

Creo Parametric 2.0 Tutorial

***Creo Parametric 1.0 Tutorial and MultiMedia DVD* was written for Creo Parametric 1.0. PTC released Creo Parametric 2.0 in the Spring of 2012. This book is fully compatible with Creo Parametric 2.0 except for the changes shown in this insert.**

Chapter 1

1-23 The ***Reorient*** command is located on the ***Named Views*** drop-down list.

1-25 To turn model colors off, go to

File > Options > Entity Display

and turn off the check beside **Show colors assigned to model surface**. If you do this often, consider adding a command to the Quick Access toolbar at the top.

Chapter 2

2-10, 20, 22, 26, etc. “Yellow” arrows are actually magenta for the extrude directions.

2-19 Sketcher open sketch endpoint dot is red instead of green; this happens on all sketches.

2-23 In the **Extrude** dashboard, the **Options** group also contains a “Taper” option.

Chapter 3

No changes.

Chapter 4

4-19 The ***Measure*** drop-down command list, in addition to allowing you to directly launch one of the analysis commands (***Length, Diameter, Area, Angle***, etc.) opens a small window containing commands to quickly change to a different measurement. Also, the result is shown on a leader and result box on the screen. You can, for example, pick some entity on the model (an edge or surface, for example) and launch different measure tools referring to that entity. This is pretty easy to figure out, and of course there are shortcut commands on the RMB.

Chapter 5

5-10 In the Children Handling sequence, references are shown in red not green.

Chapter 6

- 6-13 Arrow showing direction of view onto sketch plane is magenta, not yellow. (Also occurs on page 6-17.)
- 6-17 The *Use Edge* button in Sketcher has been renamed as *Project*. The button icon is the same.

Chapter 7

- 7-26 There is a typo on footnote 9. The Figure number should be 48.

Chapter 8

- 8-4 Format file *creo_Aformat.frm* contents updated.
- 8-8 In the table, the driven dimensions should be *addxx* and *oaddxx*.
- 8-19 The drag handles shown in Figure 20 are no longer used in Creo 2.0. You can select the dimension text and move it either along the dimension line, or perpendicular (in which case the dimension line moves with it). The witness lines can be selected one at a time (they will highlight in bold green), and one drag handle will appear beside the model. This can be used to move the witness line off the model. If additional cosmetic changes are required, select the dimension and in the RMB pop-up select *Properties*, then *Move*, *Move Text*, or *Edit Attach* commands may be used to modify the cosmetics.

Chapter 9

No changes.

Chapter 10

- 10-4 The footnote refers to a better way of creating sections. There is actually a new command on the **Model** ribbon for creating and managing sections.
- 10-11 On the *Intersect* tab of the **Extrude** dashboard, the setting to adjust the display level is called **Set Display Level**, not **Default Display Level**.
- 10-22 There is a whole new command sequence and options for creating sections, both in parts and assemblies. The Tutorial material on pages 22 through 23 is replaced with the material in the next section of this document.

Chapter 11

- 11-4 The arrow at the **Start Point** of the sweep is magenta (not yellow).
- 11-11 The *Blend* command now uses the ribbon interface and is much easier to use. The Tutorial pages 11-21 are replaced by the material in the last section of this document.

Chapter 10 – Sections

For users with Creo 2:

The next 3 pages replace the material on Sections on pages 10-22 to 10-23 in the book.

When finished, return to the book and continue with “Assembly Modeling” on page 10-23.

Sections

Cross-sections, or just sections, are very common in assembly drawings (which we will get to next). Although sections can be created in drawing mode (as we did with the pulley in Lesson #8), it is often handy to have them available when you are working in assembly mode. This can allow you to check for fit and/or interference between components, measure clearances, and just generally help understand how an assembly fits together. Sections are very easy to create. You can define multiple sections in the same model, of different types and at different locations, and display as many of them at the same time as you like.

Remind yourself of the orientation of the default assembly coordinate system - that will be relevant for what follows. Set your display as follows: shading on, colors off, datums off.

We are going to recreate the view of the assembly obtained with the assembly cut earlier in this lesson. When using an extruded cut to do this, our only options are to suppress or resume the cut and to determine which components were intersected by the cut. The **Section** command allows us to do a lot more, and much more easily. The section functionality in Creo works with parts as well as with assemblies.

-  Planar
 -  X Direction
 -  Y Direction
 -  Z Direction
 -  Offset
 -  Zone
- Figure Etgq'4/3**
 Vj g'Section
 r wnf qy p'menu

Go to the **Model** ribbon. In the **Model Display** group, click on the **Section** pull-down menu. This is shown in Figure Etgq'4/1. The **Planar**, **X**, **Y** and **Z Direction** commands will create planar sections, with the first being the most general. The **Offset** option allows you to create a section using a sketched curve to define the 2D shape of the cutting plane.

Select the **ZDirection** command. The model's **Section** dashboard opens as shown in Figure Etgq'4/2 and the **Section** dashboard opens as shown in Figure Etgq'4/3. This creates a view of the model with a cutting (or clipping) plane whose normal is the Z axis direction of the assembly default system. This is the same view we created earlier using the cutting plane. An offset value is given that can be changed by dragging on the arrow normal to the section surface or double clicking on the numeric value. Try that now, setting the offset to **20**.

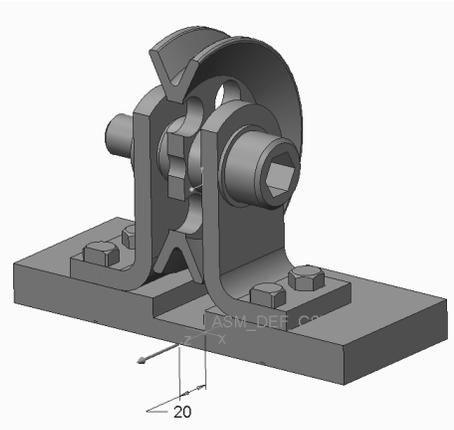


Figure Etgq'2/4 Section defined * "direction, offset 20)

Let's have a look at the dashboard - feel free to try these commands out as we go. Starting at the left, the orientation of the section normal is given in a drop-down list. The next box shows the offset distance from the chosen coordinate system (shown in the **References** tab below). The **Flip** button allows you to select which side of the section plane is removed. The **Capping Surface** button (on by default) toggles the surface created on the section plane. If the surfaces are capped, the next button lets you toggle on/off and set their color. The **Hatch** button does just that - toggles the

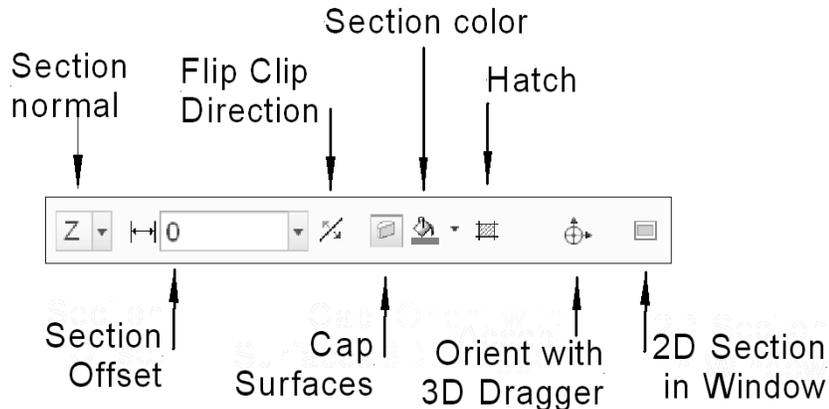


Figure Etgq'4/3 The *Section* dashboard

presence of a hatch pattern on each component. Notice that the hatch on each component is different. We will see later how these hatch patterns can be changed. Next is the *Dragger* for free positioning of the clipping plane. Select that now. Dragging on the arrows or rings allows us to change the position and orientation of the clipping plane. See Figure Creo 2-4. The *Dragger* is turned on automatically if you select *Planar* to launch the *Section* command. If you turn off the dragger, the clipping plane returns to the originally selected (Z, in this case) direction. Finally, the *2D View* button opens a new window to show the section plane in true shape (looking normal to the section). Select the *2D View* button in the dashboard again to close this window.

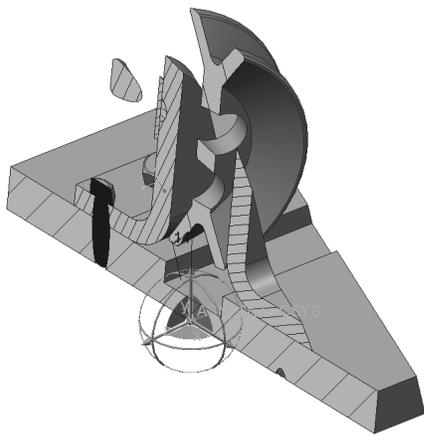


Figure Etgq'4/4 An oblique clipping plane created using the 3D dragger

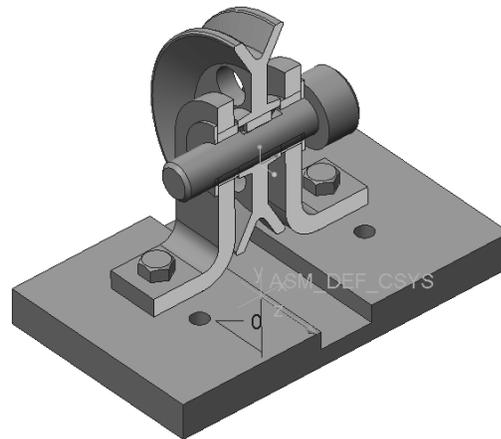


Figure Etgq'4/5 Base plate and axle components **Excluded** from section

Let's explore some of the section options in the dashboard tabs. The **References** tab lets you set the coordinate system used to determine the section normal direction. You can also pick on any plane surface in the model to specify the location and orientation of the section plane. The **Models** tab allows you to select which components to include or exclude from the section. Select the **Exclude** option and pick (with CTRL) the base plate and the axle. These components are now not clipped (Figure Creo 2-5). Note the option for selecting whether the section is defined for the assembly or a part. The next tab, **Options**, lets you toggle the visibility of interfering components. You can set the color of the interfering volume. Finally, the **Properties** tab lets you specify the name for the section,

as it would appear on a drawing. Type in "A" for the name of this section.

Change the offset back to 0, with our initial Z direction section. After setting all the section options, a middle click will accept the section definition. It will disappear from the model in the graphics window. However, it is now listed at the bottom of the model tree. Clicking on the section entry will show the outline of the sectioned parts. If you use the RMB pop-up menu, you can select *Activate* to make the section visible. With the section selected and activated, the RMB menu also allows you to execute most of the commands in the section dashboard.

Select the bushing component in one of the subassemblies, then open it using the RMB menu. Change the outer diameter to 35 and *Regenerate*. Return to the assembly window and *Activate* the section. Use the RMB and select *Edit Definition*. You should see a red highlight of the interference between the bushing and the L-bracket as shown in Figure Creo 2-6. If not, in the **Options** tab, check the box to show the interference.

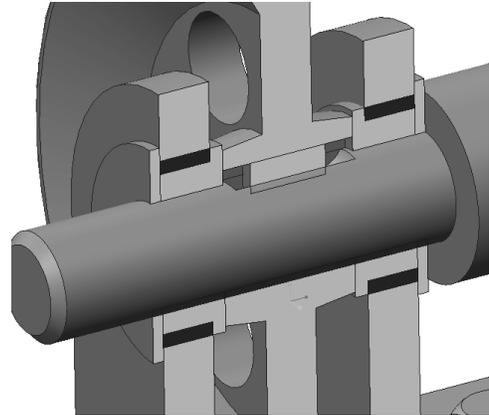


Figure Et gq'4-6 Interference of bushing outer diameter and hole in L-bracket

Note that in the RMB pop-up for sections you have options to pattern the section, flipping the clip direction, edit the hatch patterns and offset distance. You can also *Rename*, *Suppress*, or *Delete* the section. The *Show Section* toggle option controls the visibility of the hatch patterns.

Another way to deal with sections is with the *View Manager*. Open this window and select the **Sections** tab. You will see the section "A" we created previously. You can use the Edit and Option pull-down menus (and RMB pop-up) to access a number of commands for activating, editing hatch patterns, and so on. You cannot change the position or orientation of the section. Using *View Manager*, you can use defined sections as part of the display style.

Use the RMB pop-up to *Deactivate* the section. Don't forget to *Save* the assembly.

Continue with "Assembly Modeling" on page 10-23.

Chapter 11 – Blends

For users with Creo 2:

The next 9 pages replace the material on Blends on pages 11-12 to 11-21 in the book.

When finished, return to the book and continue with “Conclusion” on page 11-22.

Blends

A blend is like an extrusion with a changing cross section. Some systems call these “lofts.” The different cross sections are specified using a number of sketches or edge chains. These can be created either inside or outside the blend feature. The distance between cross sections is then specified either with references or explicit dimensions. The dimensions can be either a linear distance (forming a *parallel blend*) or an angular distance (forming a *rotational blend*), or a combination of these (a *general blend*). You can maybe imagine what a *swept blend* would involve. In the following, we will look at the first two of these. A blend can be used to create a protrusion or a cut. Some restrictions apply¹:

- ◆ At least two sections are required.
- ◆ Sections can be either selected from existing sketched curves, or created from chains of edges.
- ◆ Each section must have the same number of vertices; normally this means the same number of line (or arc) segments. This rule can be overridden using a *blend vertex* (see the on-line help for information on this).
- ◆ Each section has a starting point (one vertex on the sketch) - these must be defined to align all the sections properly or else the resulting geometry will be twisted.
- ◆ Parallel blends can use projected sections (obtained by projecting existing sketches or curves onto different planes).
- ◆ For a rotational blend, the section planes can be no more than 120 degrees apart.
- ◆ For a rotational blend, the axis of rotation is common to all sections, and all section planes must intersect the axis.

The sections of the blend can be connected either with *straight* (ie. ruled) surfaces (see Figure 47)², or with *smooth* surfaces (Figure 48).

¹ For some variations of the **Blend** command you must set the configuration option *enable_obsoleted_features* to *yes*. These commands (eg. **Project Section Blend**) will then appear on the **Commands Not in the Ribbon** list. For further information, see the on-line help.

² Note that this does not mean *flat* surfaces. A ruled surface is one where from every point along the curve defining one section there exists a straight line or edge to a matching point on the adjacent section.

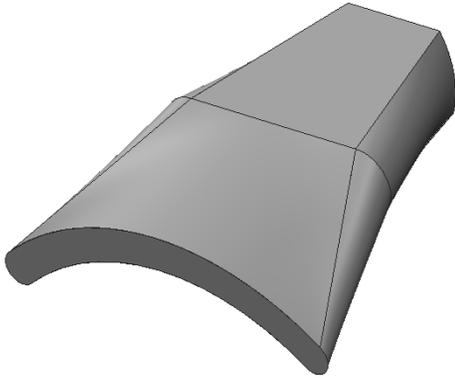


Figure 47 A *Straight* parallel blend

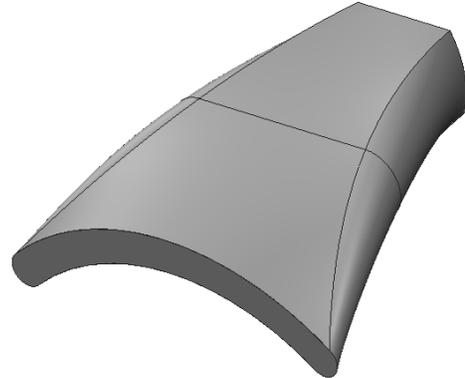


Figure 48 A *Smooth* parallel blend

Parallel Blend

This is the simplest form of a blend. We will create the part shown in Figures 47 and 48. This blend has three sections as seen in Figure 33 created using separate sketched curves on three permanent datum planes, but make datums would do just as well. This construction method has the advantage of making it easier to visualize the final blend shape.

Start up a new part called **blend1** using the default template.

Create a couple of datum planes using the **Plane** tool. For the first, click on the FRONT datum. The default is an **Offset** datum, so just enter a value of **400**. Select **OK** or middle click.

With DTM1 the active feature (green highlight), you can create DTM2 with two mouse clicks:

Plane

then middle click (since **Offset** and **400** are defaults).

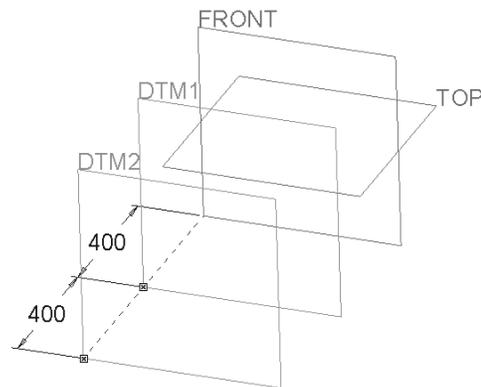


Figure 49 Datum planes to define section sketches

Now create our three sketched curves that will define the blend sections. Select **Sketch** and pick FRONT as the sketching plane. The default RIGHT reference is fine. The first sketch is shown in Figure 4: . The sketch is symmetrical about RIGHT and the lower edge is along TOP. Notice that the sketch has four vertices. This is important, since each section in the blend must have the same number of vertices.

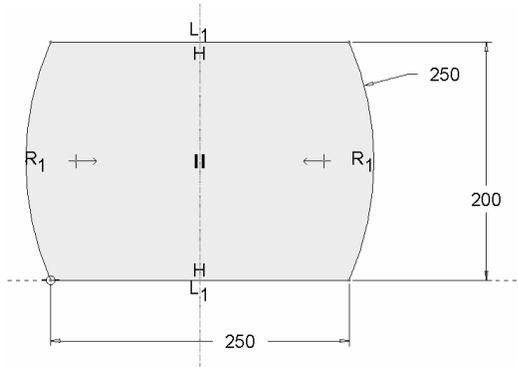


Figure 4: First sketch for blend (on FRONT)

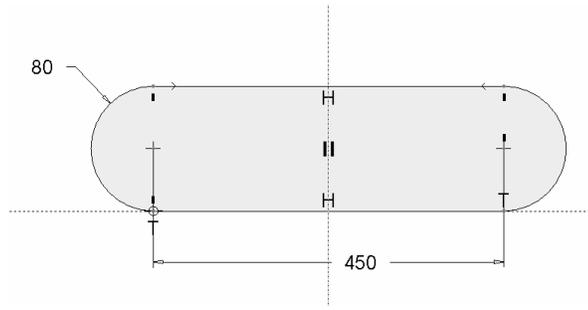


Figure 4; Second sketch for blend (on DTM1)

The second sketch (Figure 4;) is created on the second datum plane DTM1. It is also symmetric about RIGHT, and contains four vertices. Note the lower edge is on TOP.

Finally, create the third section sketch on DTM2. This sketch is shown in Figure 52. Again, it has only four vertices. When complete, your model should look like Figure 33.

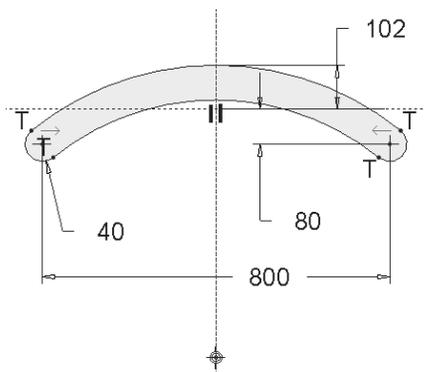


Figure 52 Third sketch for blend (on DTM2)

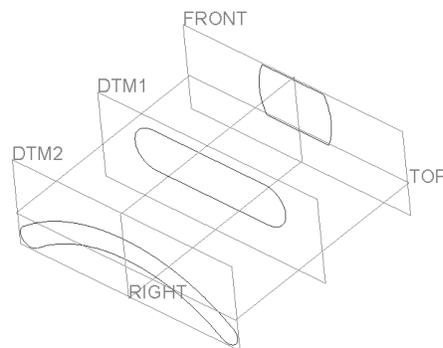


Figure 53 Three curves to define blend sections

Now we can launch the Blend command. In the **Model** ribbon, **Shapes** group drop-down, select

Blend

This opens the **Blend** dashboard. The four buttons on the dashboard are (starting from the left):

-  ***Solid Blend***
-  ***Surface Blend***
-  ***Blend with Sketched Sections*** - which means you create the sections using sketches inside the blend feature, and
-  ***Blend with Selected Sections*** - use previously created sketches to define the sections

Select the fourth button now. Pick on the sketched curve on the FRONT datum. It will highlight in bold green and a small white dot with a magenta arrow will appear on one of the vertices. This is the *Start Point* of the section. It is important that the start points of each section are chosen correctly, as we will see in a moment. Click on the start point and drag it to the top left vertex, as shown in Figure 54. We want the direction arrows on each section to go around the sketch in the same direction - in this case clockwise. If it is pointing the other way, just click on it (or use the RMB menu and select *Flip Arrow Direction*).

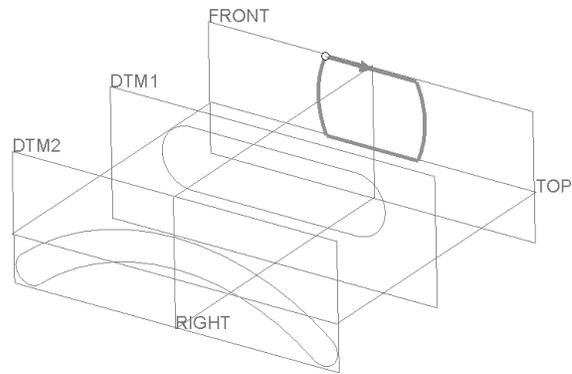


Figure 54 Blend section #1 selected. Note the position of the Start Point.

With the mouse in the graphics window, open the RMB pop-up menu and select *Insert*. Now click on the next sketch (on DTM1). It will highlight in bold green, and the blend between the two sections will preview. Once again, a white dot will appear on one of the vertices of the sketch. We need the start point to again be on the top left vertex in the section, as shown in Figure 55, to align the sequence of vertices in the second section with those in the first. If the vertex is not at the proper vertex, the blend will be warped - just drag the white dot to the desired vertex, and click on the magenta arrow to make sure the sequence direction is correct.

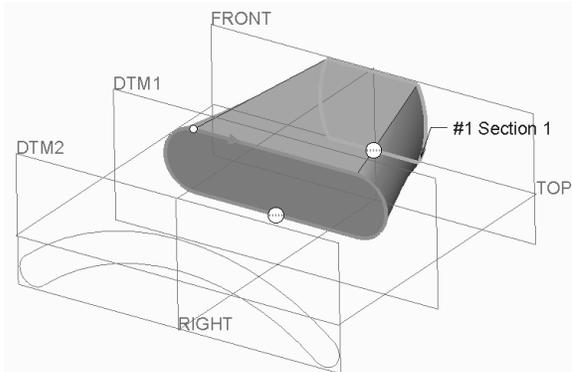


Figure 55 Blend section #2 selected

Once again open the RMB pop-up menu and select *Insert*. Click on the third sketch (on DTM2). The sketch highlights in green. As before, if the start point is not on the right vertex we will get a twisted shape (Figure 56). If so, drag the start point to the correct vertex, as in Figure 57.

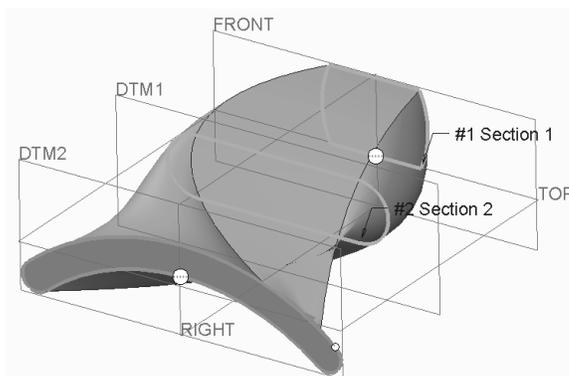


Figure 56 A warped section caused by misplaced start point

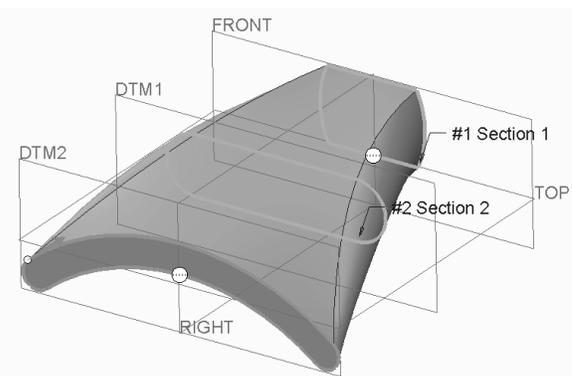


Figure 57 Blend section #3 with correct start point selected

If you accept the feature with a middle click, the blend will appear as in Figure 48 above. In the model tree, note the icon used for a blend feature. Select the feature and in the RMB select **Edit Definition**. As usual, this brings us back to the feature dashboard.

Open the **Sections** tab in the dashboard. This lists the three sections (which highlight on the model if you select one in the list) and the number of vertices in each section. In the **Options** tab, you can select between a **Smooth** blend, or a **Straight** blend (Figure 47). The **Tangency** tab gives access to controls for how the blend will merge with adjacent geometry at each end of the blend³. Finally, the **Properties** tab is to specify a non-default name for the feature.

You might like to try to **Edit** the dimensions of the blend. These are driven by the three sketched curves and the datum planes. Selecting these and using the **Edit** command in the RMB pop-up will let you change blend dimensions. Or, double click on the blend (??)

The *Shell* Command

Just for fun, here is a feature creation command we haven't mentioned before. Pre-select the blend feature (edges highlighted in green) and in the **Model** ribbon, pick the **Shell** tool . This defaults to hollowing out the highlighted object (shelling it) using a default thickness for the remaining shell wall (see the dashboard). A flip button allows you to create the shell on the inside or outside of the existing model surfaces.

In addition to shelling the model, we want to remove the two end surfaces. Open the **References** slide-up panel. This contains two panes. Click in the **Removed Surfaces** pane and pick on the front and back surfaces (using CTRL) as shown in Figure 58. These will highlight in green as they are picked. The interior faces are shown in preview orange. Change the shell thickness to **10**. The other pane in the **References** panel allows us to set different thicknesses for different regions of the shell. Accept the shell feature. The part looks like Figure 59. Open the model tree and observe the icon used to indicate the shell.

³ Move the insert point before the blend, and create two extrudes using the first and third sketches (extrude from sketch #1 going back, sketch #2 coming forward). Resume the blend and experiment with the tangency option to the extrudes coming off each end of the blend.

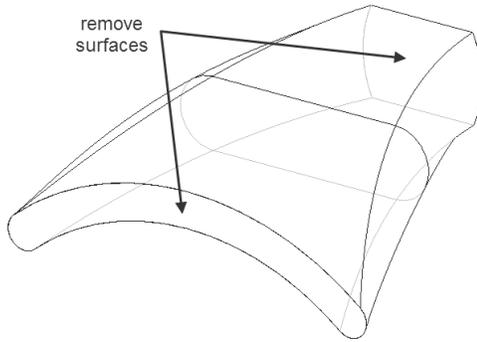


Figure 58 Surfaces to be removed during Shell creation

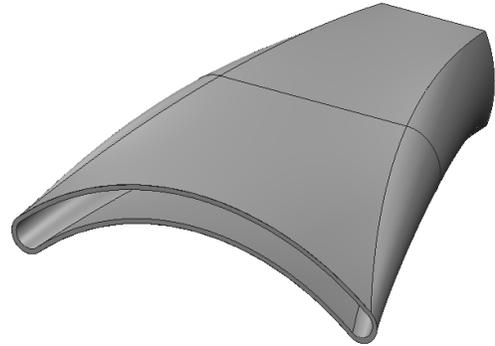


Figure 59 The *Shelled* part

The ***Shell*** command can handle quite complicated geometry, but is not foolproof. You may find that the command will fail sometimes if the geometry is too complicated (typically if a shelled surface has to pass through itself). Normally, the shell command is used fairly early in the model before this situation can develop.

Let's move on to our last new feature in this lesson. Save the current part (for later experimentation) and remove it from session.

Rotational Blend

A rotational blend is set up by specifying the cross sections on a number of sketching planes that have been rotated around a common axis - thus the sections are not parallel. We are going to make the part shown in Figure 5: . Rotational blends can create either protrusions or cuts, and can be either solid or surface features. The usual restrictions apply as to the number of vertices in each section and the start point defined on each section. For some rotational blends, consecutive sections can be no more than 120 degrees apart. The spacing between consecutive sections does not have to be the same. Sections can either be selected from existing sketches (as we will do here), or sketched internally in the feature.

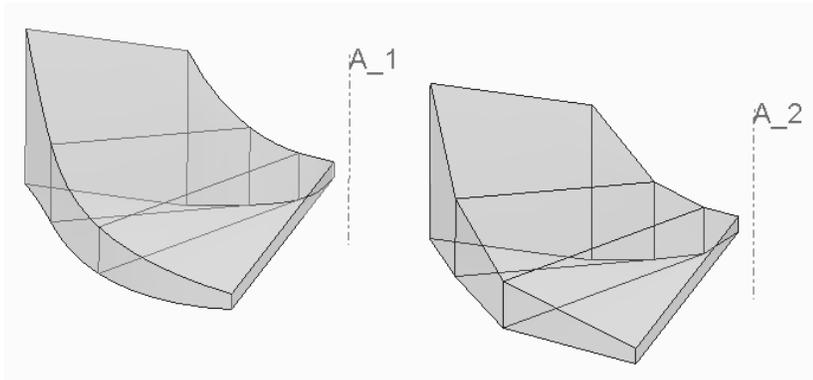
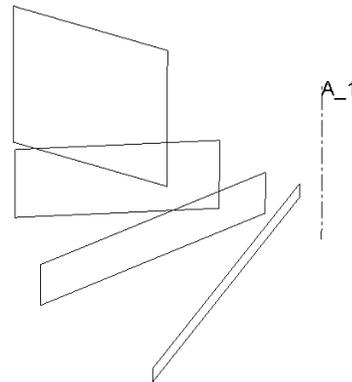


Figure 5: Two rotational blends: smooth (left), and straight (right)

Start a new part called **blend2** using the default template. You can delete the default coordinate system. It will be handy to turn on the display of the datum plane and axis tags.

We are going to create the four rectangular sketches shown at the right. The first sketch will be on FRONT. The next three will be on datum planes created using angle offsets to the previous sketch plane. These datum planes share a common axis.



Create the axis A_1 by selecting (with CTRL) the RIGHT and FRONT datum planes, then the **Axis** tool in the **Datum** group.

Figure 5; Completed section sketches for rotational blend

Create DTM1 by selecting the axis A_1 and FRONT, then **Plane**. Set an offset of **30**. This will be counterclockwise looking down from the top - you may have to enter -30 to get the angle you want, with the positive side of the new datum facing more-or-less to the front.

Create DTM2 by selecting the axis A_1 and DTM1, again with an offset of **30**. Finally, create DTM3 by selecting the axis A_1 and DTM2, with an offset of **30**. This will line up with the RIGHT datum, which you might as well **Hide** now to avoid confusion.

We have set up the section planes this way so that we can easily adjust the angle between successive sections. If you use a parameter and some relations, you can set these all equal, or make them some function of other geometry in the part.

Create the four sketched curves shown in Figures 62 through 65 on the datum planes indicated. In each sketch, the horizontal reference is TOP, and the second sketching reference is the axis A_1. If the screen gets a little cluttered after the first couple of sketches, go ahead and hide them. You can imagine that using references to existing geometry in the new section sketches could complicate matters here if we want to make a very robust feature.

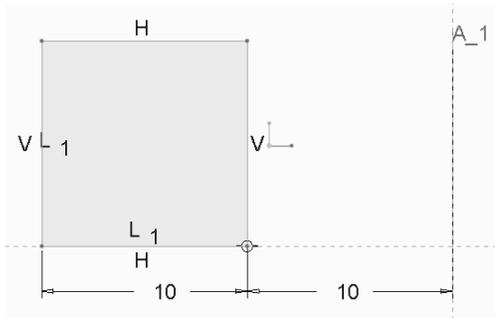


Figure 62 Section 1 on FRONT

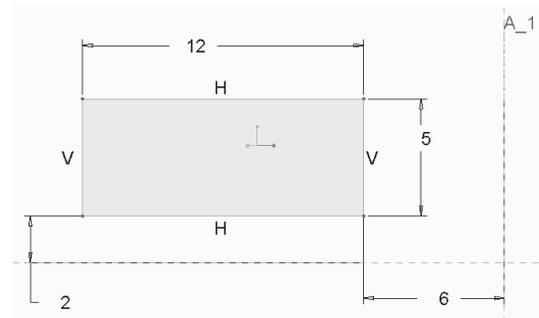


Figure 63 Section 2 on DTM1

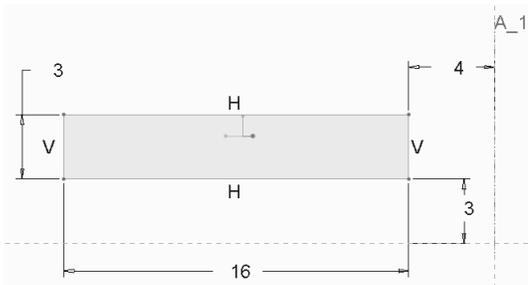


Figure 64 Section 3 on DTM2

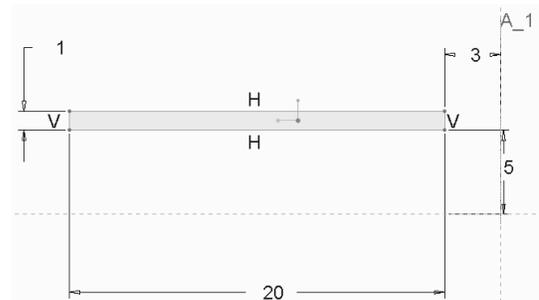


Figure 65 Section 4 on DTM3

Launch the **Rotational Blend** command in the **Shapes** group overflow menu. The dashboard opens, looking very similar to the dashboard for the parallel blend we did earlier. There is one new button for specifying the rotation axis. Pick the button for **Selected Sections** (fourth from left).

Select the first sketch (on FRONT). Observe the start point, and drag it to the top left corner if necessary, as shown in Figure 66.

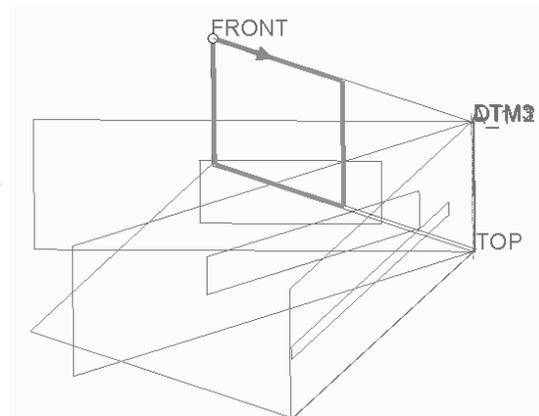


Figure 66 First section selected

Now open the RMB pop-up menu and select **Insert**. Pick on the second sketch. The sketch will highlight and the blend will preview as in Figure 67. If the start point is on the wrong vertex, the blend will be twisted.

Continue using the RMB pop-up menu and *Insert* to select the third and fourth blend sections, making sure the start points all line up. The final previewed feature is shown in Figure 68.

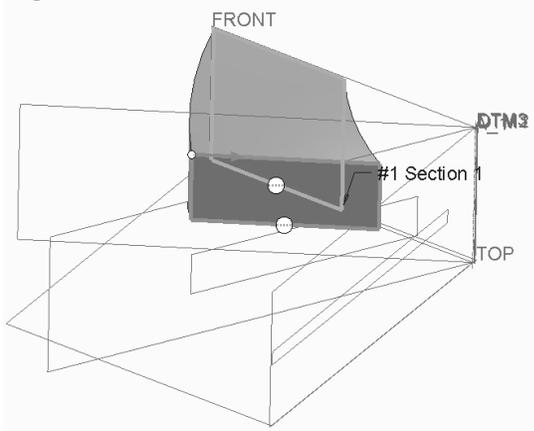


Figure 67 Second section selected

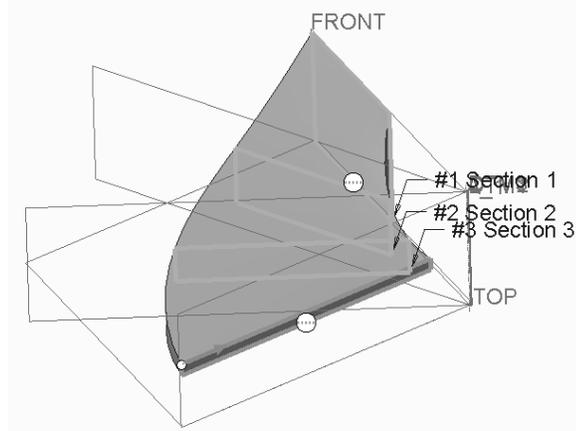


Figure 68 Final section selected

Notice that the rotation blend axis was chosen automatically in this case. For some options of the command you must select that yourself, in which case you can use any axis or straight edge. Of course, you only do this once, since the same axis is used for all sections. If you open the **Sections** tab, you will see the same options we saw before (radio buttons for **Sketched** or **Selected** sections, listed sections, number of vertices, etc.). In the **Options** tab, you can select either *Smooth* or *Straight* blends. There is an additional option here for connecting the end and start sections of the rotational blend⁴. This creates a closed feature. The **Tangency** options operate the same as the parallel blend (note that these are available with the RMB pop-up menu when the large white circles are selected at the start and end sections), and the **Properties** tab allows you to specify a name for the feature.

Accept the feature and close the dashboard. Experiment with the dimensions of the feature (the datum plane offsets, sketch dimensions, etc.). The variety of geometry you can create is quite remarkable, as shown in Figure 69.

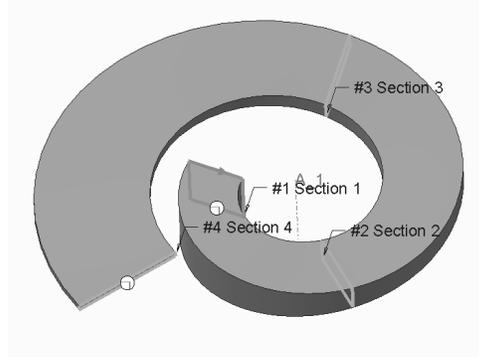


Figure 69 Sections at 140° offset

That completes our limited presentation of blends. As you can see, blends contain a lot of geometric information and are therefore a bit more difficult to set up. However, they offer considerable flexibility and can create very complex shapes not attainable with the simpler features. There are advanced features (swept blends and helical blends, for example) that offer even more complexity/flexibility. Consult the on-line help for information about these.

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.....E q p v l p w g ' y k j ' \$ E q p e n w k q p \$ ' q p ' r c i g ' 33/440'

⁴ This may not work for all possible blends. For example, in this case the ends cannot be connected if you select a **Straight** blend.