Split Command in Inventor

A summary of the split command within Inventor and showing some of its many uses, face split, trim solid and spilt solid.

All Inventor Users

The 'Split' command within Inventor is a really useful tool that is quite often forgotten about or overlooked, this white paper gives you a brief run down of its uses.

First we need to start with a solid body. For this example I am going to use a simple cuboid shape. You can then access the 'Split' command from the modify section of the '3D Model' Tab in your ribbon.
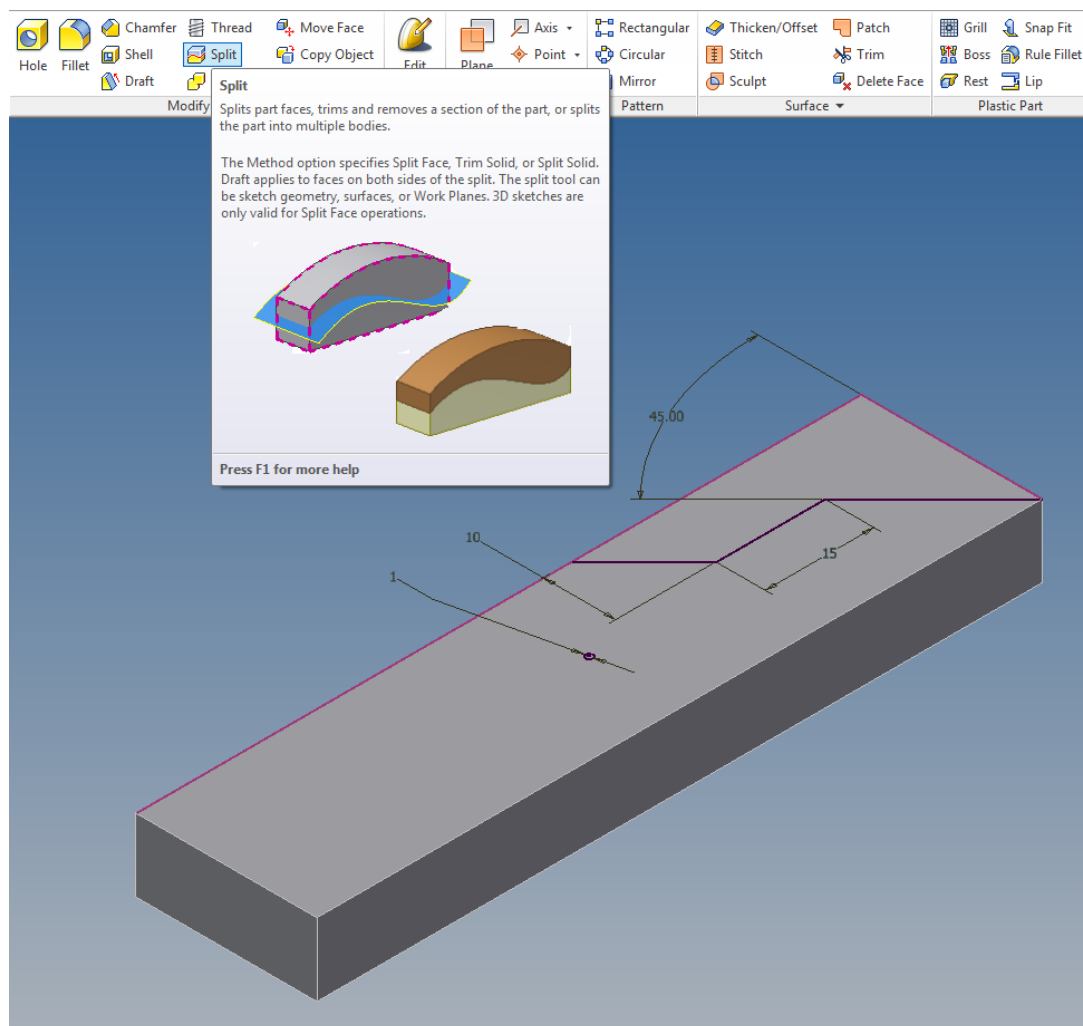

As you can see I have drawn a sketch on the top face with a number of different lines in. This will allow me to use the different functions in the split command. You don't need to draw a sketch as planes can also be used.
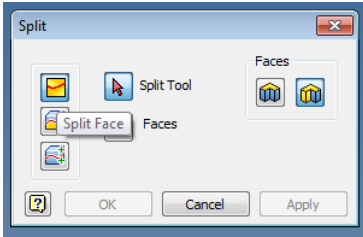
'Split Face' allows you to use a sketch or plane to split a face into sections.

To use this first select the 'Split' command from the ribbon and then the 'Split Face' function. You then will need to select your Split Tool (a sketch or plane) and then the face you wish to split. For my first example I used the XY Plane as my Split Tool and selected the top face to split.
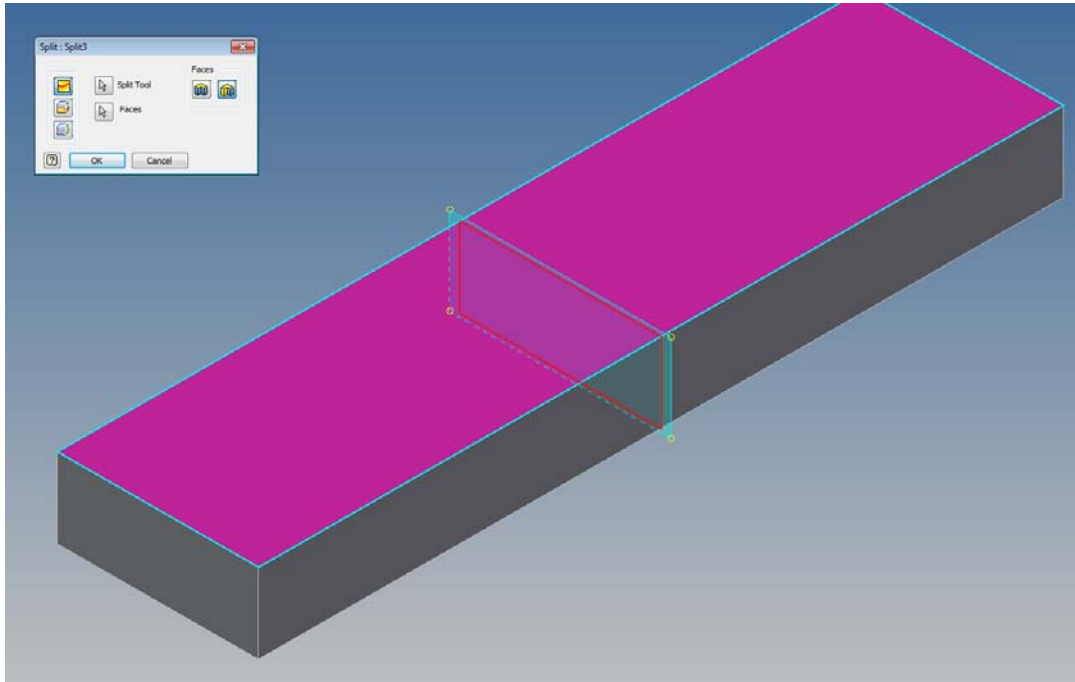

The result from the 'Face Split' is that the top face of my cuboid is now two individual selectable faces.
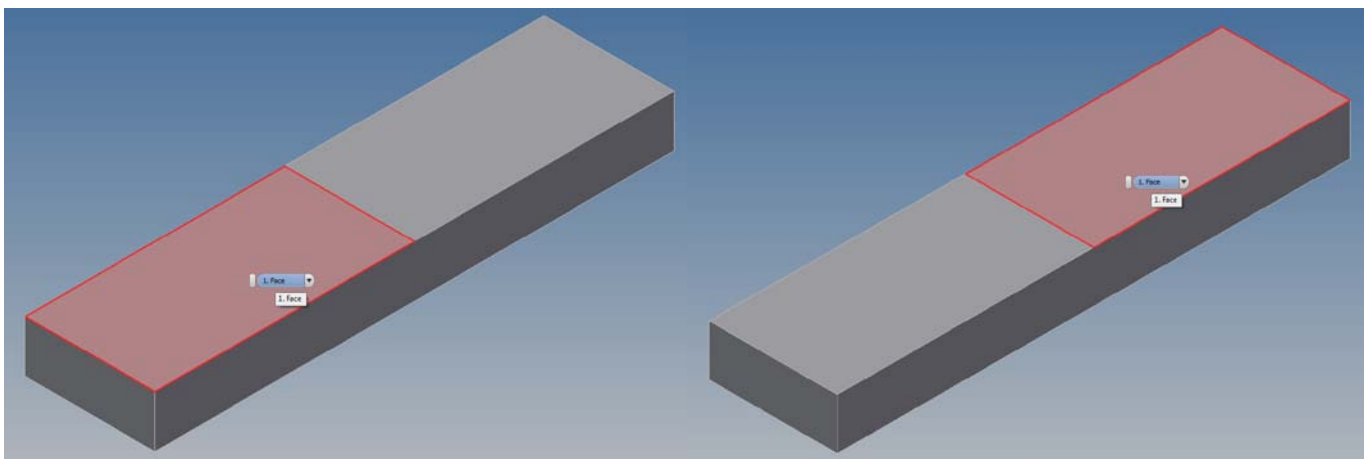

When you are creating the 'Face Split' there is also an option which allows you to split all the faces intersected by your chosen Split Tool. This can be useful if you are creating plastic parts which need all of the faces drafted for a mould split line.
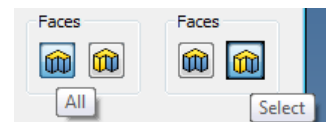
But for this example we have just split the top face which we can then apply different draft angles to.
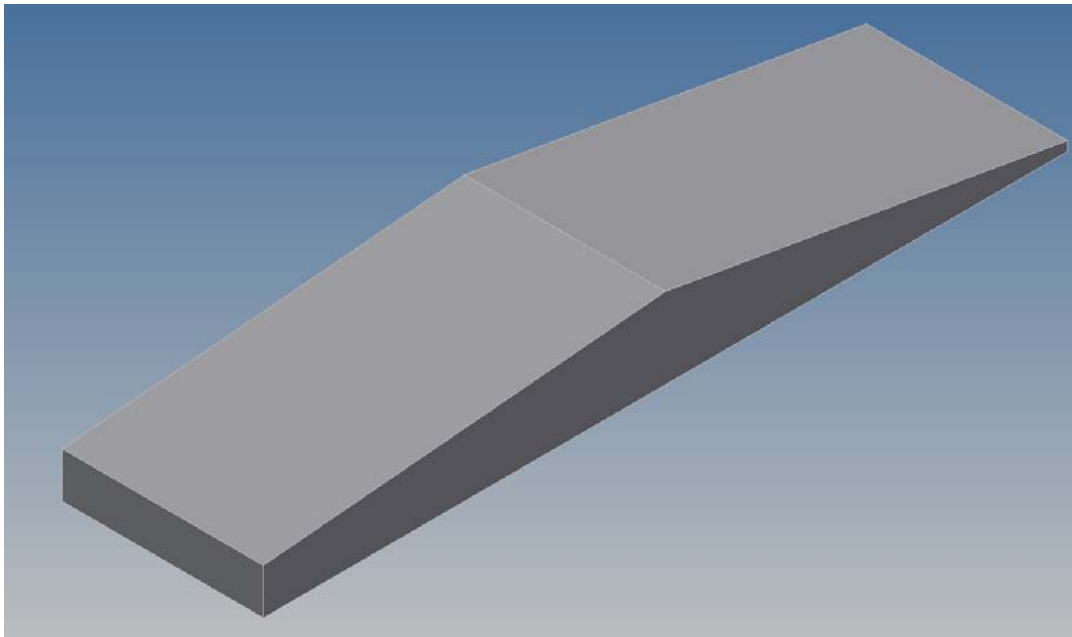
A second use of the 'Split Face' function in conjunction with drafting is it allows you not to draft a complete face. Below you can see the Split Face function using the circle from my sketch as the Split Tool. Now when drafting this new face is not affected so you either get a spigot or a hole depending which way your draft angle is selected.
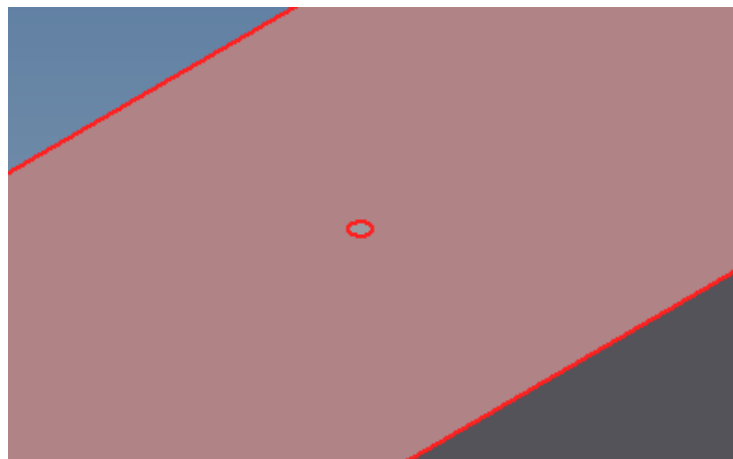
A third use can be in 'Stress Analysis'. When applying forces you can only use faces or vertices. So if we want to apply a force to the centre of a face we need to first create a split face in the centre of the full face similar to the circular face above.

A forth use of the 'Split Face' tool allows you to create complex shelled parts. In the below example the top face was split with a simple rectangle. Then when creating the shell the centre face was selected and the "Automatic Face Chain" option was unchecked. This allowed the part below to be created.
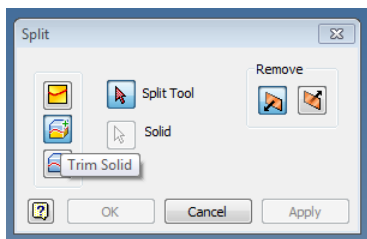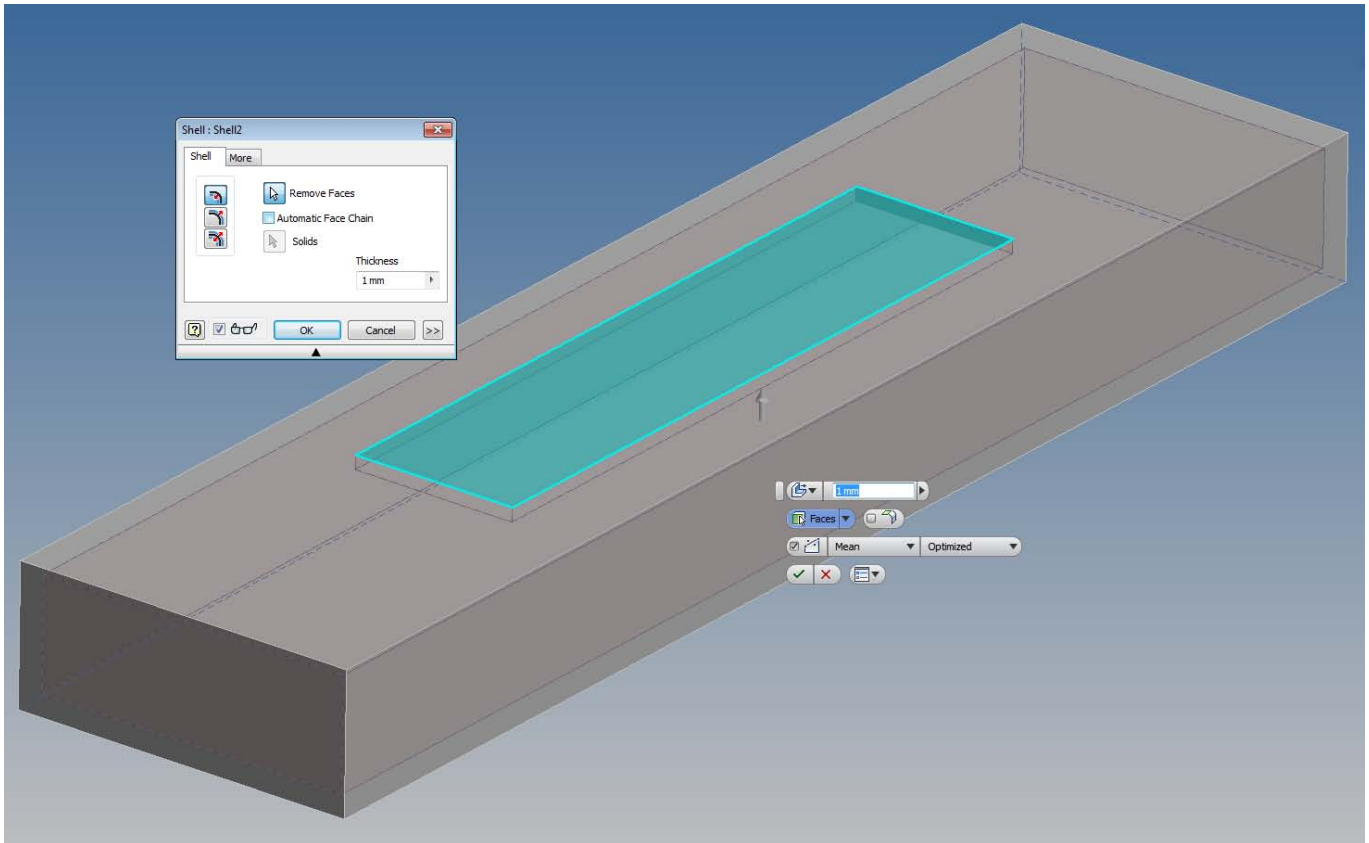

The next function available to us in the 'Split' command is 'Trim Solid'.

This function allows you again to use a sketch or plane but this time instead of splitting a face it allows you to trim a solid. If you need to cut the end off a part or put a complicated chamfer/draft angle this is a very useful function and a simpler way of trimming a part, compared to drawing an oversized sketch and using the extrude option.

First you need a sketch or plane to use as your Split Tool. For this example the crooked line in the original sketch is selected (note, you can select a line from shared sketches and you can share sketches already embed into a 'Split'). Then select which solid body you want to trim. In this case there is only one so it is automatically selected.

We can then decide which side of the Split Tool we want to keep and trim.
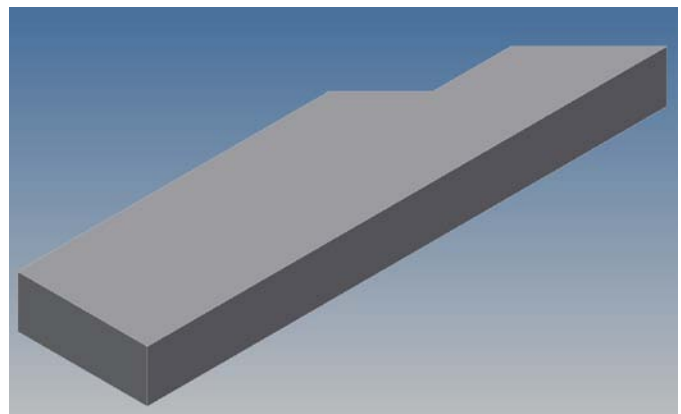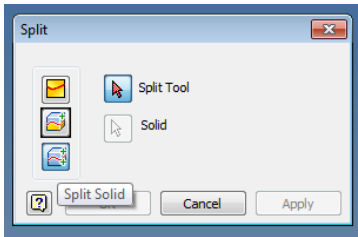
The final function in the 'Split' command is 'Split Solids'.

This function is similar to both the previous functions shown, but it separates one solid into multiple solids. Again you are required to select a Split Tool and a solid body. If we use the same Split Tool from the previous example, the crooked line. But instead of choosing which side to trim away both sides of the line are kept, but they are now separate solid bodies.

We then get the resulting output which is shown below.
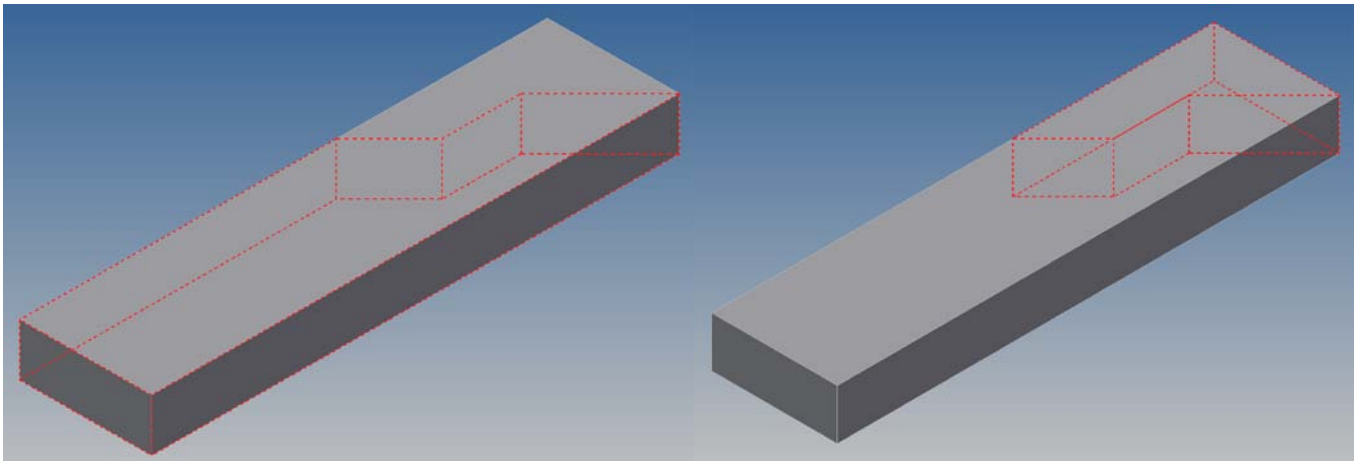
Having two solid bodies in your part allows you to then assign different material properties to each of these solid bodies, calculate individual masses, apply combined or separate features to each body, save off each body as individual parts (but still linked to the original .ipt) and much more (note, keep an eye on your model tree).

You can find a really good video on CADline Community about working with multiple solid bodies within one part.