**Mould using Zero Offset**

In the following tutorial we combine manually defining surfaces with mould tools to create our mould. Where a mould is too complex to rely on automatic core / cavity detection we can revert to manually defining all surfaces.

These surfaces can be created by offsetting “solid” faces by Zero millimetres to create replica “surfaces”.

Having created our:
- Core surfaces
- Cavity surfaces
- Split surfaces.

The core and cavity inserts will finally be defined.

Here we will use this simple component to demonstrate the concept.

**Modelling the component**

First of all model the component according the dimensions shown below.

We will now create core and cavity inserts. Leave an allowance of 40mm in all directions. (Ensure that all parting surfaces extend the extremities of the mould).
Modelling the split surfaces

We will now create the split surfaces. First we will create those areas which deviate from the flat. We will then fill in the remaining flat portions.

First using the sweep option, create a ruled surface for the region shown of appropriate length. Use the edge shown as the “reference vector” to guide the direction of the surface.

Repeat the process on the opposite side.

Next sketch the boundary for the flat portion of the split surface. Again make larger than the intended mould blocks. (use convert entities where appropriate to utilise existing edges).

Again repeat the process for the rear of the component.

The component and its associated split surface should now look as shown opposite.

The split surface is currently represented as 4 individual surfaces in the surfaces bodies’ folder. These surfaces may now be joined into a single surface using the knit command.

Not only does this act as a good housekeeping exercise, (by tidying 4 surfaces into one) it also enables gaps to be detected and repaired if such gaps exist.
Defining Cavity surfaces

Now rather than using automatic detection we will manually create the cavity surfaces.

This will be achieved by offsetting the surfaces of the solid by zero mm.

To offset the surfaces choose: Insert Surface Offset.

Regular selection selects just one surface however by right clicking and choosing “select tangency” all surfaces which are tangential will be selected.

Finally specify a offset distance of 0mm and choose accept.

As these surfaces come from a solid they will already be knitted or joined together.

Defining Core surfaces

To define the core surfaces repeat the process on the underside, again using the select tangency option.

This will not select all the surfaces needed. Use the control key (Ctrl) to select the additional surfaces (shown) which bring the core surfaces up to meet the existing split surfaces.
Creating the mould

Finally to create the mould. Draw a rectangle around the part. (specify appropriate dimensions to leave 40mm in all directions). Next issue the “tooling split” command.

The core cavity and split surfaces have not been categorised automatically into subfolders and therefore will not be automatically detected.

Instead they can be manually selected simply by activating each section in turn and then selecting the appropriate set of surfaces.

The command should work provided there is no surface overlapping and provided the split surface extends beyond the boundaries of the mould.

The resulting mould if expanded should look as shown opposite.

Assembling and drafting the mould

Finally export the mould components to individual parts, re-assemble them using SolidWorks assembly and finally draft the mould using regular orthographic views, isometric and exploded views.