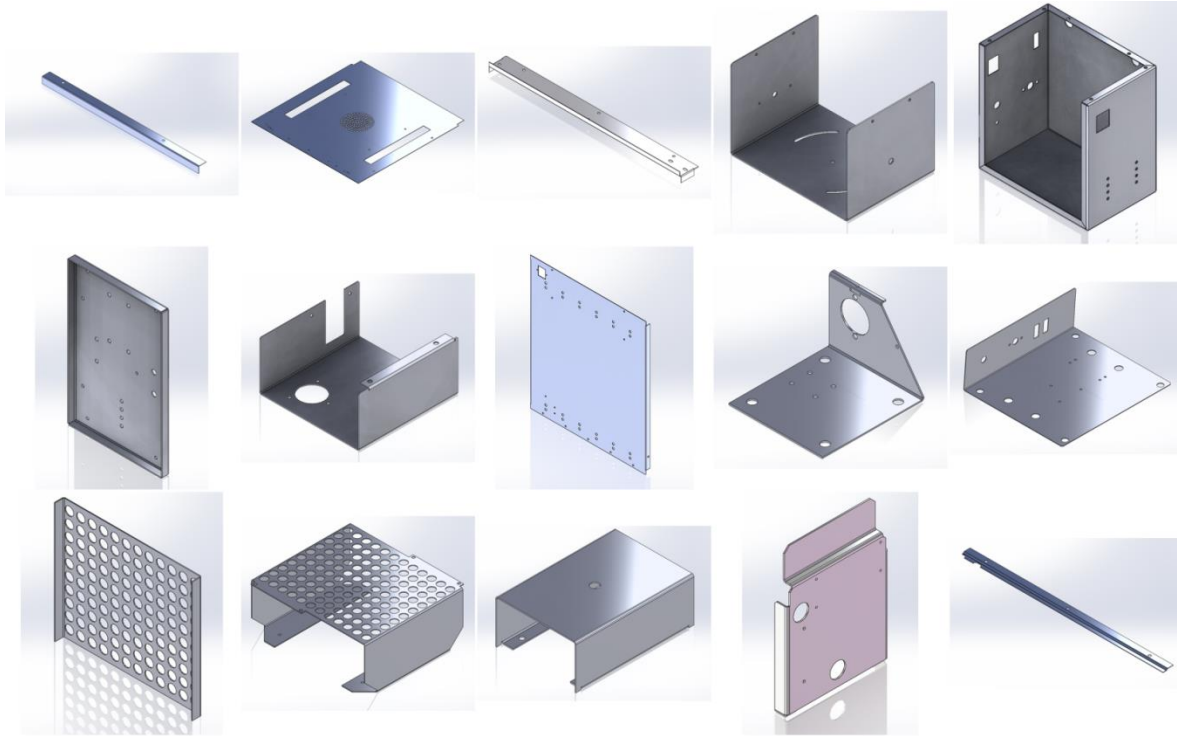


DESIGNING WITH SHEET METAL



Objective

The purpose of this short paper is to demonstrate how I approach the design of sheet metal assemblies.

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Designing with Sheet Metal

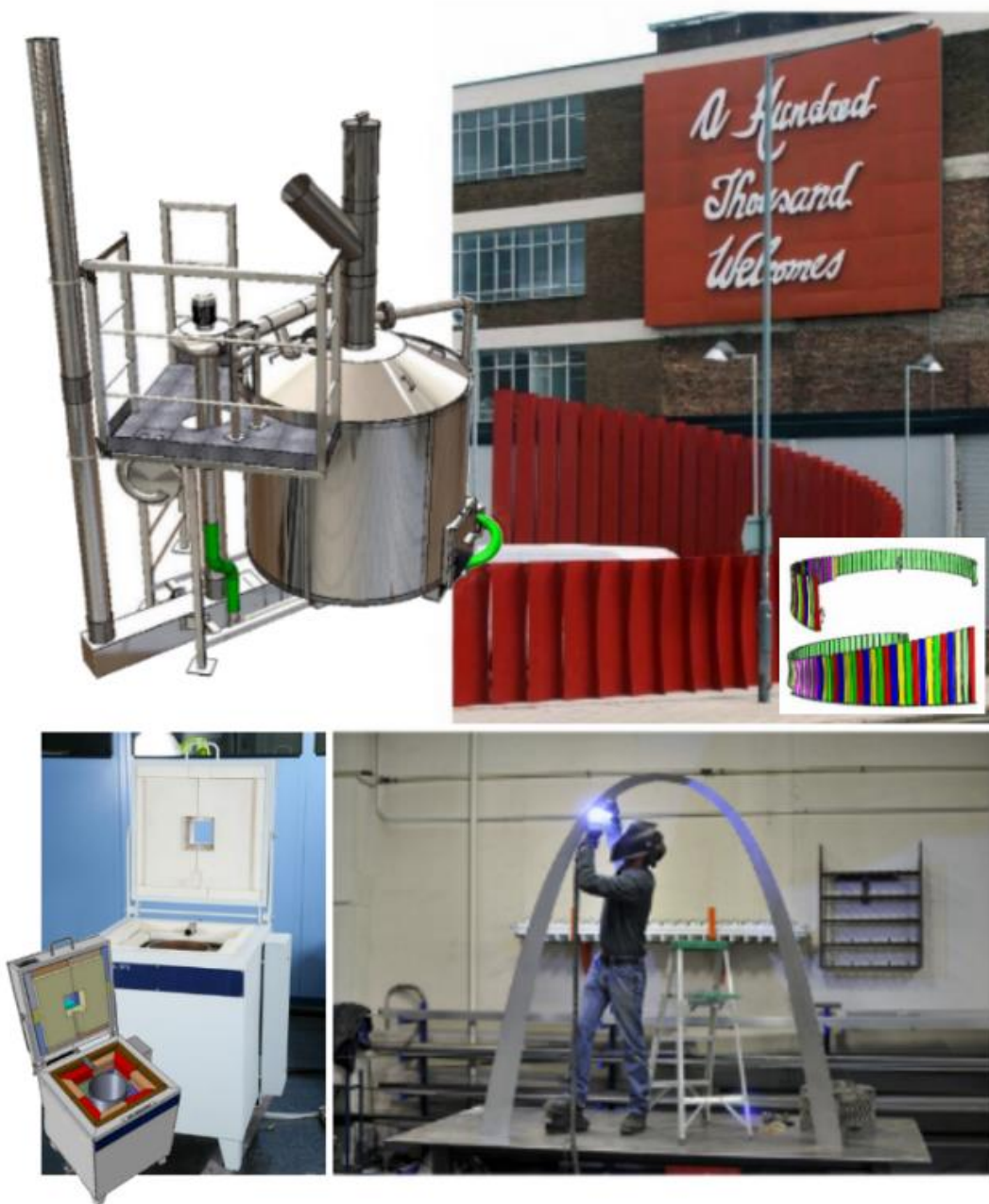
Introduction

Sheet metal parts are found in an enormous amount of day-to-day products.

Both sheet aluminium and sheet steel are tough, low cost and very versatile solutions.

My goal with this paper is to give you a quick overview and then show you how I will typically tackle a complicated sheet metal assembly.

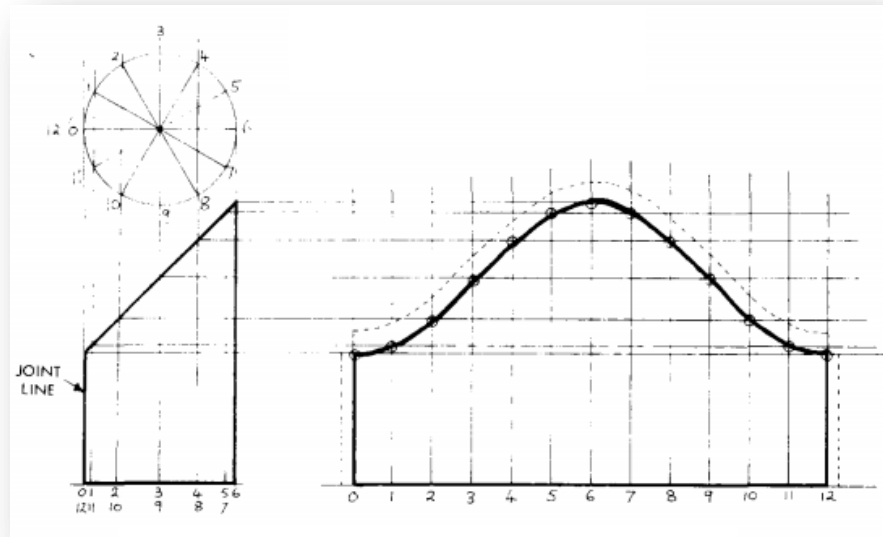
On this basis we start overleaf with the software that I use...



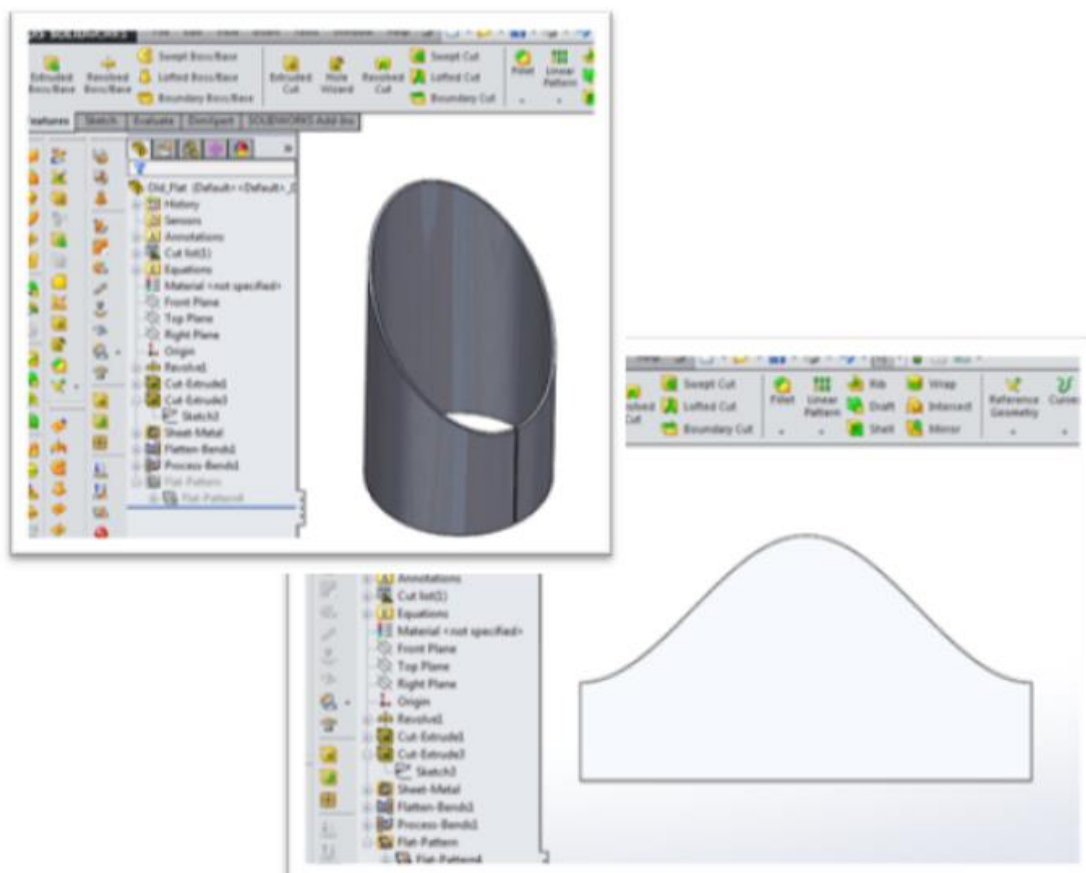
Portfolio examples clockwise from top left: Industrial equipment for creating biochar, an award winning UK art installation, a scale model of the world's largest arch and an industrial temperature calibration unit

Sheet Metal Software

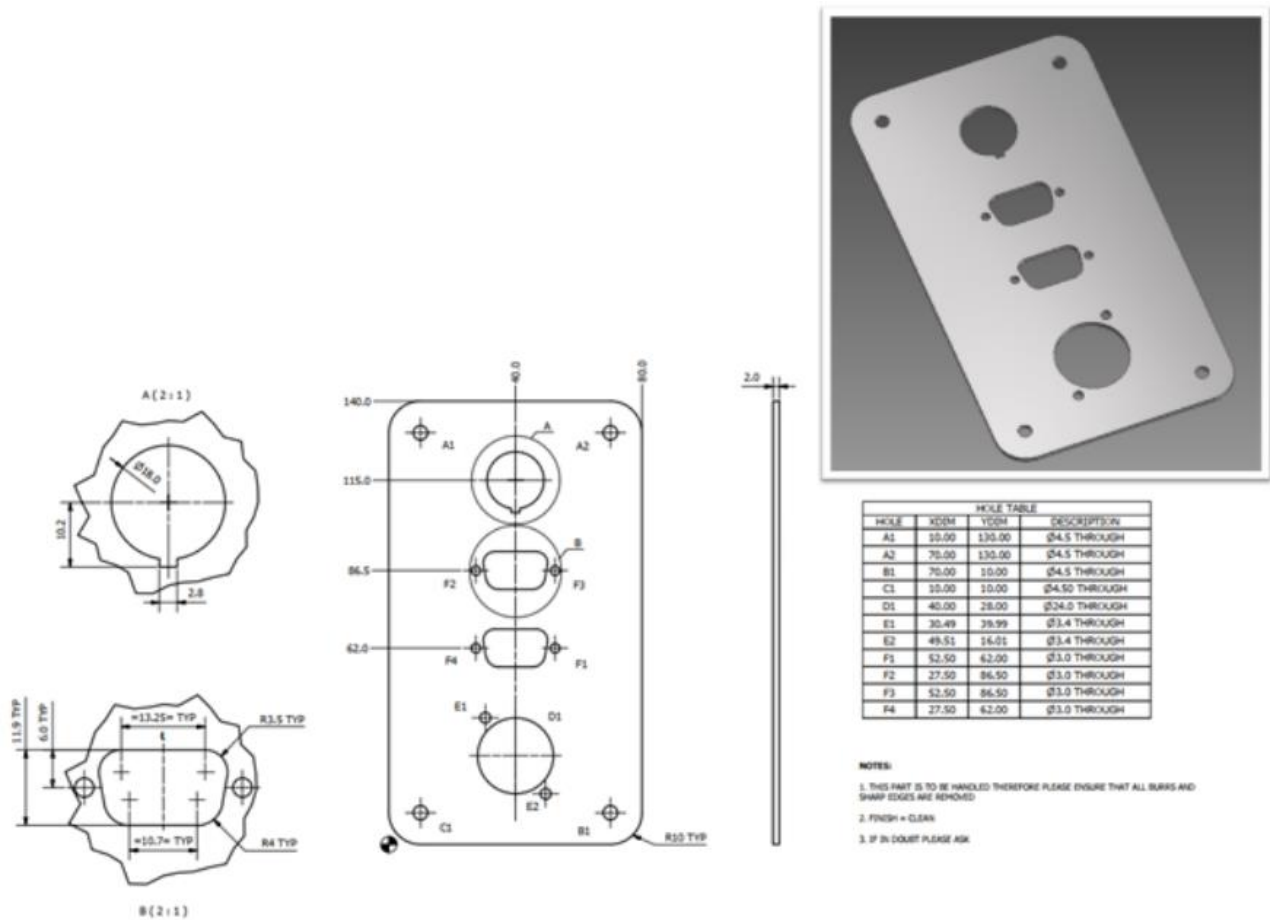
You might be aware that to make a part from sheet metal we typically need a pattern of some kind. In the past the patterns for the cutting and folding of sheet metal were skillfully developed and marked out by hand.



Computer aided design (CAD) software allows us to follow a similar process but in reverse. We create 3D models of the sheet metal part which can then be unfolded down into the corresponding 'flat pattern'.

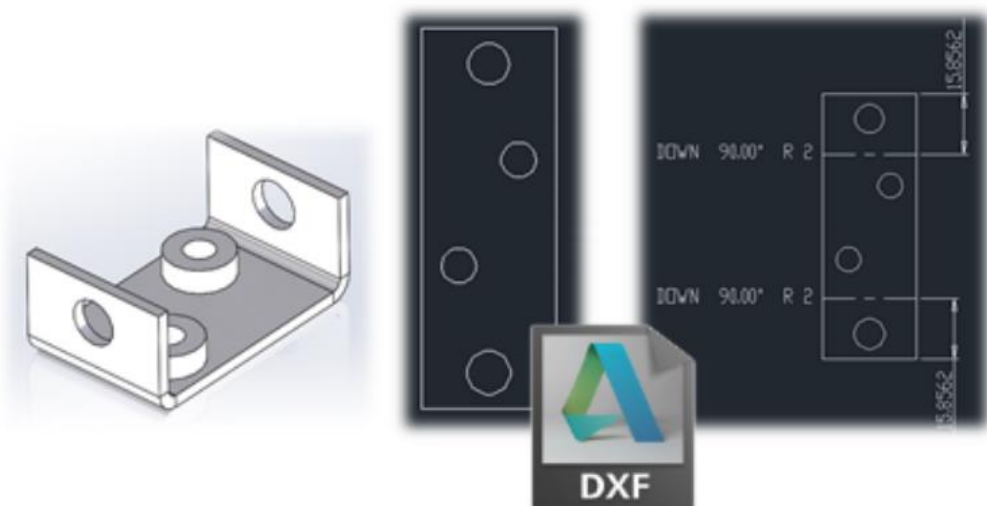


CAD software also allows us to take the 3D model and then place specific views of it onto a sheet of paper. This process is what leads to a dimensioned engineering drawing.



In addition to creating manufacturing drawings it is common to work with a nominated fabricator and supply then an additional element known as a '.DXF file'.

A .DXF file contains the cutting profile and any additional folding information that the fabricator might require for the production of the part.



The CAD software that I use for designing sheet metal parts will be either Solidworks or Autodesk Inventor. The preference over which I will pick is usually dictated by the client's requirements.

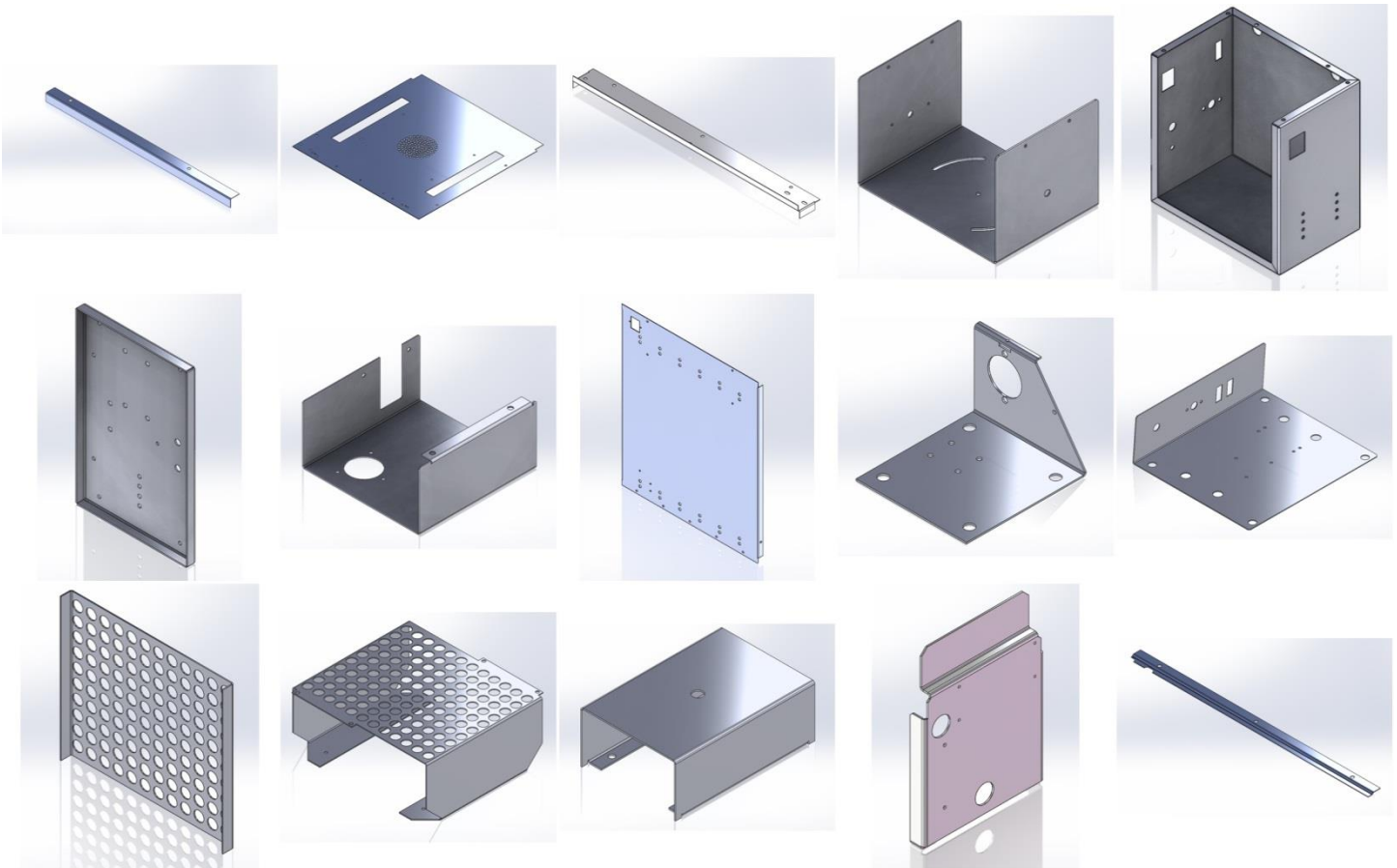
I have extensive experience with both and have found that there is virtually no difference between their sheet metal capabilities.



solidworks.com



autodesk.eu/products/inventor/overview



Skeletal Modelling

When designing with sheet metal the approach I typically use is one known as “*skeletal modelling*”.

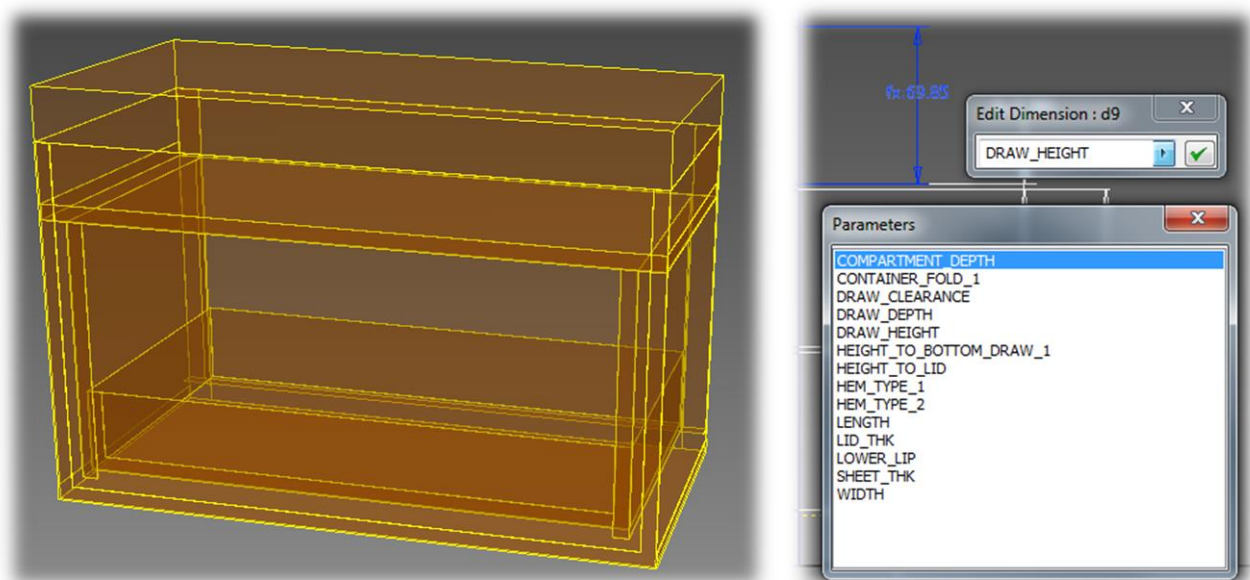
To explain this technique and why it can help gain us an advantage, the design of a tool chest using Autodesk Inventor will help serve as an example...



Now at the start of any design job it is not uncommon to be unsure as to what the exact final dimensions will need to be.

To get round this problem I use what is known as a ‘*master part file*’.

A master part file is the foundation or ‘*skeleton*’ upon which the rest of the design will be built upon and it typically contains only the most important surfaces of the design.



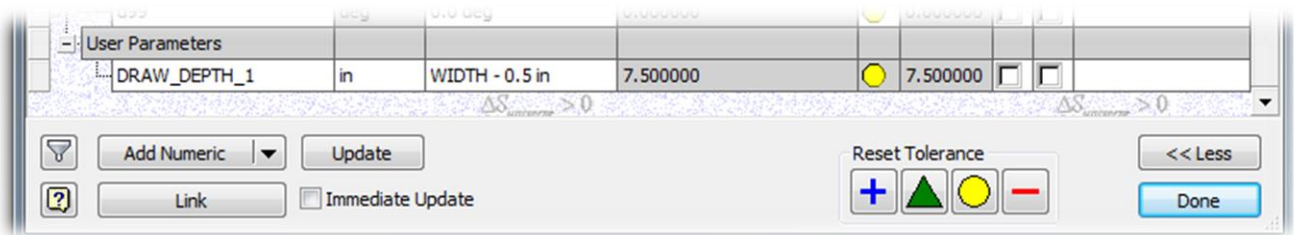
The key surfaces that you can see shown in yellow above are controlled in turn by ‘*key design parameters*’.

For the tool chest the key design parameters include;

- *total height*
- *total width*
- *total length*
- *sheet metal thickness*
- *number of drawers, etc.*

Typically a proportion of the key design parameters will also be used to generate '*design constraints*'.

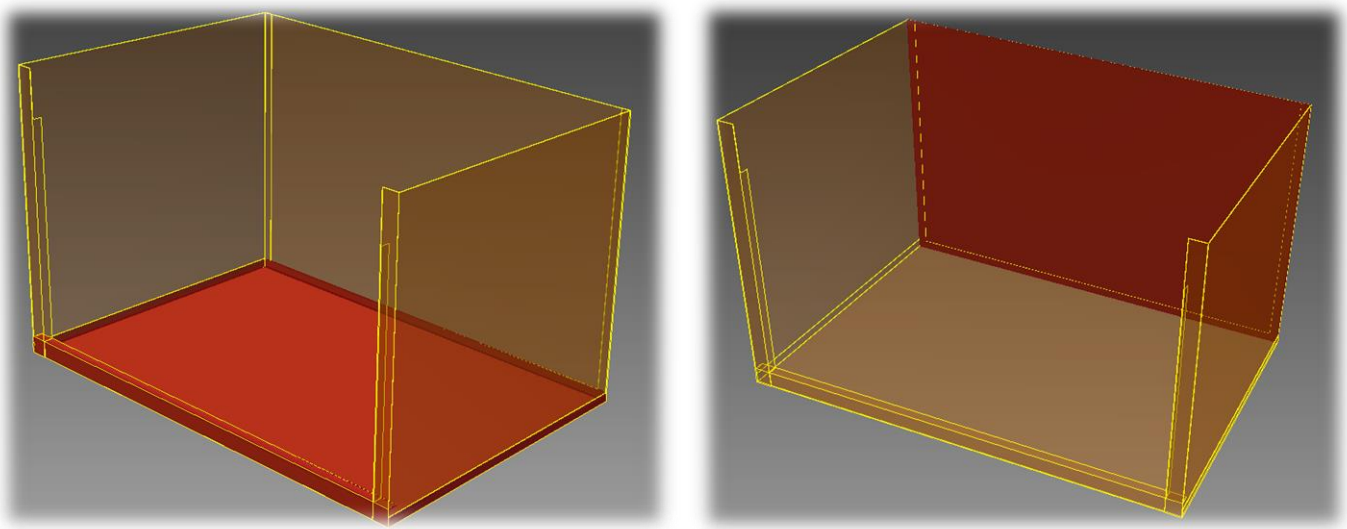
Ensuring that the tool chest drawers always finish day half an inch away from the inside face of the back wall is an example of a design constraint.



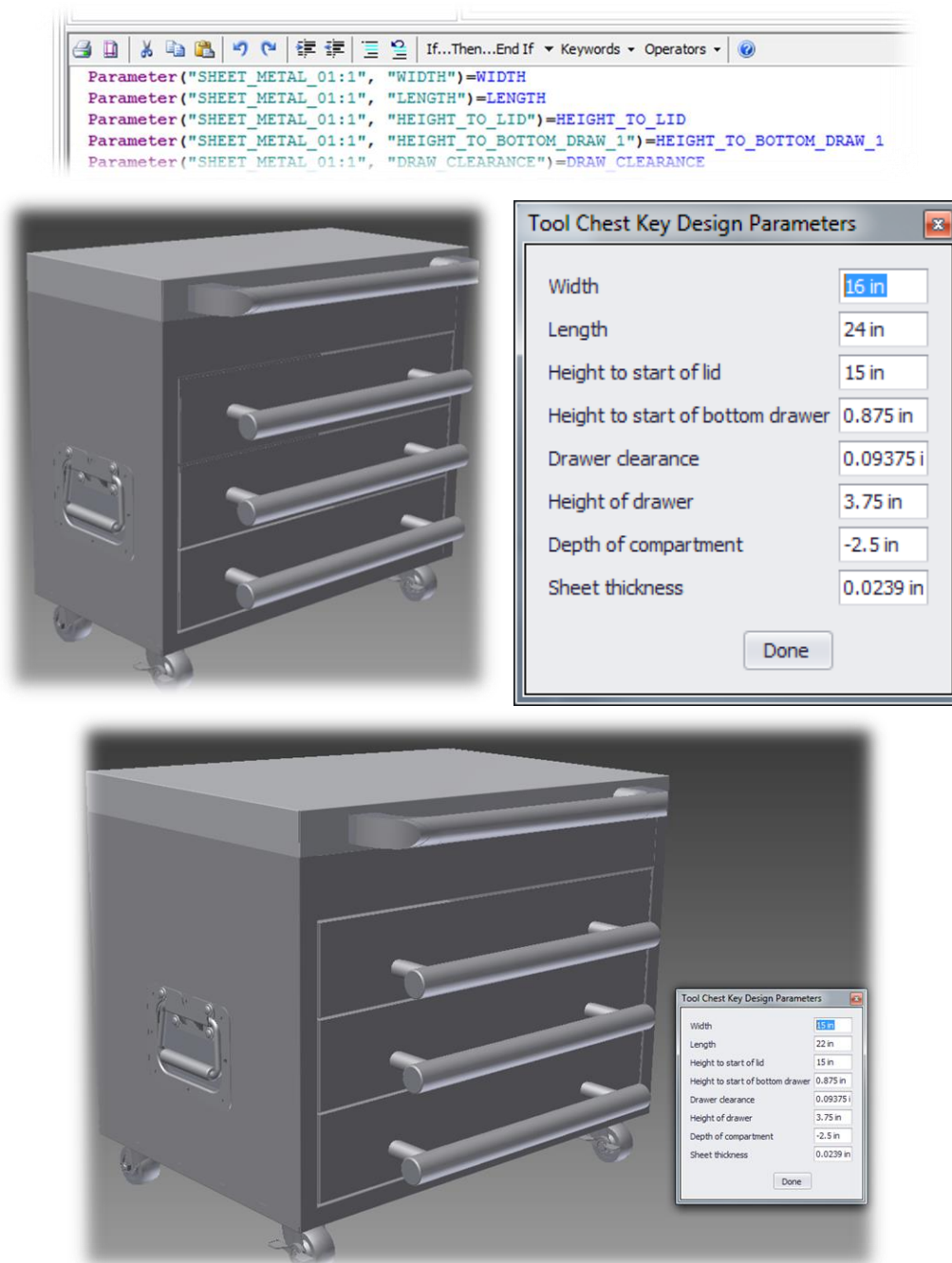
For more complex problems I will pass numbers back-and-forth with Microsoft Excel. Calculations can be performed within the spreadsheet and the results can then be transferred back in to drive the CAD file.



So once I have the master part file set up I can use it as an underlay to accurately create each of the individual sheet metal parts.



Then when I am happy with the design I will bring all of the parts together to form an assembly. The final step is to then write a small snippet of iLogic¹ code that creates a form within the assembly. This form gives us a very quick and easy way to quickly alter the key design parameters.



In the illustration above I decided to increase both the depth and width of the tool chest via the form. As soon as I click “Done” every single sheet metal part instantly updated to meet the new dimensions.

So once we are happy with all of the dimensions what happens next? Well typically I would be asked to create a rendering of the assembly and then submit a full set of manufacturing drawings for final approval.

1. iLogic is a very powerful programming add-on that comes with every seat of Autodesk Inventor. The Solidworks alternative to achieve the same results would be via design tables or through a separate piece of software known as DriveWorks.

Summary

So why go to the trouble of working the way that I have shown? Would it not be a lot faster just to draw up sheet metal parts on their own and then create an assembly?

Well yes... for many projects that will in fact be the most expedient approach.

However complicated design projects demand robust CAD assemblies that we can quickly adjust to accommodate changes made upstream.

By taking the time to build in flexibility at the start it can substantially reduce the time it takes to get a set of sheet metal drawings to the fabricators.

Case Study

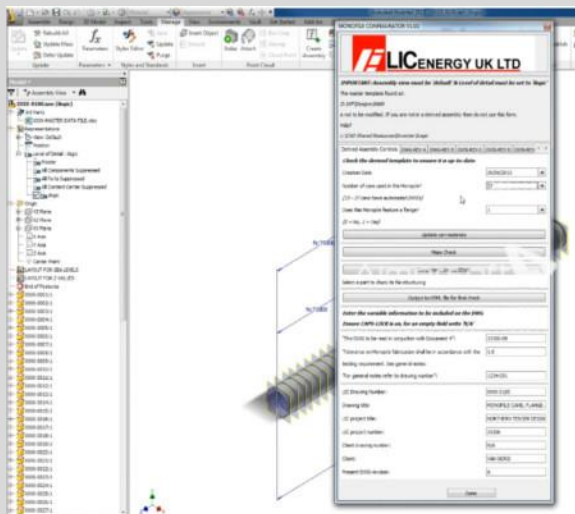
A monopile is essentially a large tube made from rolled steel plate² driven down into the seabed to allow for the installation of a wind turbine.

The problem is that the design of each monopile must differ to account for the local site characteristics. These slight differences create a huge amount of repetitive design and draughting work.

In much the same way that I have shown for the tool chest in this short paper I used Autodesk Inventor to create a configurable master monopile that allowed for approved dimensions to be inserted via a bespoke form.

This bespoke form could then automatically generate a 3D assembly along with all of the associated manufacturing drawings.

Skeletal modelling reduced the time spent modelling and draughting a monopile tower from one **week** to less than one **hour**. As my client was responsible for 116 monopiles the investment spent in establishing a new workflow was repaid in a short period of time.



2. I consider anything from 0.25mm to 6mm to be sheet metal. Anything over 6mm I will call plate and anything under 0.25mm is known as foil

