1 Benefits of creating a robust CAD file

The benefits of creating a well thought out CAD model are not always immediately obvious. Problems with modelling are not usually apparent until a design change or update is required. Common issues from design updated to poorly constructed CAD geometry include.

- Loss of symmetry
- Miss-placement of features
- Parts not mating during assembly
- Features that cannot be machined.

For any part there are hundreds, if not thousands of ways a CAD model could be constructed. For example, a simple turned shaft with a cross hole can be created in many ways. There is technically no right or wrong way to create a CAD model, but a well thought out model is easy to modify and keeps the design intent of a part intact, no matter what features change during updates.

1.1 Headed shaft with a cross hole

Let's first consider some design inputs and the design intent for a simple shaft as shown in **Error! Reference source not found. Error! Reference source not found.**.

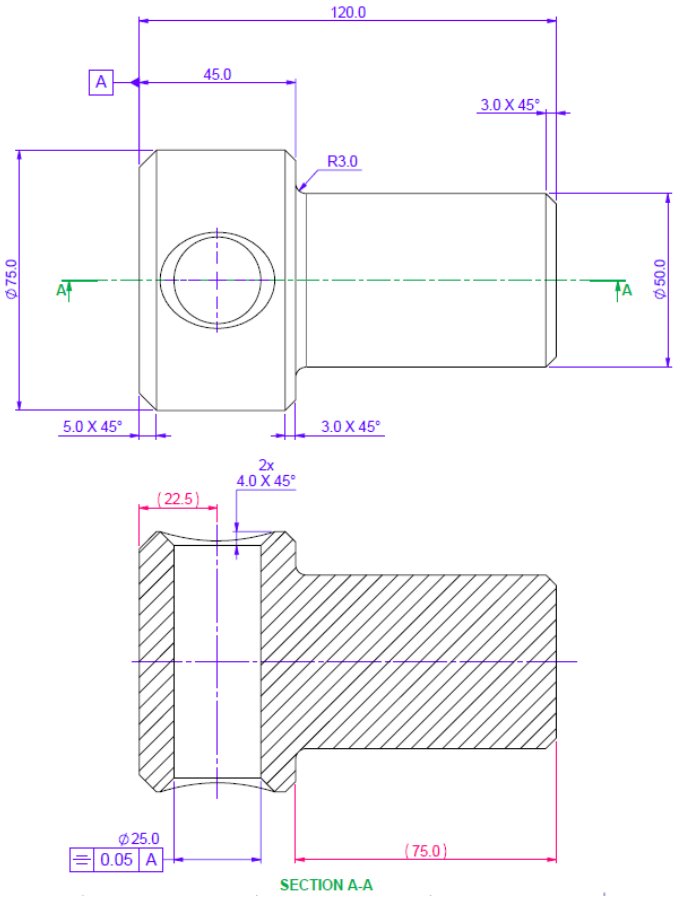

Figure 1 Simple turned shaft with cross hole

1.1.1 The Design Intent

It can bee seen from the drawing in Figure 1 that the simple shaft has some very basic design intent. The drawing has a lot of information about the size of the headed shaft, this is "normal" manufacturing data. All the BLUE dimensions drive the geometry and the RED dimensions are for information (they over-define the main dimensions). But it is hopefully clear that the main design intent is that…

1. The shaft is intended to be turned on a lathe
2. The shaft has a cross hole that is symmetric about the head.

- *Exercise 1:*
  Model the headed shaft shown in Figure 1.
  What is the least number of features required to model the geometry as drawn?

1.2 Creating the model

There are many ways to create a model of the headed bolt, my personal preference would be to create a revolved boss feature for the main body and a revolved cut feature for the hole.

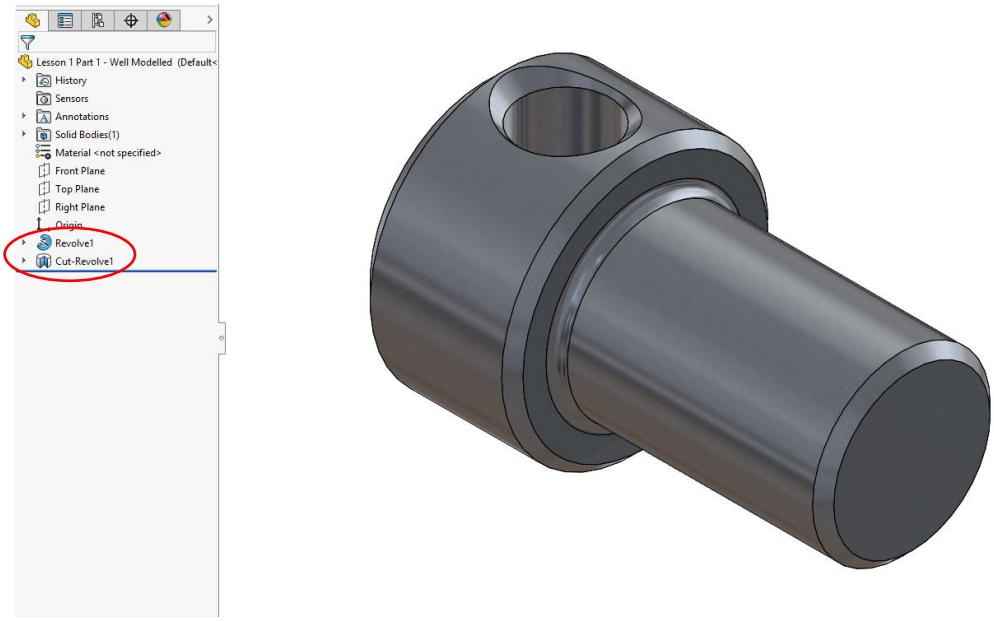

Figure 2Headed shaft: Modelled in 2 features

There are many ways the geometry could have been created! You may have chosen to extrude 2 cylinders and then cut through a cross hole, add the chamfers and then the fillet. Or you may have modelled a cylinder and used a cut revolve to remove the material on the smaller diameter shaft… there are countless ways in which you could have created your model.

However, there is only 1 way to create the chamfer on the through hole correctly without creating additional reference geometry such as plains. If you created the chamfer using the chamfer tool, then unfortunately your part is going to be challenging to machine… If you chamfer looks like the one shown in Figure 3 it is has not been created correctly.

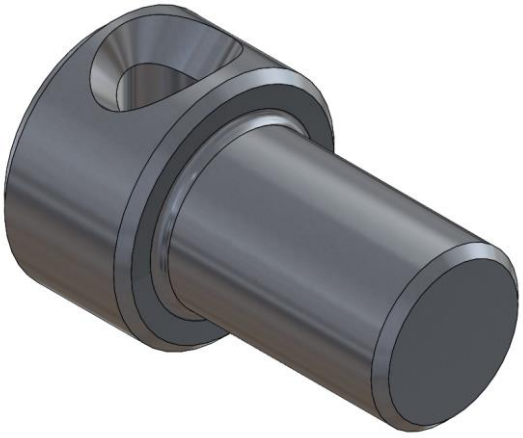

Figure 3 Incorrectly created chamfer feature

1.2.1 Let's look at how I created the geometry.

I started with a sketch on the front plane, I included all the features that were required to turn the component in a single fully defined sketch (Figure 4).

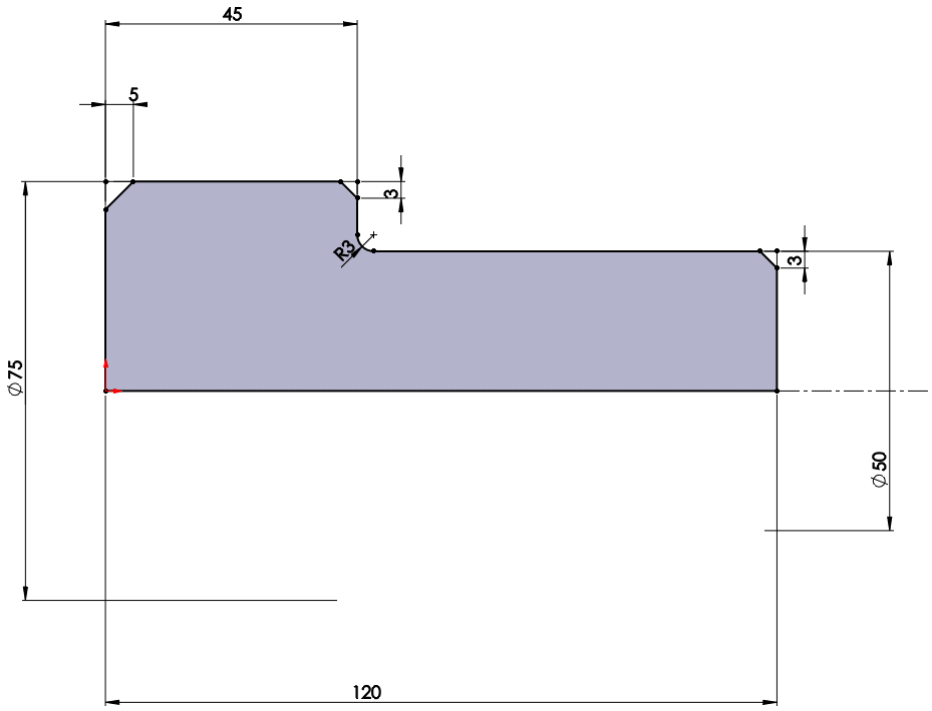

Figure 4 Sketch for Revolved Boss

45° chamfers were created using construction lines with equal length relations and a distance as shown in Figure 5. This eliminates the need to as the chamfers as additional features.

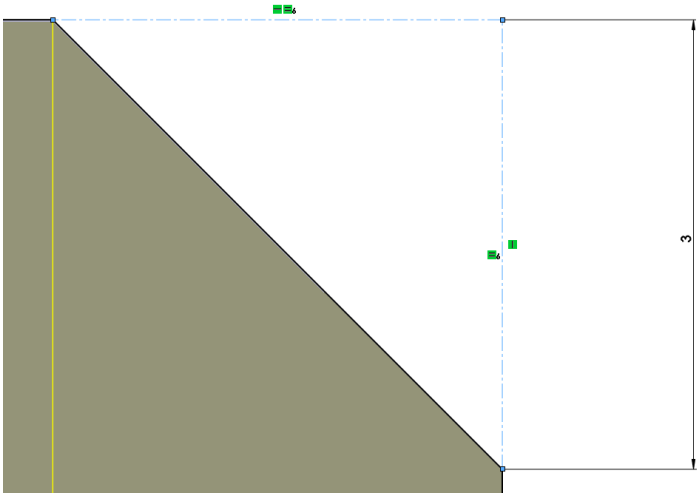

Figure 5 Constructing a Chamfer feature

With the main body modelled I then created a cut revolve to put in the hole feature and its chamfers (Figure 6). I was mindful of the original design intent, that the cross hole needs to be in the idle of the head. So, I draw a construction line across the width of the head and then drew another construction line that was coincident with the midpoint of the first one. Now, the hole will be in the centre of the head, no matter how long the head feature is.

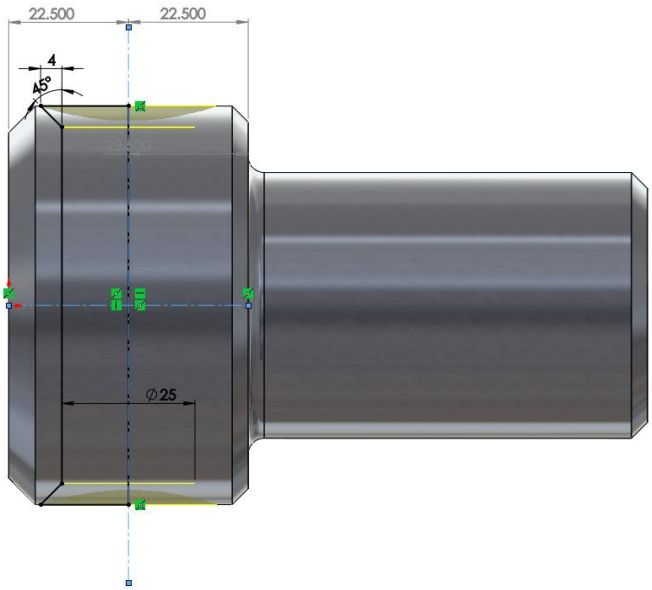

Figure 6 Cut Revolve Feature

This first lesson demonstrates the importance of creating CAD geometry that reflects the design intent and the manufacturing processes being used. Had the hole feature not been correctly specified in the CAD model, subsequent model updates that changed the length of the head feature would place the cross hole out of position to the original design intent.