

Wrench Meshing exercise

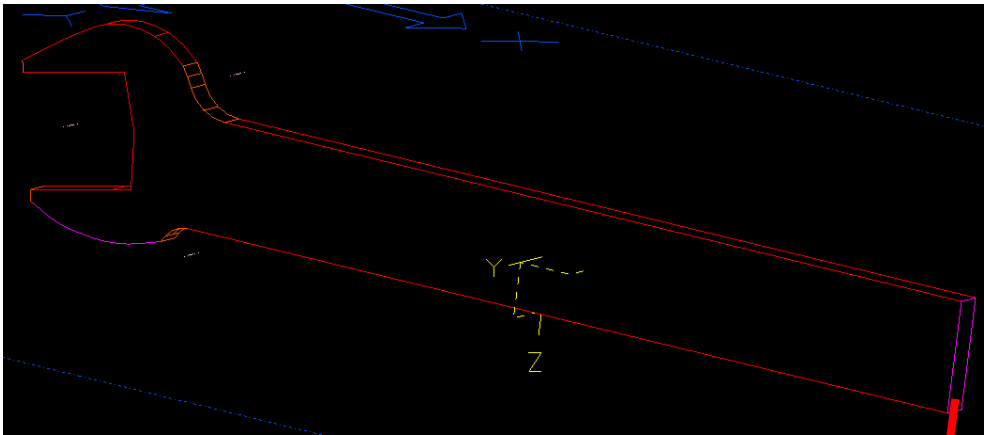
A Steel Wrench is subjected to a 100N load as shown below with the faces shown being constrained so that they can not move on the nut. Assess the wrench geometry exactly as it is with a generic 1mm mesh to determine the maximum stress seen. Answer the following;

- a. How many elements were generated? (information>advanced simulation>simulation summary)
- b. How long did the run take? (look at the end of the log file for the 'real' time taken)
- c. What is the maximum stress seen?

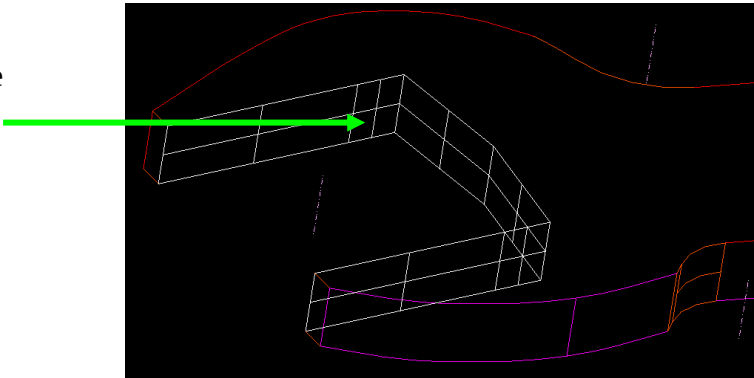
Why is a 1mm mesh a suitable size for the majority of the wrench?

File>Close>all parts. Reopen the original part. Now IDEALISE the part to remove the Honda Logo and resolve the model, answer a, b and c again and write your answers in the below table.

Using these results, refine the mesh in the areas of high stress. Re-solve this model and again fill in the below table



The 6 faces of the jaw are constrained in 6 DOFs

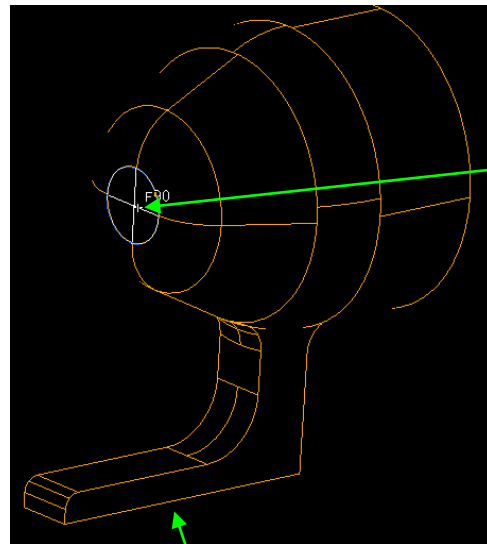
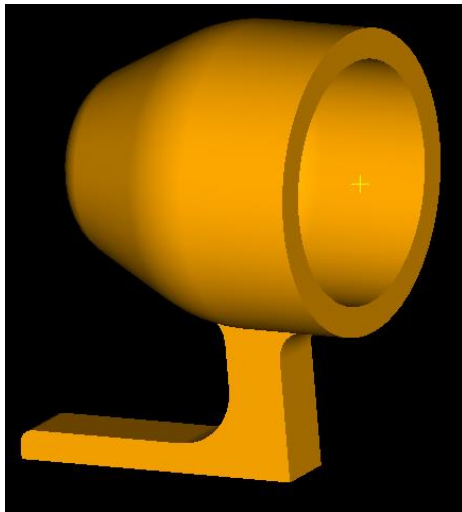
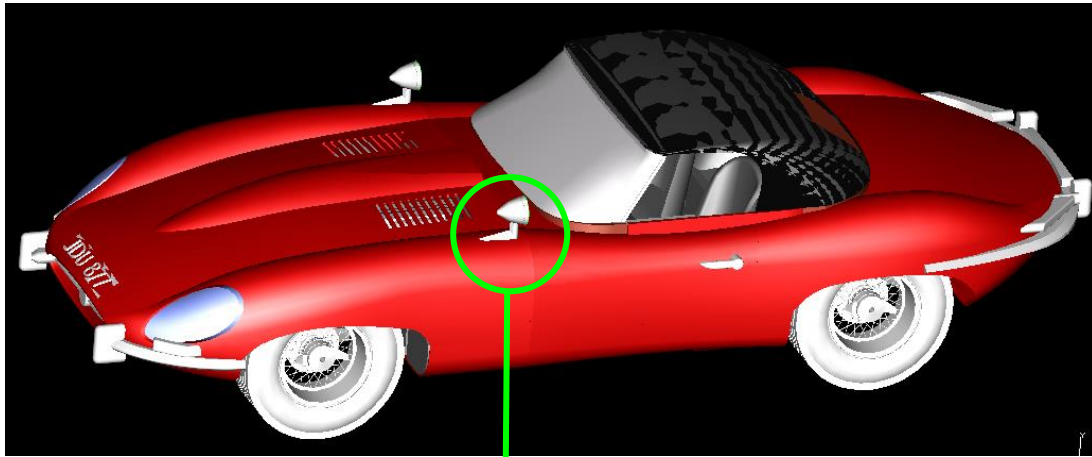


Comment on how realistic this FE model is.

Comment on the three models that you have generated.

	Original Geometry	IDEALISED Geometry	Refined Mesh
Max Stress			
Number of elements			
Time taken for analysis			

Loading of an automotive wing mirror



Force is applied to this surface

Part is glued on this surface

The above shows a proposed automotive wing mirror housing which will be made from Aluminium. The mirror will be glued to the wing at the shown surface. In order to represent the loading that the mirror will see due to the wind resistance a specification has been written to load the face shown with 100N and assess the stress seen in the bracket.

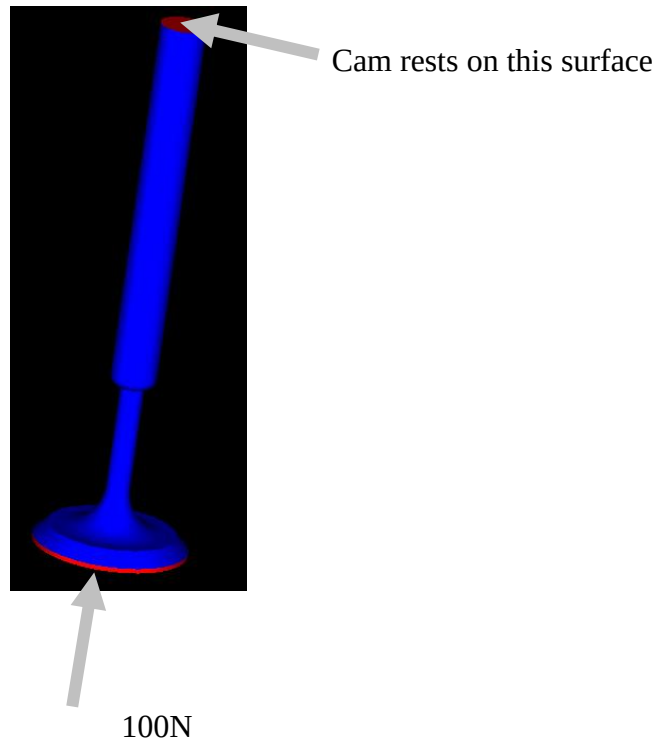
- Assess the design as per the specification and determine the maximum stress.
- Comment on the validity of the specification.

Intake Valve

During the combustion cycle the below steel valve is subjected to a force of 100N, whilst being held shut by the cam resting on the opposite end as shown.

Determine the stress seen by the valve. Try an initial mesh size of 3mm. Comment on this mesh size. Is it suitable? Assess the stress results.

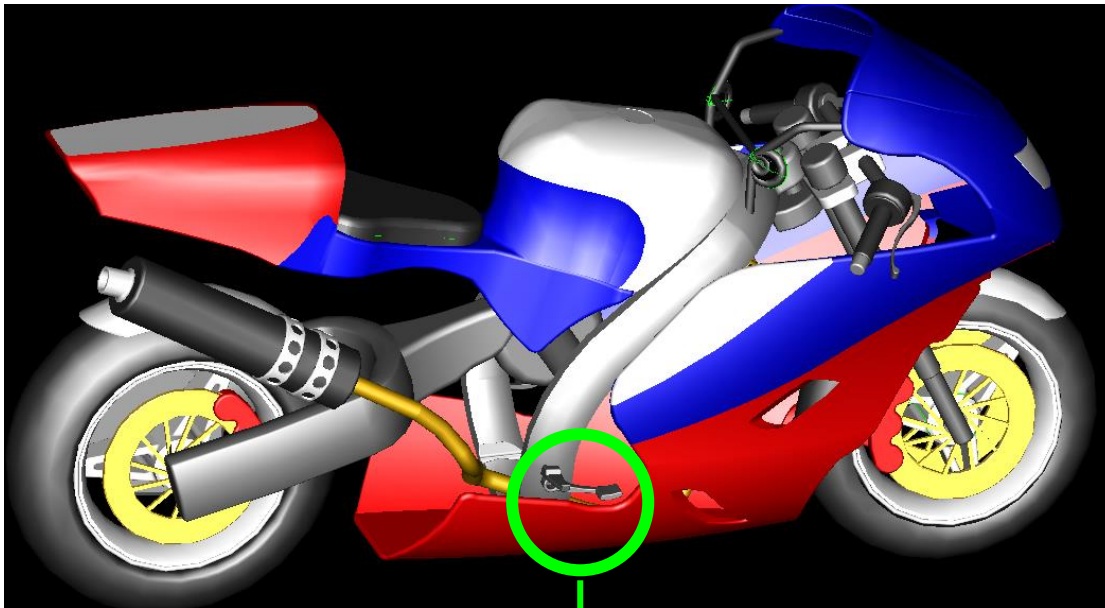
Look at the “unaveraged” stress plot, what does this tell us about the mesh? How many nodes and elements were in this model? How long did it take to solve?



Refine the mesh as appropriate to gain a better answer, compare the number of nodes and the run time.

Comment on how realistic this model is.

Bike Rear Brake Lever

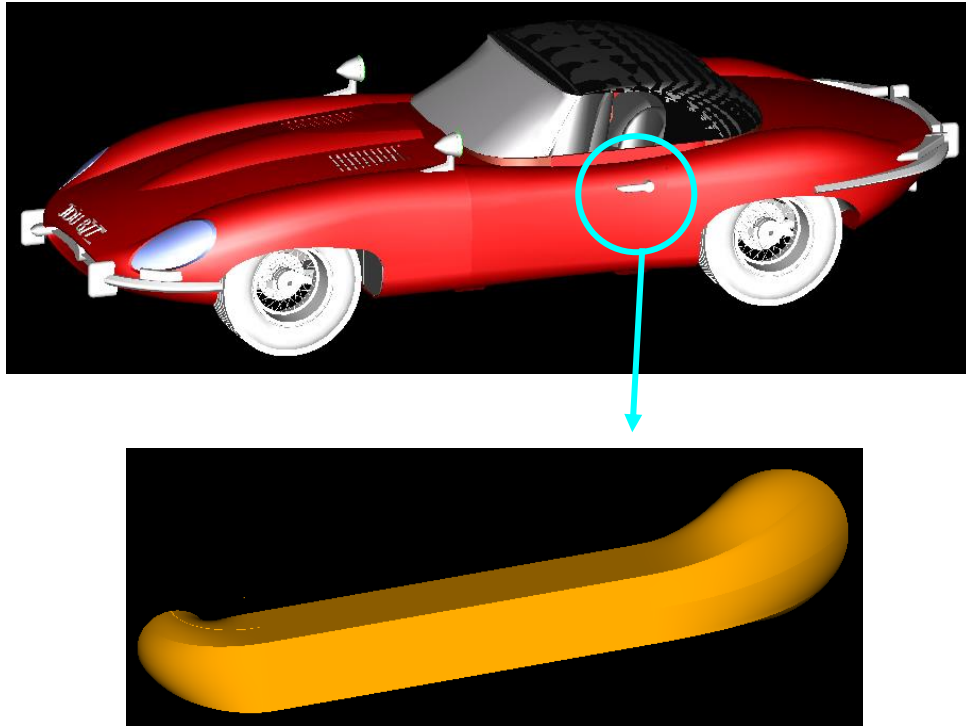


A steel bike rear brake lever has been designed as shown above and is to be made from “generic isotropic steel”. Under normal working conditions this is expected to withstand a force of 100N being applied by the rider whilst being held rigidly on the gearbox shaft.

Assess this component and determine the maximum stress seen under these conditions.

Car door handle – Static pull test

The below picture shows a proposed car door handle



This handle will be made from plastic and when attached to the door must withstand a load of 1kN applied to the handle when pulling the door open without the material yielding.

Assess this handle and comment on the results.