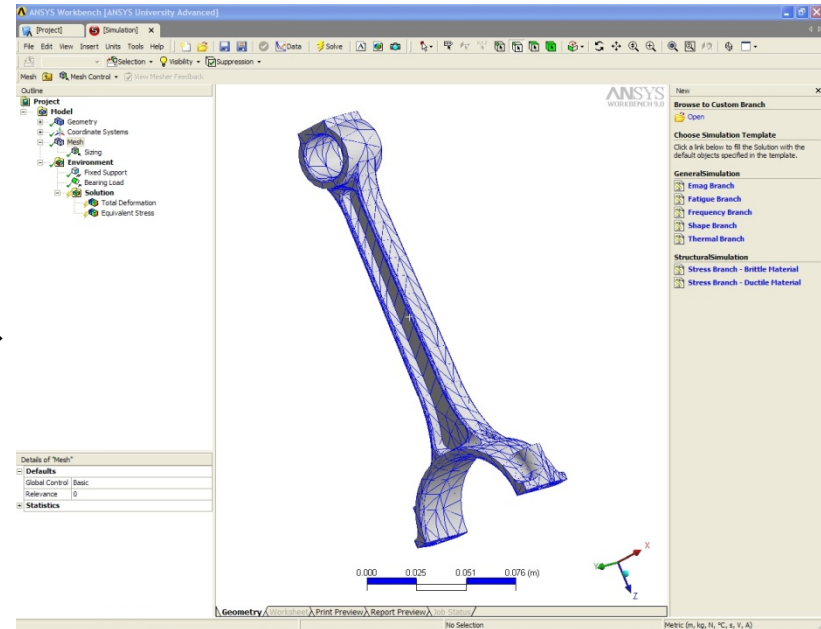
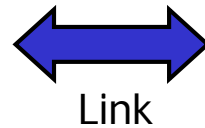
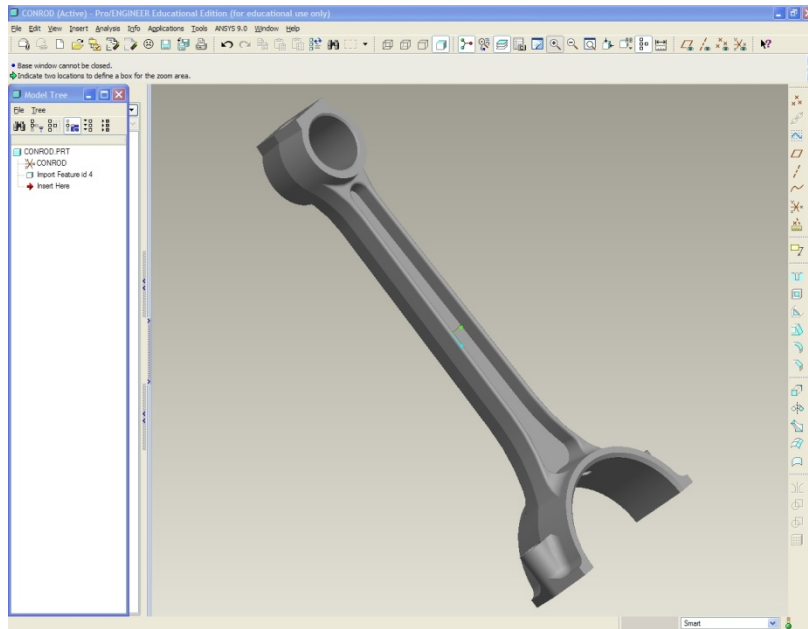
- The basics of FEA, and how to use ANSYS Workbench.
- The importance of appropriate loads/constraints.
- Mesh refinement and using resources efficiently.

This (final) tutorial

- Working efficiently with Creo and Ansys Workbench.
- Tutorial exercise on the Creo/Ansys link.
- Verification of FEA analyses. How realistic are your results? Two recommended approaches.
- Tutorial exercise on FEA analysis verification.

The link between Creo and Workbench

ANSYS Workbench can be used in conjunction with a variety of CAD packages, including Creo.



Using Creo and Workbench together, you can analyse designs as you make modifications, iteratively exploring how changes affect the design performance.

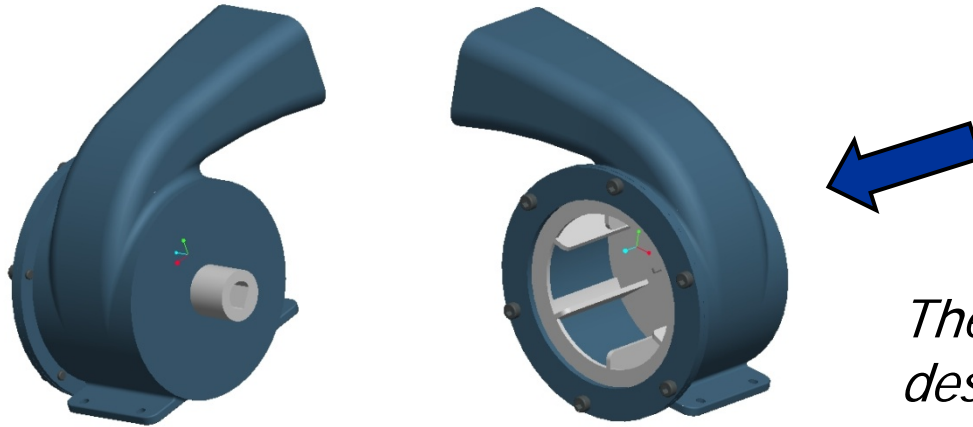
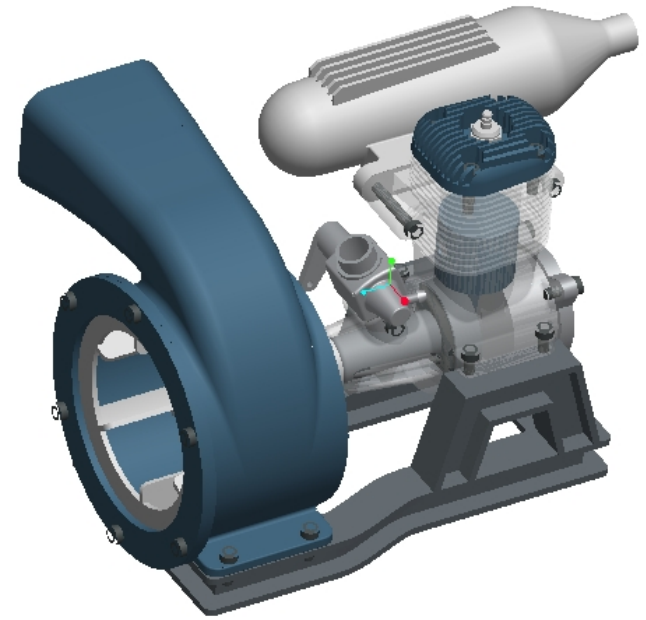
Using the two packages together **should be** relatively easy.

I recommend the following approach:

- Open Creo first, opening the part that you want to analyse with ANSYS Workbench.
- When you are ready to perform an analysis, open ANSYS Workbench, and start a new simulation.
- It is then easy to import the Creo part into ANSYS (we'll discuss details of this soon). Carry out your FEA work as usual.
- If you want to make changes to the geometry, do so in Creo.
- You can then swap over to ANSYS, and update the geometry. Your design will develop through several of these iterations.

Creo ⇌ Workbench Exercise

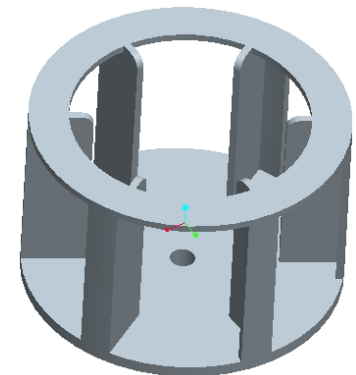
We will consider a single component of the motor/blower assembly shown.



The blower is effectively a pump designed to move air

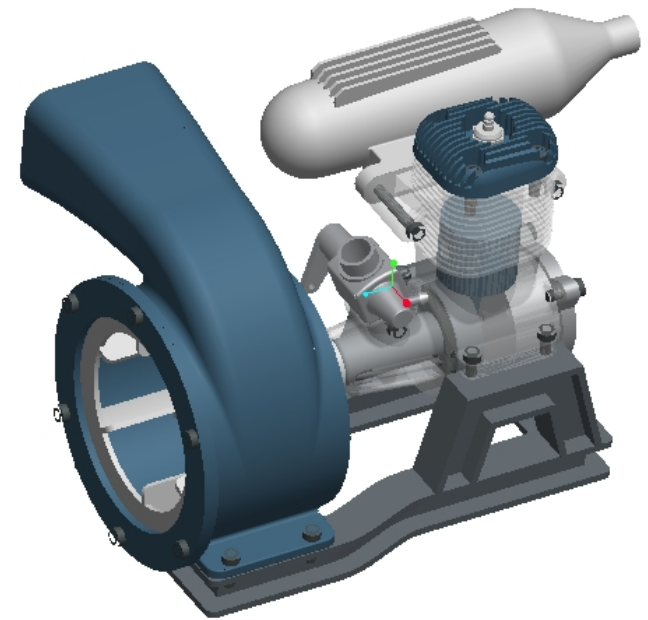
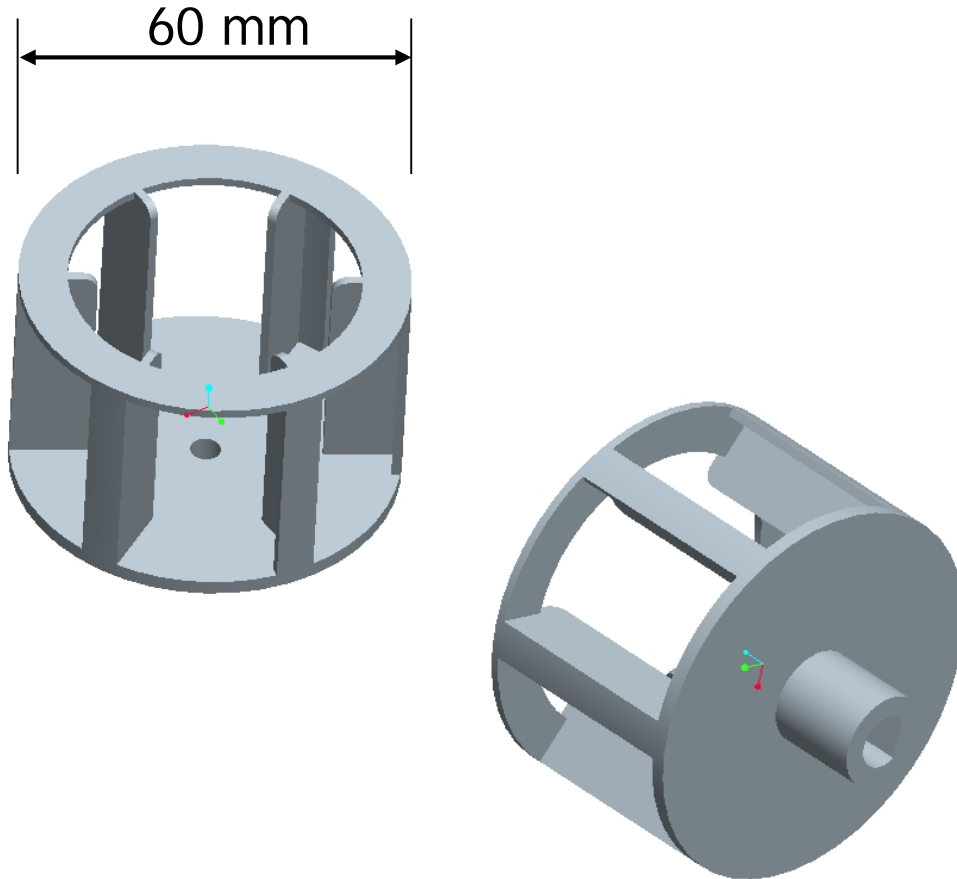
We will focus on the impeller, which is the main moving component of the blower.

Can you imagine how the stiffness and deformation of the impeller will change if modifications are made to its structure?



Creo ⇔ Workbench Exercise

This system is built around a small single cylinder engine. Note the small dimensions of the impeller...



Mounted on a 5 mm diameter shaft.

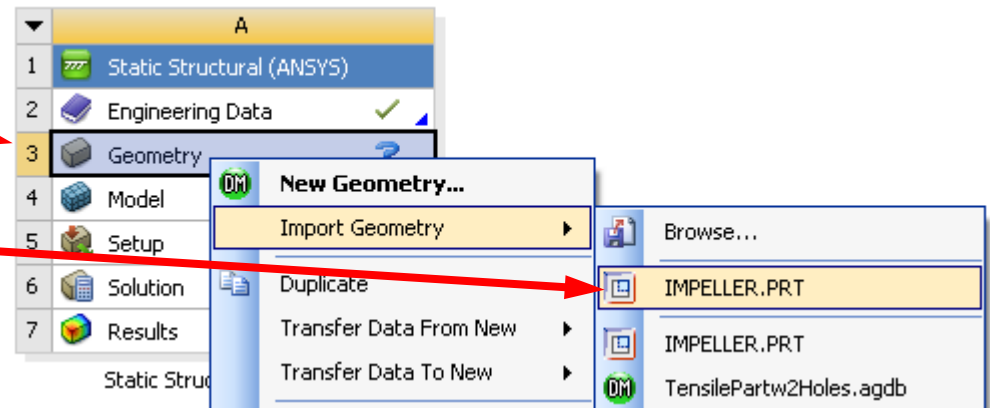
Getting started

- Launch Creo.
- Open the following part file:
Where you have saved it .../Tutorial4/impeller.prt
- Launch “**ANSYS Workbench**”. Choose a Static Structural Simulation.
- In Workbench, import the geometry...

- Right click step 3 – “Geometry”

- The Creo part file should be listed as an option. Select this file.

- The part should then be imported into ANSYS.

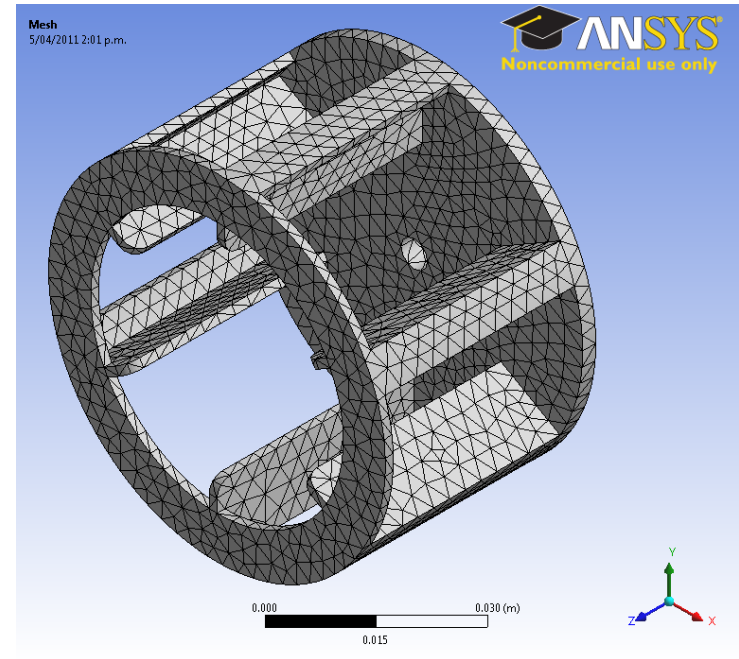
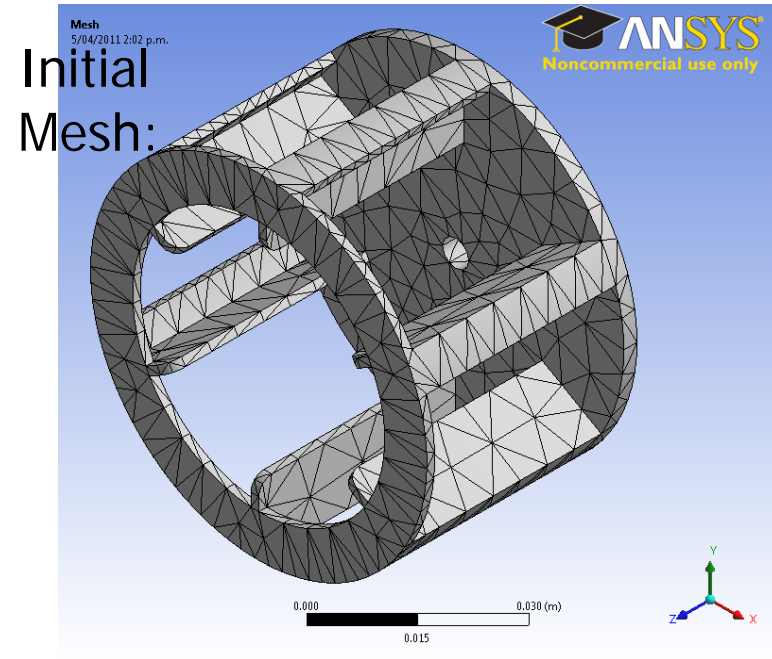


**Save your project locally
or it may not solve!**

In your own time

- Right click on Mesh in the Outline...Update

The initial mesh looks quite coarse, let's start with a global refinement.

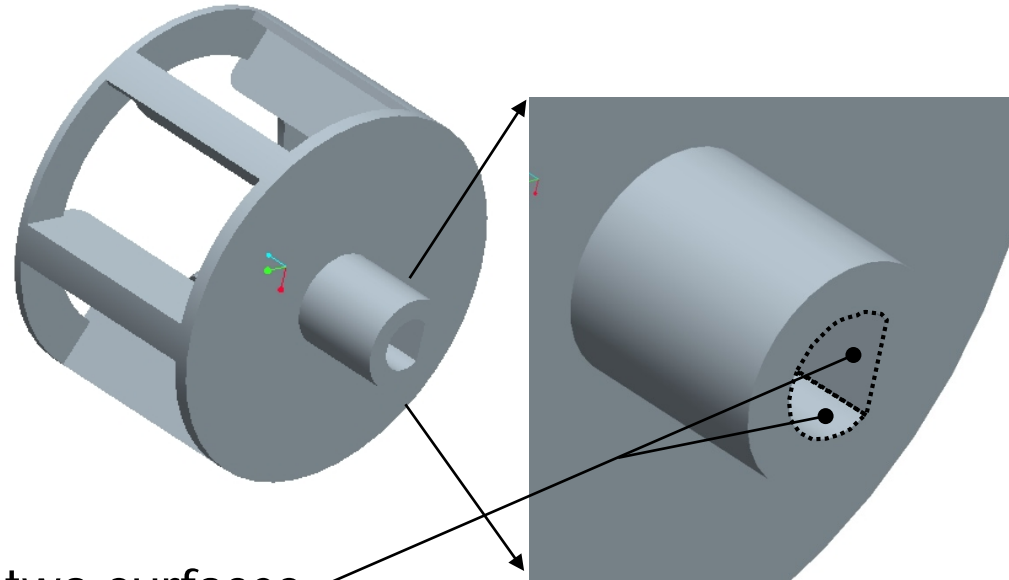
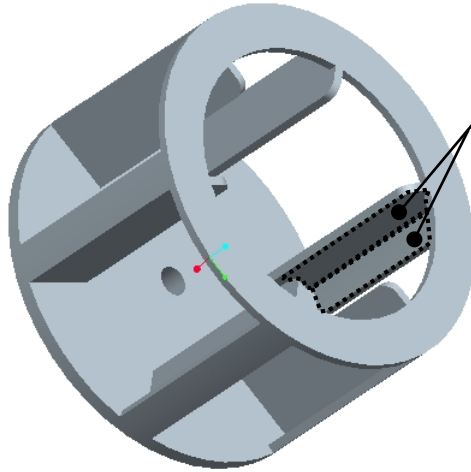


- Select the entire part, right click on the mesh – insert a Sizing. Enter an element size of 0.003m. Generate mesh...
- Specify the material. Import Aluminium Alloy.

Application of loads/constraints

Apply the loads/constraints shown below

Apply a **Pressure** of 50kPa to these two surfaces of a blade. Repeat for all blades.



Apply a **Fixed Support** at these two surfaces.

Forgotten how to do this?

Consult Tutorial 3.
(Available on Canvas)

Establishing results

- Click on Solution in the Outline
- From the Solution bar menu, choose the following results to present:
 - 1) Total Deformation
 - 2) Equivalent Stress

The model is now ready to solve.

Solve!

- Spend some time examining the results
 - How does the deformation look? Maximum displacement?
 - What is the maximum equiv. stress? Where does it occur?
 - What torque is generated at the shaft output?

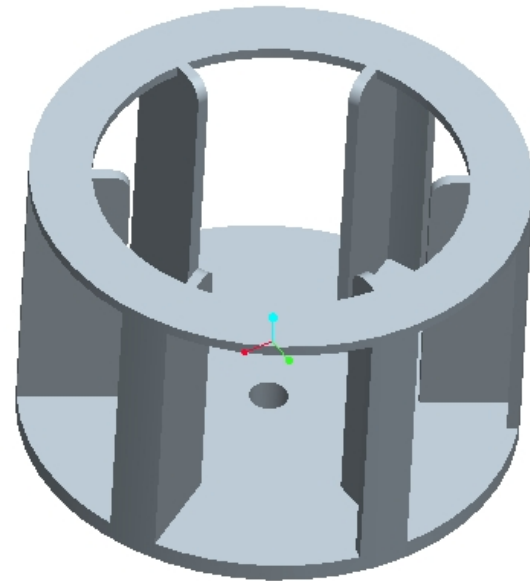
HINT: Introduce a new result, a torque or force probe at the fixed support

Creating a modified model

We'll now make a modification to the part design in Creo.

- Take some time to explore how the Creo impeller model was generated.
- Modify the thickness of one of the blades, reducing it by 50%. To do this, you need to edit the definition of the appropriate protrusion.

If you adjust the first blade, the change will be copied to the others.



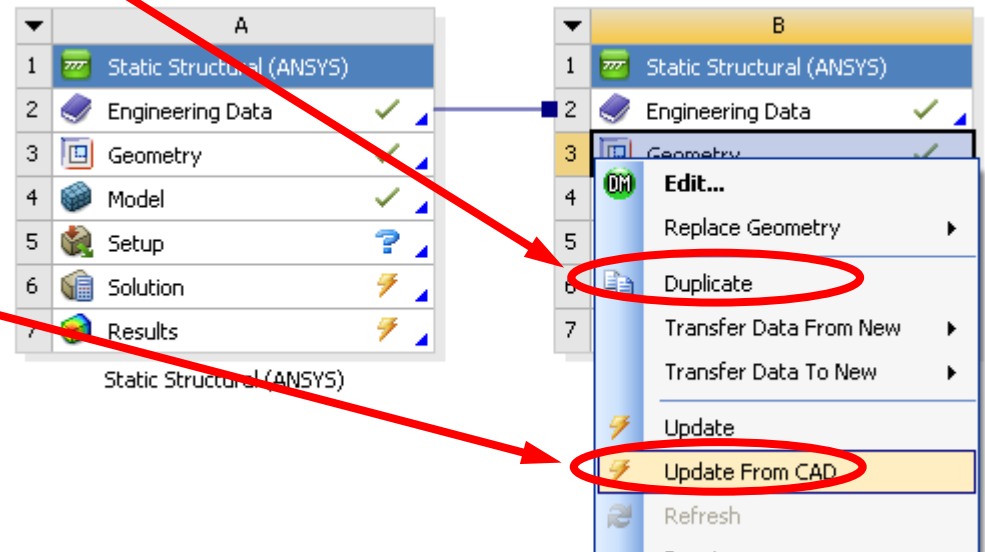
Creating a modified model

Back in Workbench, we now create a duplicate of the previous model at the “Geometry” level.

- First, rename the original model “Original Model”.
- Now, right click on “Original Model” at the “Geometry” step and select duplicate. Rename the new model “Thin Blades”.

- Within the “Thin Blades” model, right-click on geometry - choose: “Update From CAD”

The modified geometry should now have been imported to ANSYS.



In your own time

- Preview the mesh again.
- Check whether your loads/constraints are still valid. Has changing the geometry affected any of them?
- You are now ready to solve. Hit the **solve button** again.

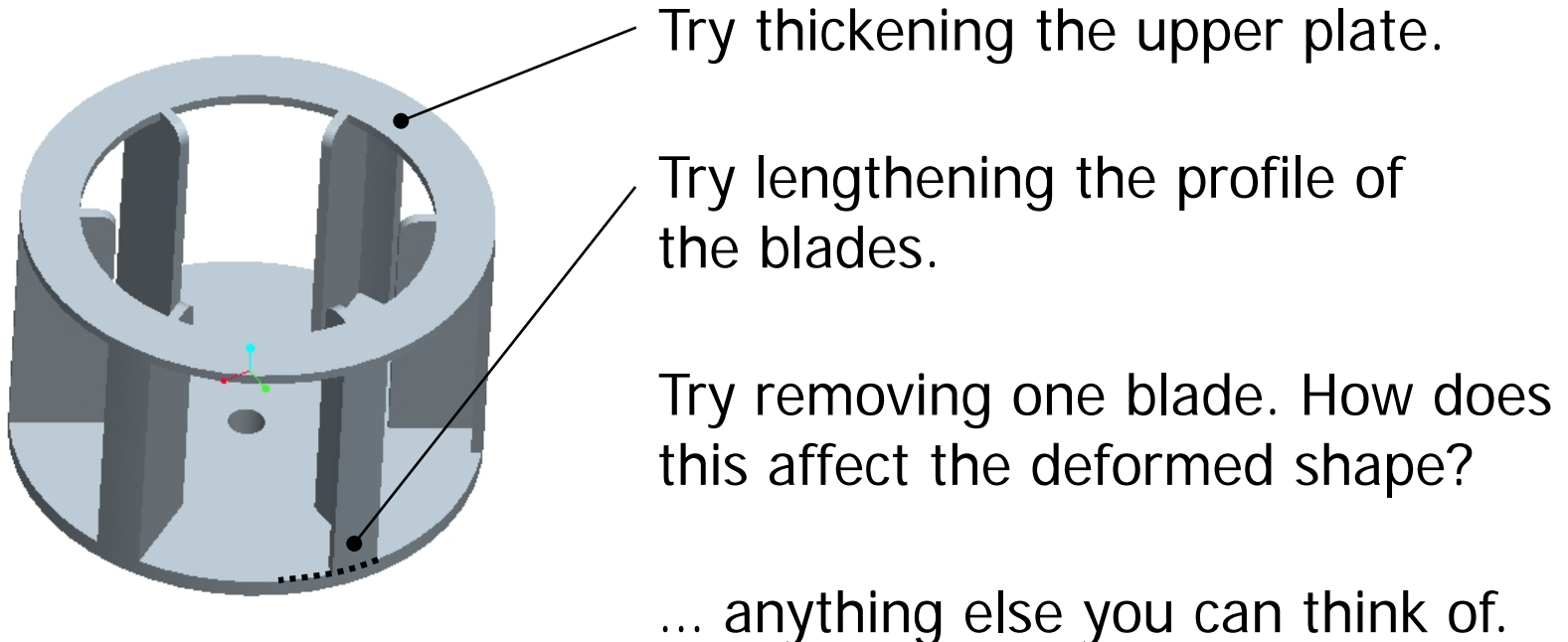
You are now able to compare results from the two models. By clicking on results in either “Original Model” or “Thin Blades”, you can directly compare them.

- How has the maximum displacement changed?
- How has the maximum equiv stress changed? Has the position changed?
- Have the blades deformed more or less?

Further design modifications

Try some further modifications to the geometry. Points to consider...

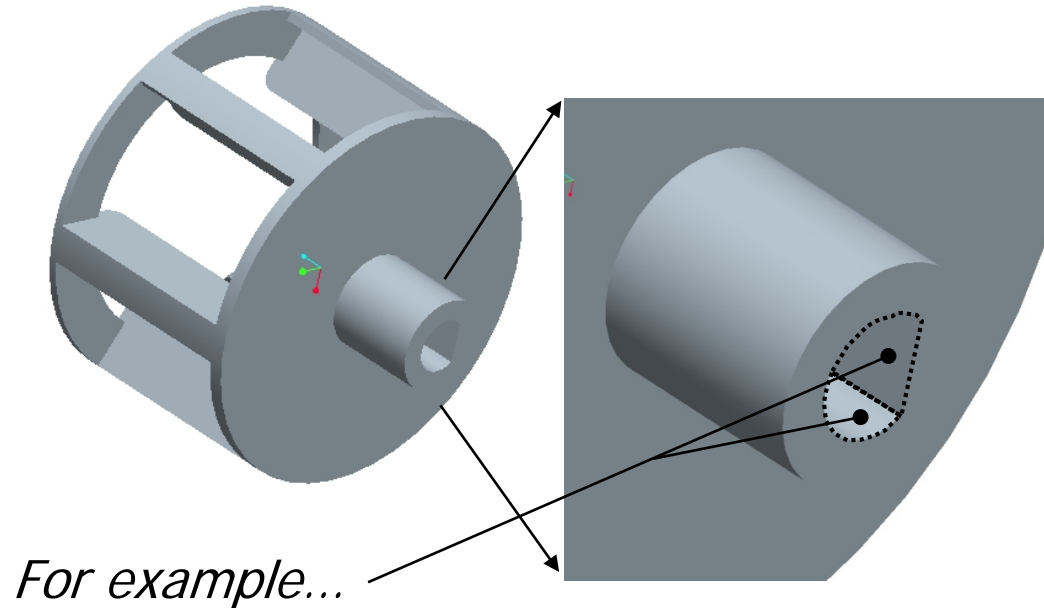
- When creating a significant modification, its worth creating a copy of a previous model.
- Remember to modify your load/constraints if necessary.



Determining reactions at supports

During your structural bracket design in Project B, you will want to determine reaction forces at supports. This is done using **result probes**.

- First, click on the Solution.
- Select the surfaces at which you applied a support.
- Use the probe drop down menu in the solution bar.



- Choose the type you want, and the appropriate result options (total magnitude? Components?).

In your own time

Consider the Details Window.

- You can specify a specific component if you like.

Details of "Force Reaction" 4

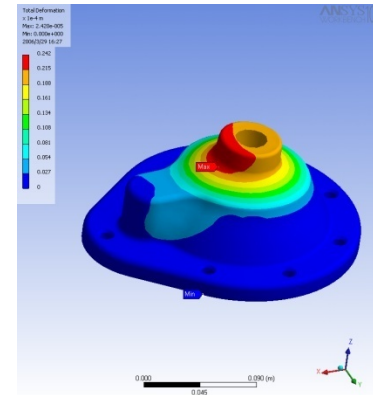
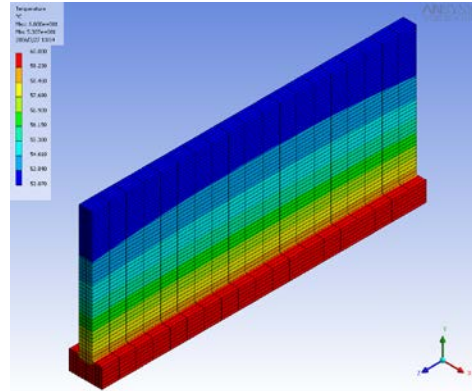
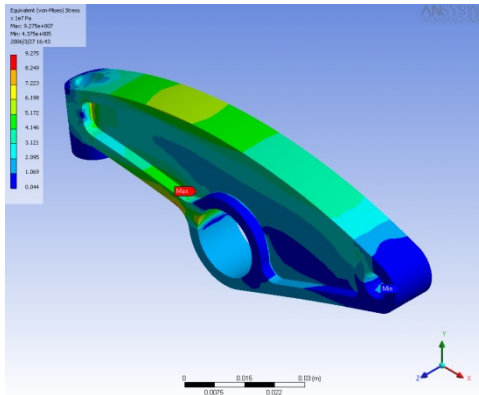
Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Fixed Support
Orientation	Global Coordinate System
Options	
Result Selection	All
Display Time	All
Results	
<input type="checkbox"/> X Axis	X Axis
<input type="checkbox"/> Y Axis	Y Axis
<input type="checkbox"/> Z Axis	Z Axis
<input type="checkbox"/> Total	Total
<input type="checkbox"/> X Axis	-3.5901e-011 N
<input type="checkbox"/> Total	3.6485e-011 N
Maximum Value Over Time	
<input type="checkbox"/> X Axis	4.7293e-012 N
<input type="checkbox"/> Y Axis	-4.4579e-012 N
<input type="checkbox"/> Z Axis	-3.5901e-011 N
<input type="checkbox"/> Total	3.6485e-011 N
Minimum Value Over Time	
<input type="checkbox"/> X Axis	4.7293e-012 N
<input type="checkbox"/> Y Axis	-4.4579e-012 N
<input type="checkbox"/> Z Axis	-3.5901e-011 N
<input type="checkbox"/> Total	3.6485e-011 N
Information	
Time	1. s
Load Step	1
Substep	1
Iteration Number	1

Verification: How accurate is your analysis?

In lectures and tutorials you have now learned:

- The basic concept of how FEM/FEA works
- How to carry out basic Finite Element Analysis
- The importance of applying appropriate loads/constraints
- How to perform global and local mesh refinement

We've considered a variety of models and analysis results



CRITICAL QUESTION: How can we tell if the results are realistic?

Verification: How accurate is your analysis?

Remember that any model is **an approximation to reality**, whether it is a simple hand calculation or a sophisticated numerical analysis. It is your responsibility to:

- Understand the physics and assumptions you are applying.
- Ensure that the results you produce are an appropriate approximation to reality (considering what you need to know and how much time is available).

I will introduce some recommended approaches that will give you confidence in the FEA results you obtain. Such an approach is a requirement of a serious FEA study.

You must consider this issue in Project B and demonstrate that you have done so, in your report.

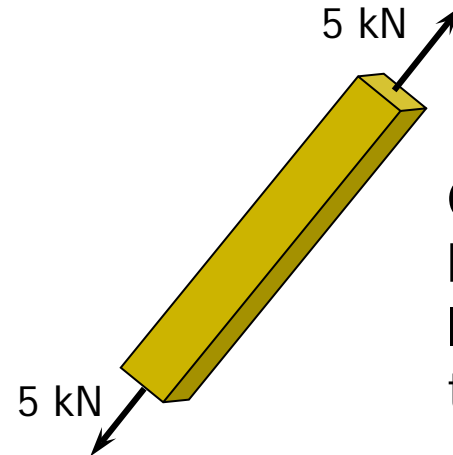
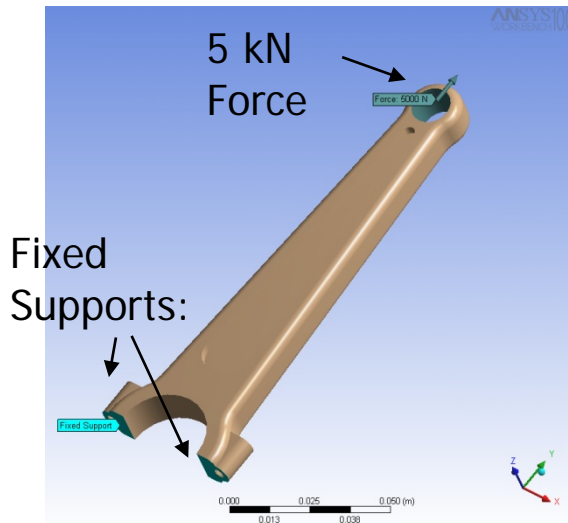
Recommended Approach 1:

Comparison to Hand Calculations

FEA allows analysis of complex problems. We can work on problems difficult to analyse by hand, exploring them in detail.

However, where possible, hand calculations should be completed to verify the magnitude of your FEA results.

Example:



Constant X-section bar, simple axial loading (we revised this in lectures!)

- Have you made any basic mistakes? (materials? B.C.s?)
- Are your FEA results realistic?

Recommended Approach 1:

Comparison to Hand Calculations

The following describes a recommended procedure for analysis with FEA. Following this philosophy will help you produce realistic results, more than just pretty pictures.

Step 1: Establish Problem on Paper (Workbook)

- Lay the problem out on paper, establishing:
 - Rough geometry
 - Material properties
 - Appropriate loads/constraints
 - Results required (Stiffness? Strength? Heat flow?)
- Define a simplified problem, that you can analyse by hand. The hand calculations should provide the same sort of results required from the FEA analysis.

Step 2: Initial Analysis with FEA and Verification

An initial FEA analysis is performed, the results being compared to your hand calculations.

- Utilise the mesh recommended by Workbench, and solve.
- If results are close to the hand calculations, move to Step 3.

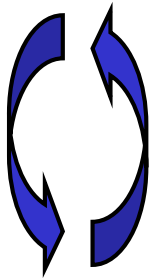
How close is close enough? Depends on how well your models represent the problem.

- If the FEA results do not compare well to the hand calcs:
 - Check for **silly mistakes**. Wrong material properties? Incorrectly entered loads/constraints? Are you interpreting the results correctly? Other issues...
 - Still a problem? Consider refining the mesh.
 - Still a problem? Reconsider your hand calcs...

Step 3: Mesh Refinement, Results Convergence

Once comparison to hand calculations has given you confidence in your FEA analysis, look to refine the accuracy.

- Identify the key results you require, and write down in your workbook the values obtained from the initial FEA analysis.
- Carry out mesh refinement:
 - Globally, if you think more elements are needed in general.
 - Locally, in areas where results vary quickly in space.
- Solve, check key results, and write down the new values.

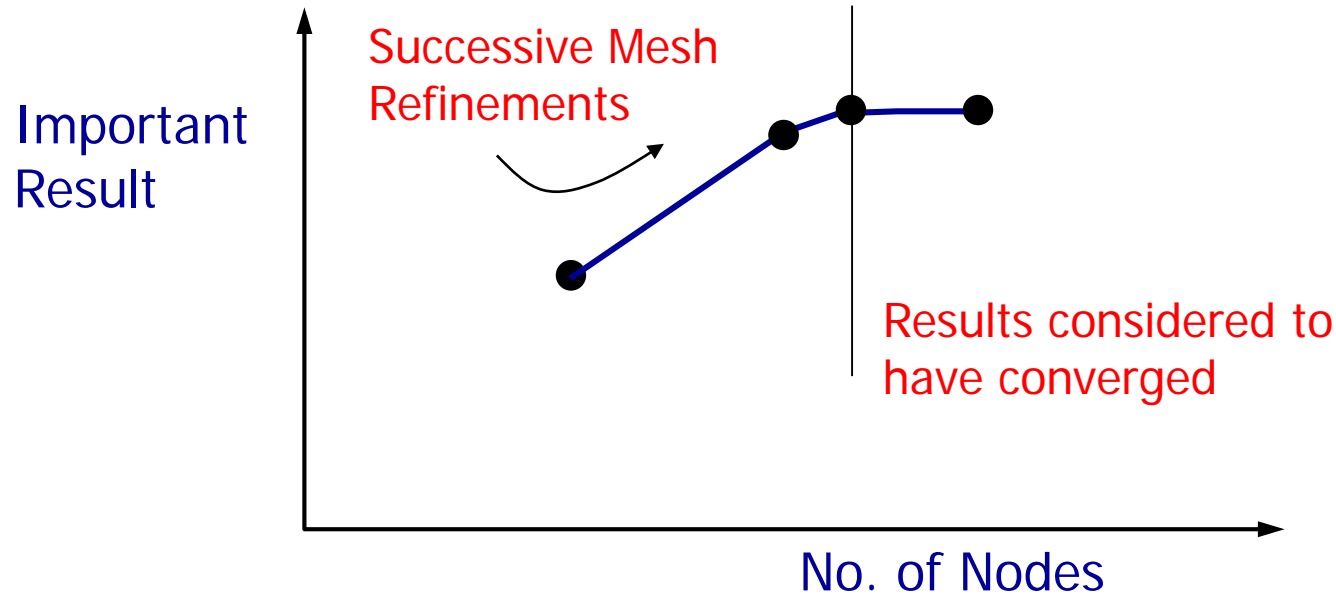


Iterate this *refine-solve-check* loop until the results do not change significantly and you can conclude that the results have converged.

You're Done!

RECALL: Mesh Refinement

The aim of mesh refinement is to have the **results converge**, while utilising resources efficiently (reasonable no. nodes/elem).



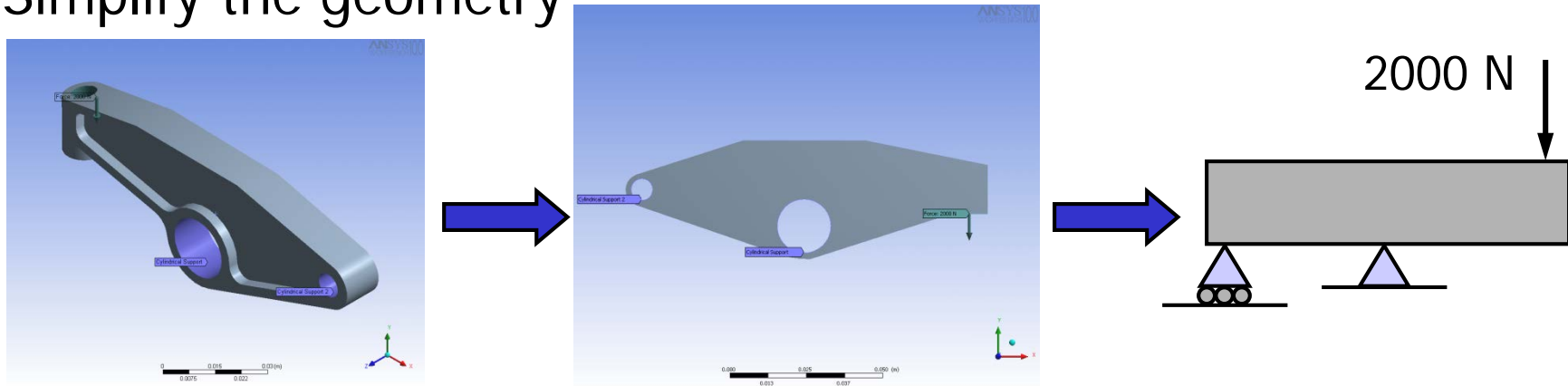
Mesh refinement can be done **globally** (over the whole model), or **locally** (where results are changing rapidly). You will **iteratively improve your mesh**, until you are convinced that the **important results have converged**.

Simplifying models

Simplifying a complex problem enough to do a hand calculation can be difficult. It is something that you will improve at with more experience.

However, there are several approaches you can take:

- Simplify the geometry

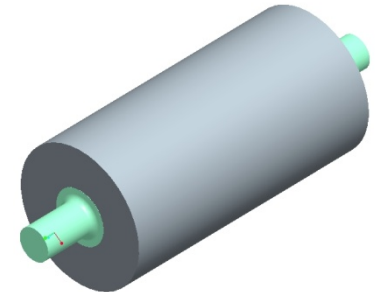
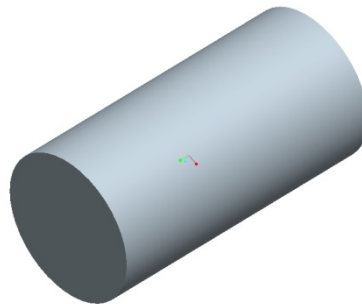
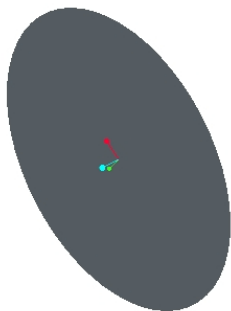
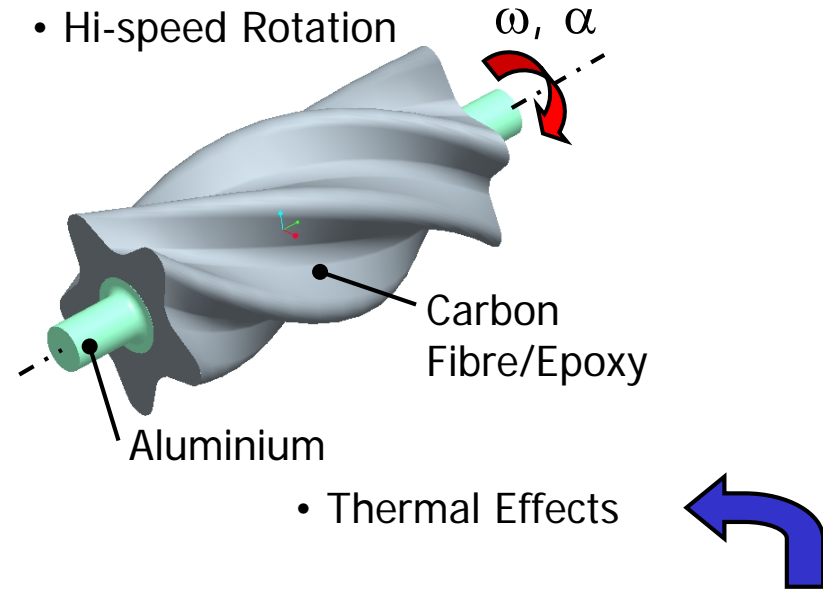


- Simplify the material properties
- Simplify load/constraints

Recommended Approach 2: Progressive Development towards a Complex Model

Sometimes, the component under analysis is very complex and it is too difficult to complete a useful hand calculation.

Another way to gain confidence in your results is to consider a series of models with FEA...



simplified...

... through to the complete model.

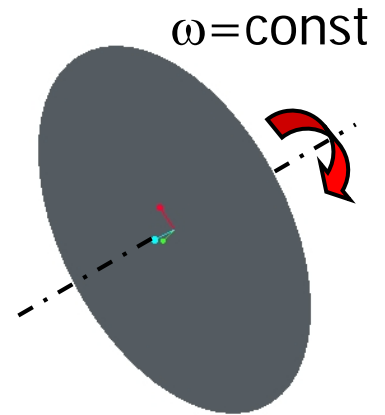
Recommended Approach 2 (A Case Study)

As part of a previous 4th year project, the complex 3D rotor of a supercharger was analysed. The aim was to use composites.

a) Rotating, thin disk

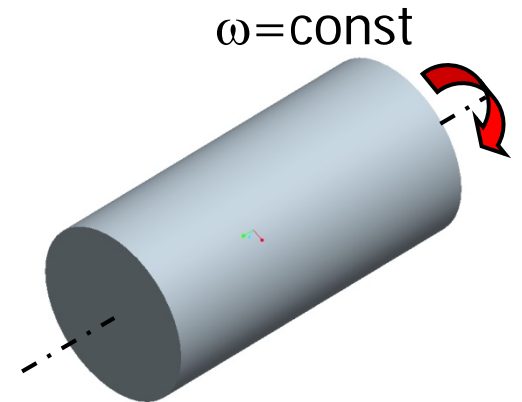
This allowed comparison to analytical solutions (refer to your textbook!).

This gave confidence that ANSYS was dealing well with rotating bodies.



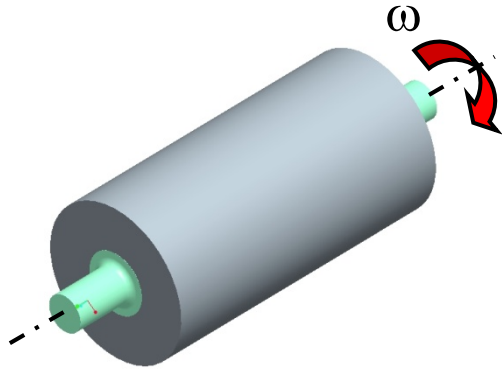
b) Rotating, 100 mm cylinder

A simple extension to case (a) moving towards a real-world geometry.



Recommended Approach 2 (A Case Study)

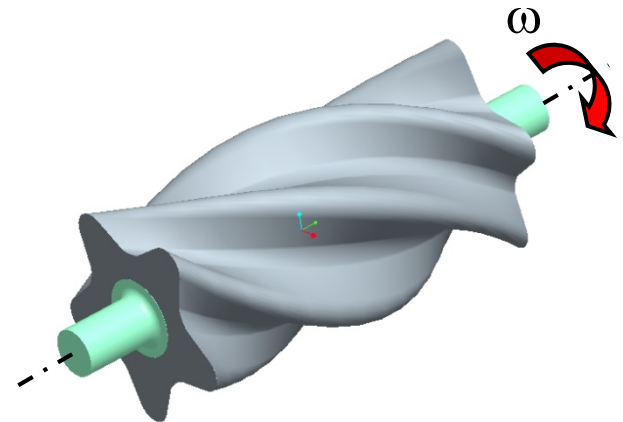
c) Rotating, composite cylinder and aluminium core



An aluminium axle was inserted, based in a carbon fibre/epoxy shell. This could possibly be analysed by hand, but it would be more difficult. An FEA model could be built of this intermediate case and verified against (a,b).

d) Complete Model

Models (a-c) provide comparison points and enough confidence to analyse the final geometry.

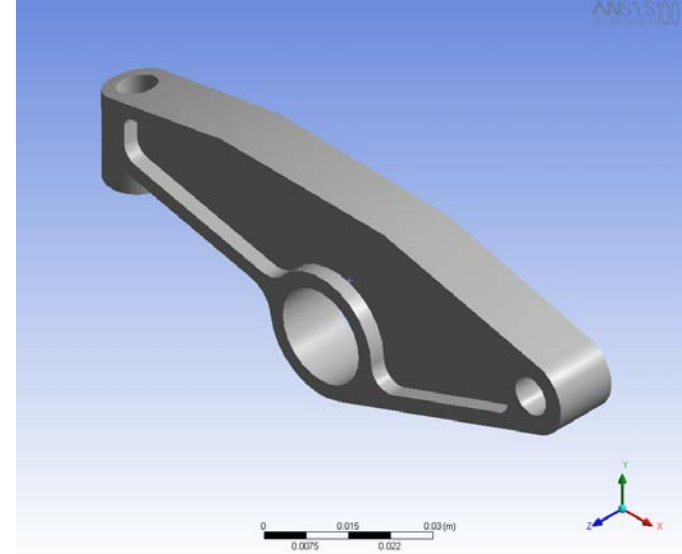


Often, a solid model will evolve and be progressively detailed at the same time as the analysis!

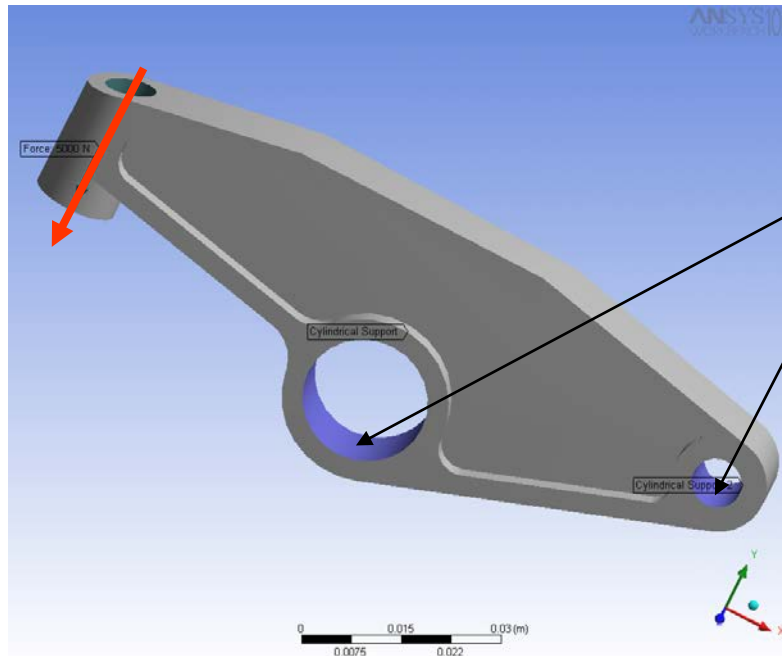
Results verification: Exercise

Consider again a structural analysis of the lever shown. In this tutorial exercise you will:

- Verify FEA against a hand calculation
- Refine the mesh, checking for convergence



A 5kN force is applied through the hole as shown.



The component is mounted on shafts at these two locations, and is considered free to rotate about these shafts.

Step 1: Establish Problem on Paper (Workbook)

- Lay the complete problem out on paper

- Sketch the geometry
- Write down the material properties

Structural Steel: $E=200 \text{ GPa}$, $\nu=0.3$, $\rho=7850 \text{ kg/m}^3$

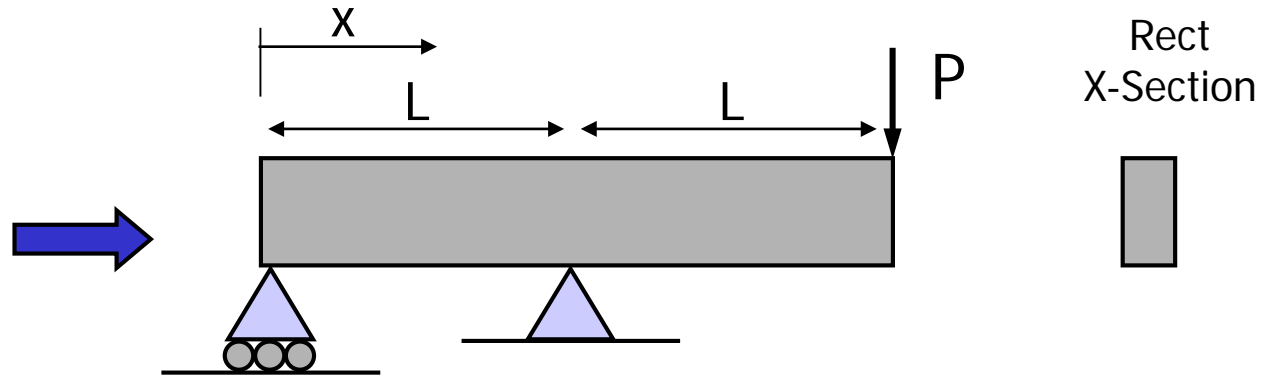
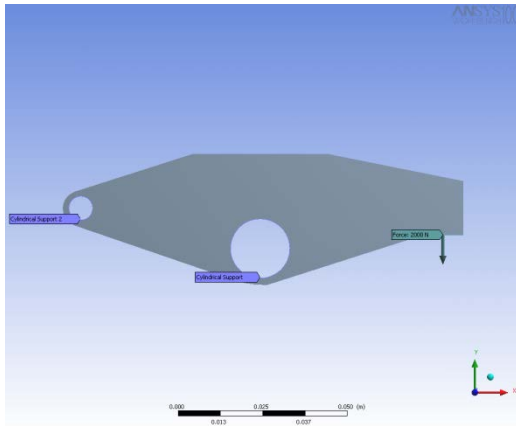
$$\sigma_{Yd}=250 \text{ MPa}$$

- What are the appropriate loads/constraints
- What result(s) will you compare to ANSYS? In this case:
Maximum displacement? Where does it occur?

IMPORTANT: To benefit from this exercise, you must work with a workbook/notepad and interact with your computer.

Step 1: Establish Problem on Paper (Workbook)

- Define a simplified model/problem that is solvable by hand.



HINT: Do you remember how to do this? Refer to your MoM lecture notes or textbook, eg. Section 12.3 and Example 12.5 in Hibbeler 9th Ed.

$$(EI_z) \frac{d^2 v}{dx^2} = M_{xz} = 2P \langle x - L \rangle - Px$$

$$(EI_z) v = \frac{P}{3} \langle x - L \rangle^3 - \frac{Px^3}{6} + \frac{PL^2 x}{6}$$

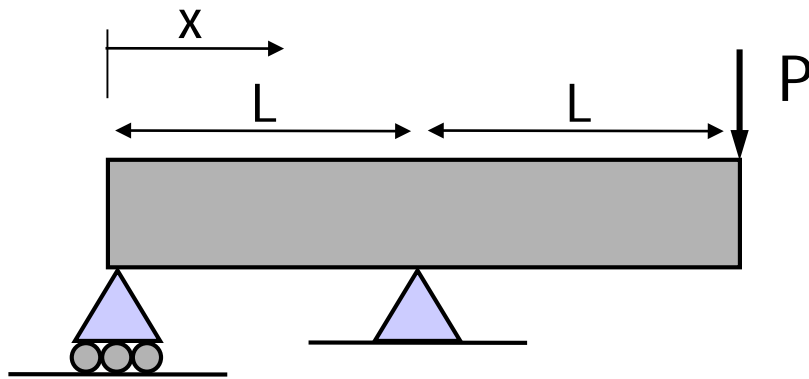
$$\sigma_x = -\frac{M_{xz}}{I_z} \cdot y'$$

Remember, for a rect. X-Section: $I_z = \frac{bd^3}{12}$

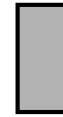
Step 1: Establish Problem on Paper (Workbook)

- Calculate results for comparison to ANSYS.

WRITE THEM DOWN!

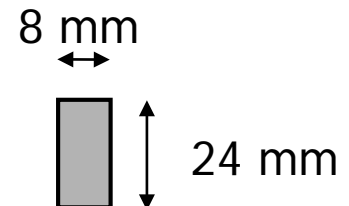
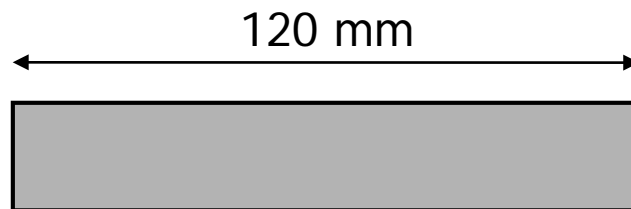


Rect
X-Section



- What estimate for L ?
- X-sectional measurements?

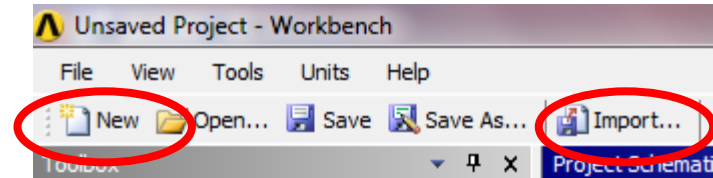
- I'd prefer you estimate the simplified geometry yourselves, but these are the values I used:



Step 2: Initial Analysis with FEA, and verification

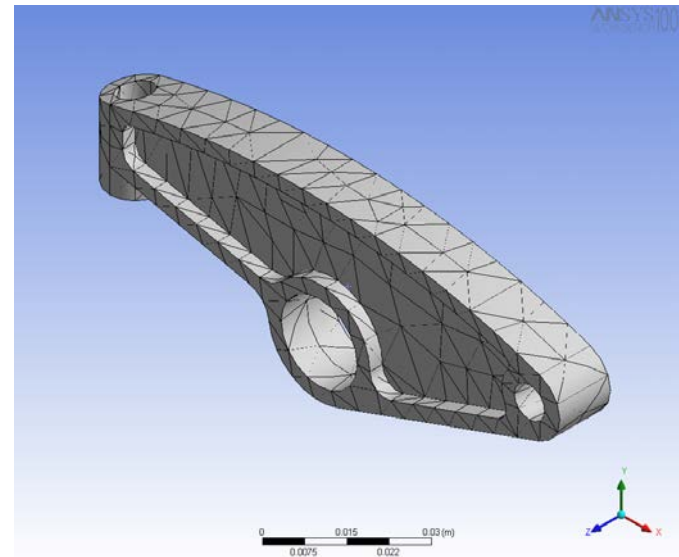
- Save your previous project.
- Start a new project (Click “New” in the workbench window) Click “Import...” to import a “mechanical file”. **NOTE that it is a *.dsdb file:**

Where you have saved it
.../Tutorial4/3DLever.dsdb



***NOTE:** This is different to our usual approach of loading geometry files. There were some problems in previous years, so we are loading a mechanical file database.*

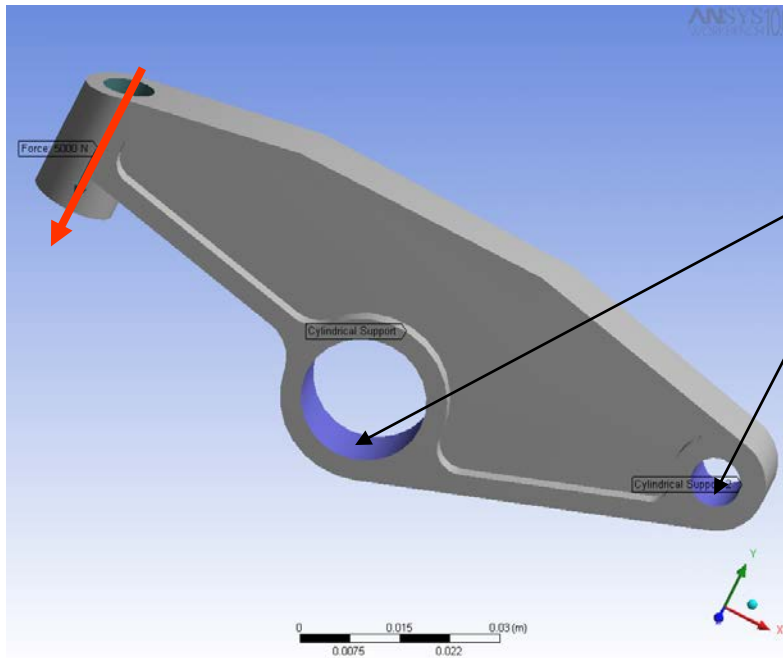
- Review the mesh...



Step 2: Initial Analysis with FEA and verification

- Specify the material. Ensure Structural Steel is selected.
- Apply the loads/constraints shown below...

5kN force,
parallel to
the hole.



Apply cylindrical
supports to these
surfaces.

IMPORTANT: You have
three types of motion to
specify. Apply:

Radial: Fixed

Axial: Fixed

Tangential: Free

*Change
this one*

This will allow only rotation
about the support shafts.

Step 2: Initial Analysis with FEA and verification

- Establishing results. Setup the following...
 - 1) Total Deformation
 - 2) Equivalent Stress

The model is now ready to solve. Hit the **Solve button...**

- Examine the results, comparing to your hand calculations.
 - Are the deflections close enough? Are they the right order of magnitude, or have you made a major mistake?
Ask a tutor if you are unsure.
 - If you're confident, move on. FIRST, write down max. displacement, max. equiv. stress, and where they occur.

Step 3: Mesh Refinement, Results Convergence

At this point you should have confidence in your initial FEA results. However, are they as accurate as they could be? Mesh refinement is required, until the important results converge.

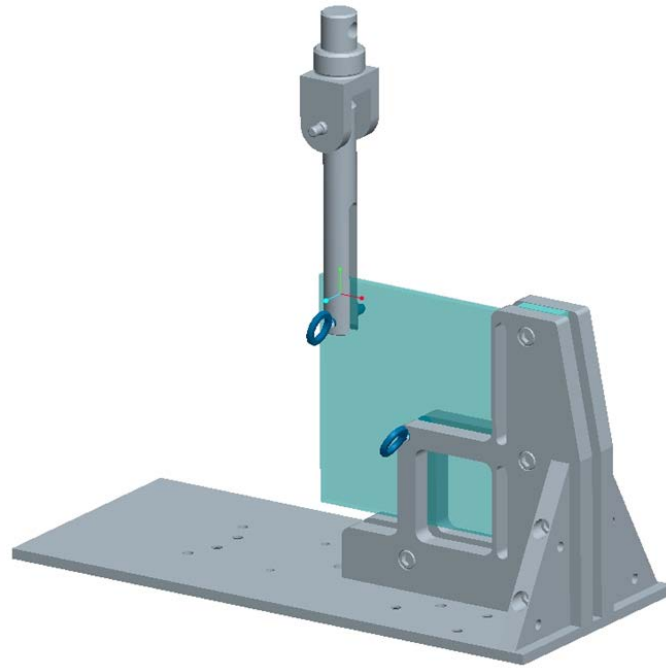
Now apply successive mesh refinements, considering:

- Both **global** and **local** mesh refinement techniques. I suggest a global approach initially, performing local refinement to ensure convergence of max/min stresses.
- Make copies of your major mesh refinements (and associated results), keeping a record of your process.
- Make sure you write down key results after each iteration, and consider how they are changing.

Have key results converged? **You're done!**

All done! You should now have everything you need for the 2017 Brake Lever project.

Important Note: You need to save your Creo file locally BEFORE you can import geometry to ANSYS.



HINT: Do you want to create a 2D part in Creo?

Use a "Surface -> Fill" tool, on the front plane.