Mouse mould using Mould tools

In the following exercise you will model the core/cavity for a mouse similar to that used in a previous tutorial. This component may be downloaded from:

S:\classes\sean dalton\1\tooling

Create a folder called mousemould on your H: drive then copy mousetop.STEP into this folder. Open the component called mousetop.STEP. The component is supplied to you as a step file. This is one of a number of neutral formats used for translating geometry between CAD systems. A feature of this translation process is that the ‘history’ of how the component was created is lost. All that remains is the resulting geometry as a single item in the feature manager ‘Imported1’. However this does not prevent us from modifying the part if required.

N.B. For the purposes of the exercise some features already modelled have been removed to illustrate certain concepts.

Draft analysis

Injection moulded components must have some degree of draft or taper to facilitate easy removal from a mould.

Draft analysis identifies surfaces which require draft i.e. surfaces less than a specified angle to the line of draw.

For this exercise we will specify a minimum angle of 1 degree. Draft analysis also identifies those surfaces which will be formed by the core and those which will be formed by the cavity.

To evaluate the component select draft analysis icon (or Tools Draft analysis).

Select the plane shown and specify an angle of 1 degree.

An selecting the plane an arrow will appear. The arrow should point toward the cavity side of the mould. If not it may be reverse by clicking on the arrow or selecting the icon.

Select ‘Face classification’ followed by [Calculation].

Face classification produces a count of the number of surfaces of each type.

Green surfaces are surfaces which will be formed by the cavity side of the mould. Red surfaces are
surfaces which will be formed by the core side of the mould. 

**Yellow surfaces** are those which require draft.

Choose ☑️ to accept the results of the draft analysis and when prompted choose **yes** to retain the colours.

**Applying draft where required**

Add a draft of 2 degrees to the inside of the lip using the flat face (upper most edge in the view shown) as the Neutral plane.

Also draft the screw hole bosses outwards by 3 degrees using the flat face at the end as the neutral. As all 3 bosses share a common neutral plane they can all be drafted in the one operation.

On re-running the draft analysis it will be found that these surfaces now turn red also.

The part is now ‘ready for moulding’.

**Applying scaling**

All component shrink as they cool therefore in order for a component to achieve its required final dimension it must be moulded oversize to allow for shrinkage. For this exercise we will allow **5%** for shrinkage.

To apply the shrinkage factor select scale. 
(Insert Molds Scale…).

Scaling may be about: Centroid, Origin, or Co-ordinates system (i.e. a user define coordinate system) and may be with either uniform or may be scaled by different amounts in each direction.

In this case apply a uniform scale of **1.05** about the centroid i.e. centre of gravity.

Conclude by choosing ☑️.

**Generating parting lines**

Run the parting line command (Insert Molds Parting line).

Again selecting the top plane and ensuring the arrow points upwards select [draft analysis].

This evaluates the component and identifies the edges which will form the parting line. (This will later be used to create the split surface).

**Adding shut off surfaces**
Where ever there are holes in a part these will be created by metal to metal contact between the core and cavity; what is referred to as shut offs. We need to determine these in order to define the surface in this part of both the core and cavity.

To determine the shut-offs. Choose Insert Molds Shutoffs. This identifies all of the lower edges at the button openings.

Choose accept to create these shutout surfaces. Not only does this command create surfaces in the region of the holes it also creates two complete sets of surfaces. On examining the feature manager more closely we see near the top two new categories of feature. Surface bodies and Solid Bodies.

Expanding the solid bodies folder we see a single item. This is the original solid geometry of the mouse. Expanding the surface bodies folder we find two items:

- Cavity surface bodies
- Core surface bodies.

Showing cavity surface bodies and hiding everything else reveals all of the green cavity surfaces. These are the surfaces which will form the exterior of the part. Similarly showing all the core surface bodies (and hiding the rest) show all of the red core surfaces or surfaces which will form the interior of the part.

**Creating parting surfaces**

The split line may be tangential to the component surface, normal to a components surface or perpendicular to the line of pull. Select Perpendicular to Pull. The parting line created earlier is automatically added to the parting line box.

This surface can fan out all the way forming the complete split surface of the mould or it go out a short distance and then drop to a lower main parting surface. This will result in an interlock which helps the core and cavity locate together accurately. We will opt for the latter.
Specify a distance of **10mm** for the parting surface to fan out.

**Preparing for the Tooling Split**

Create a parallel plane **15mm** below the existing top plane.

Now select the Tooling Split command. Rather than splitting an existing solid block, the core and cavity will be created by extruding the block and splitting it in the same step, using the Split Tool command. Select the parallel plane just created. This will put you into sketching mode.

Select the view normal to option and create the sketch shown opposite. On completion exit the sketch.

On doing so Solid Works automatically detect the core cavity and parting surfaces.

Specify distances of **60mm** upwards and **30mm** downward.

Activate the interlock option and specify an angle of **3 degrees**.

This creates the core and cavity inserts.

A slice through the resulting tool reveals the core cavity and the component (shown cut for illustration purposes only).

**Moving Core/Cavity apart**

To examine the core and cavity in detail we will move the core and cavity apart.

This can be achieved by using the move/copy tool (or Insert Feature Move/copy bodies).
Select the top portion of the mould or cavity half and issue the move copy command. You will initially be presented with assembly like constraints however at the bottom of the property manage you will see a

Select this option.

You have the option of creating a copy (do not select as we do not require a copy on this occasion)

Alternatively you can manipulate the part interactively by used using the graphical tool shown opposite.

Selecting and dragging an arrow allows you to drag the components in that specific direction. Selecting and dragging a hoops or rings allows you the rotate the object about a specific axis. Selecting the purples quarter segments allows you to drag the component in that particular plane.

Use the move/copy bodies to reposition the core and cavity as shown. Examine the core and cavity surfaces. Once finished move the two halves back together by suppressing both move/copy operations.

While we now have 3 distinct solid bodies we are still working within a single part file. For drafting purposes, it is preferable to export these solid bodies to three separate stand alone parts. To do so:

**Exporting as separate parts**

Expand the solid bodies folder and choose the item which represents the core. Right click on the item and choose Insert into new part.

Solid works creates a new part containing the core geometry. In the dialog which appears, specify the name CORE.SLDPRT and save it in your working folder.
Repeat the process for the cavity block and save it with the name CAVITY.SLDPRT.

Finally, create a stand alone model of the part itself as MOUSETOP.SLDPRT.

Now, create a new assembly called MOUSE-MOULD.SLDASM and assemble the core cavity and part model.

You will now add some finishing detail.

**Adding finishing detail**

The modelled components are only the core and cavity inserts. To complete the functioning injection moulding tool these will need to be inserted into a mould base.

As the both insert will be inserted into milled out pockets we will need to round the corners and insert threaded holes on the top and bottom faces.

Round the corner using a fillet radius of 20mm and insert M10 tapped holes 30mm deep at the centres of these fillets.

Repeat the process for the core insert. This time the holes will need to be shallower so as not to pierce through.

**Submission Details**

Create a drawing of the mouse mould assembly which contains:

- Appropriate orthographic views – i.e. plan, elevation, side views
- Section views and detail views as appropriate
- Regular isometric and exploded isometric views
- Parts list and item references

Make effective use of orthographic, section and detail views to communicate and highlight model detail.

All orthographic views should be contained in a single drawing on an A4 sheet. Isometric views and Bill of Materials may be supplied on a second sheet.