deskartes

Design Expert

Industrial Design Software
For Windows

First Edition

David Cheshire: “Introduction To CAID Using Design Expert”.

July 2003.

Copyright

© 2003 D CHESHIRE. All rights reserved. No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language in any form by any means without the written permission of the Author.

The Author reserves the right to revise this publication and to make changes from time to time without the obligation to notify any person of such revisions and changes.

Trade Marks

The DESKARTES name and the logo are trademarks of DESKARTES Oy. Other brand and product names are trademarks and registered trademarks of their respective owners.

Credits

Designed and written by David Cheshire.

Contact Address

DESKARTES OY

Helsinki,

Finland

http://www.deskartes.com

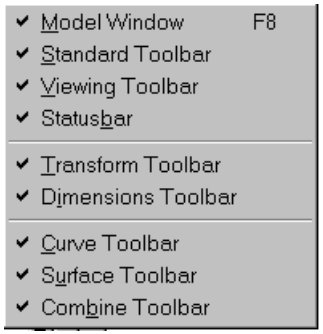
Chapter 1 – Basic Concepts **4**
Icons 4
Model Window 4
Transformations 5
Chapter 2 – First Modeling Exercise **8**
The Body 9
Curve Expert - Row 1 Buttons 10
Curve Expert - Row 2&3 Buttons 11
Curve Expert - Row 4&5 Buttons 11
Curve Expert - Row 4&5 Buttons 12
Curve Expert - Row 6&7 Buttons 13
Curve Expert - Row 8 Buttons 13
Curve Expert - Row 8 Buttons 14
The Body – Continued 15
The Band 15
The Cap and Top 17
The Clip 17
The Nib 21
Chapter 3 - More Complex Modeling **22**
The Base 23
The Body 24
The Lid 25
The Handle 25
The Spout 28
The Finger Grips 30
Chapter 4 – Rendering **32**
Define Materials 33
Advanced Material Properties 34
Define Scene 35
Chapter 5 – Advanced Modeling **36**
Bristles 42
Logo 43
More Cutting and Blending 44
Chapter 6 – Advanced Rendering **47**
Define Lights 48
Scene Edit 49
Define Texture 50
3D Building 51

Chapter 1 – Basic Concepts

Before starting to work with the software it would be good to discuss some basic concepts and methods.

Icons

As with most windows based software the main user interface you will use to drive the software are the icons at the top of the screen. You can choose which icons appear on the screen with the command ***View:Toolbars*** which displays the following submenu…



As you can see the ticks next to each option indicate that all icons are currently displayed.

Most icons simply activate a command when pressed. Other icons set Design Expert to operate in a particular mode, which is indicated by the icon staying depressed after clicking.

As an example the buttons in the view toolbar are grouped according to their function, within each group one button stays depressed when it is clicked until another in the group is pressed.

The current mode of operation is set. The four choices are Pick Geometry. Rotate View, Pan View, or interactive Zoom.

The current viewing direction is set. The four choices are the three orthogonal views along an axis or an isometric view direction.

The display mode is set. The three options are wireframe, shaded or both.

Model Window

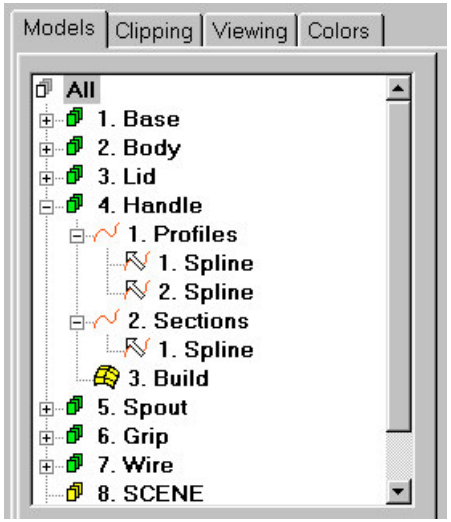
Perhaps the most important item on the screen is the Model Window – in particular the Models tab in this window. This shows a breakdown of the current model structure.

Each model is made up of different parts. Parts have a name to help you distinguish them. You can change the default name by clicking on the name and typing a new name.

Next to each part name is a plus sign (+) indicating there is information below the part. Parts can hold information about Profile, Silhouette, Section or General curves. These are known as Curve Sets since they can store a

group of related curves. The part also contains one or more surfaces.

Each of the curve sets can hold several actual curves.



By clicking on the names in the window you determine the item that subsequent commands are applied to. For example clicking on a curve and issuing the ***Edit:Delete*** command will delete the curve. Click on a curve set and issuing the same command will delete the curve set and all the curves in it. Of course, ***Edit:Delete*** with the part selected will delete the part and everything under it.

Transformations

An important part of modeling in any CAD system is the ability to move individual parts of the design into position. Design Expert allows you to move parts in this way but since you will design most of your parts in the correct position – a valuable feature of Design Expert – you may not use this feature as much as you expect. The following example shows how to use these features.

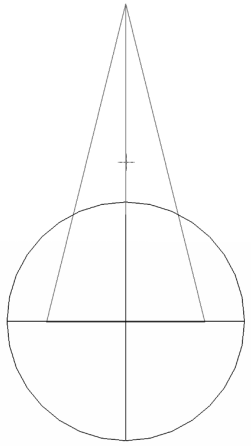
We will design a small earring using the standard 'building blocks' that Design Expert provides.

Starting with the sun shape at the bottom of the earring. Create the first element of the design the central globe. Use ***Surface:Primitive***, choose the Sphere tab and type a radius of 4. The surface primitive will be created and to hold it a part will also be created. They will both be

called Sphere. To avoid confusion click on the part name in the models tab and type in a new name 'Earring'.

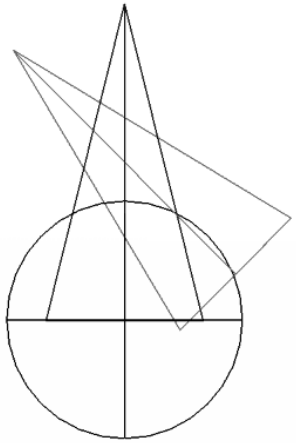
Now create a new surface this time choosing the Cone tab. The required dimensions for this are height=8 and radius=2. The other buttons on the tab choose the direction of the cone, which should be set to +Z.

The cone is created but it is not in the correct position it can be moved now until it is in the correct position. We do this graphically so that the move is visually correct. It is much easier to do graphical transformations when looking from an orthographic view point so set the view to the X axis now with ***View:Move View Point:Ortho View***. Choose ***Transform:Move:Interactive,*** click and hold the left mouse button with the cursor near the cone. Move the cone upwards until the base of the cone is level with the center of the sphere.



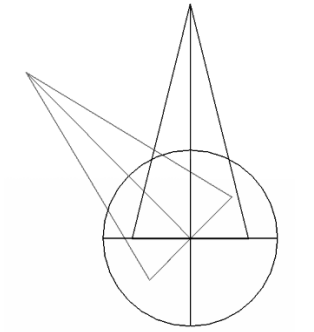
You may see a small red cross drawn at the center of the cone this is called the fixed point. If not you can display it with ***Transform:Fix point:Show.*** It is important in certain transformation operations particularly rotate and scale. You can think of it as the center of rotation or scaling.

The rays of the sun are repeats of this cone. The easiest way to create these rays is by copying the first one. Use ***Transform:Duplicate*** to make and identical copy of the cone (since it is a copy it is created exactly on top of the original). The copy can now be rotated into its new position. Choose ***Transform:Rotate:Numeric*** and type 45 and choose X as the axis.

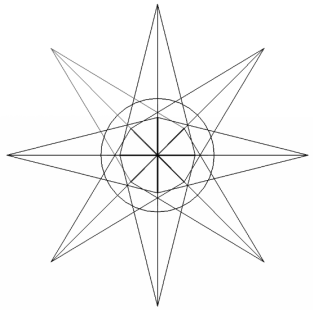


Unfortunately the rotation you have applied is incorrect because the center of rotation (the fix point) is incorrectly positioned. So undo the last command (***Edit:Undo Rotation***) to put the cone back. To correctly position the fix point use the command ***Transform:Fix***

Point:Numeric and in the dialog type 0 into the x, y and z boxes. Choose ***Transform:Rotate:Numeric*** and type 45 and choose X as the axis an now the rotation should occur correctly.

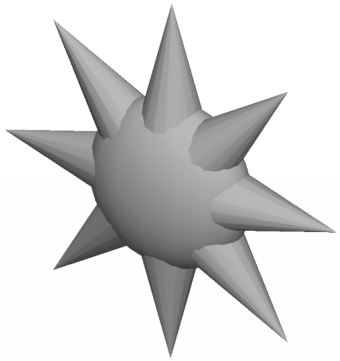


To complete the suns rays a series of ***Duplicate*** followed by ***Rotation*** commands are required. This is easily achieved by repeating a series of ***Transform:Duplicate*** followed by ***Transform:Repeat*** commands. You can make this even easier by noting the shortcut keys that are assigned to the commands - <control>D and <control>R.



To complete the sun shape the central sphere should be 'squashed' to flatten it into a disc. Select the sphere in the model tab and choose ***Transform:Scale:Numeric*** typing 0.6 in the X box and 1 in Y and Z. If necessary, set Uniform scaling mode off.

Change the view direction to isometric and turn on shading .



To complete the design, first set the view to X and then create a radius 2 sphere moved and duplicated to taste. Then create a torus with inner radius 2 and outer radius of 4 in X direction, move it into position then perform two duplicate and move operations. The center torus will also need to be rotated by 90 around Z.

Finally, two cylinders are required. The first has a radius of 3 and height of 5 in direction X. The second a radius of 0.5 and height of 15.

On completion of the model ***File:Save*** it to disk as you will need it later on in the tutorial.

Chapter 2 – First Modeling Exercise

Now its time to start creating shapes from our own imagination – not just combing existing shapes. The following two modeling exercises create very different objects but it will become apparent that the techniques used for both are very similar.

Like all modeling exercises the fist task is to break down the product into simple parts that can be modeled independently. Look at the model now and decide how you would break down the pen into manageable parts.

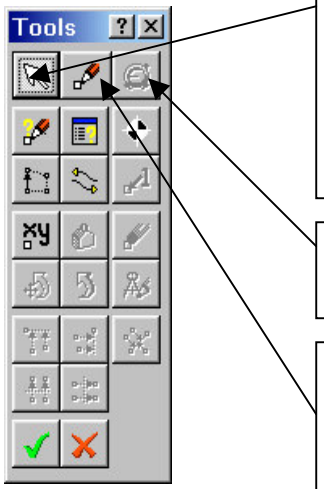
The Body

The body of the pen seems the most logical place to start. It is circular so the obvious method of creating this shape is by rotation. This means you need to draw a profile curve representing the shape of (half) of the body when viewed from the side.

Start by creating a new model (***File:New***) if you have another model open. The window should go blank and the model tab should be cleared. You are now ready to start the modeling process.

A new curve can be created using ***Curve:Create:Free Curve***. In the subsequent dialog box choose the direction as X-Axis and ensure the curve type is Profile. Note that a new part will automatically be created for you since no parts exist yet. After clicking OK Curve Expert mode will be entered and you will be able to draw the shape required. Since this is the first time you have met Curve Expert it is worth taking a few moments to learn the functions that can be used to define the curve shapes you want.

Curve Expert - Row 1 Buttons

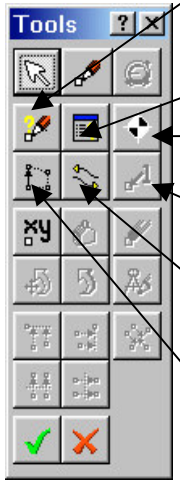


Sets the editing mode to point selection. This is indicated by an arrow cursor. In this mode points can be selected. Multiple Selection can be achieved by the SHIFT key. Once selected these points can be moved, scaled (CONTROL key) or rotated (SHIFT key). The right mouse button reverts to edit mode.

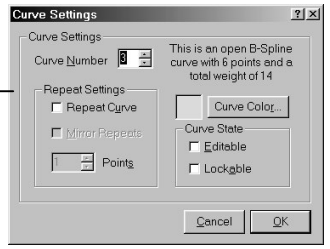
Undo the previous operation. A single level of undo is available within curve editor.

This button sets the editing mode to drawing. This is indicated by the crosshair cursor. In this mode points can be added by pointing and clicking. To add points to the start or end of a curve first click near the first or last point. To add points along the curve click on the dotted control polygon. The right mouse button reverts to selection mode.

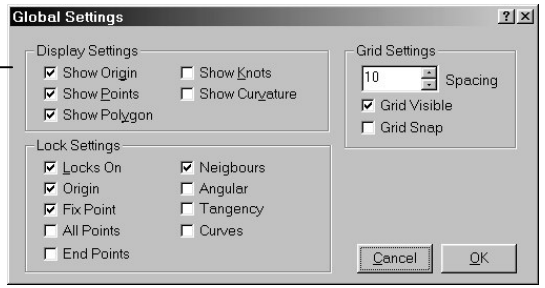
Curve Expert - Row 2&3 Buttons



The settings that define the appearance and operation of a curve can be altered using the dialogue box created by this command. These include curve color, curve editability and repeat curve settings.



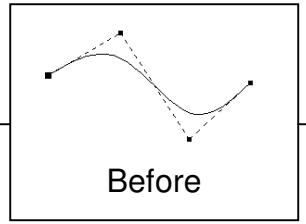
The settings that define the general operation and appearance of the curve editor can be altered using the dialogue box created by this command. These include the display of curve polygons etc. and the operation of locks. The size of the snap grid

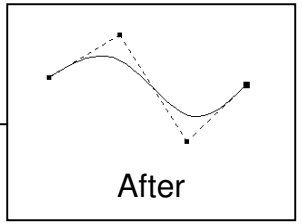


Creates a dialogue box which can be used to alter the position of the fixed point marker . Alternatively the marker can also be dragged visually on the screen.

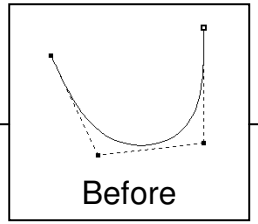
Sets the start point of a closed curve to the last selected control point of the curve. This is particularly useful when defining cross-section curves since if the start points are not aligned the surface may appear twisted along its length.

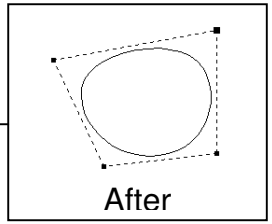
Reverses the direction of the current curve. This means that the first point will become the last point and the last point will become the first etc. The direction of curves used to create a surface should be consistent to ensure the surface is not twisted.



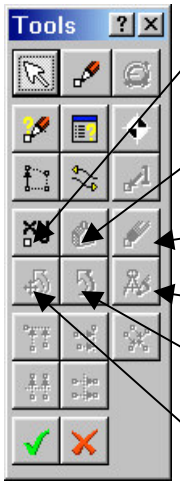


Opens a closed curve and closes an open curve. An open curve is one where the start and end of the curve are distinct. With a closed curve the start and end are coincident and the curve forms a loop.

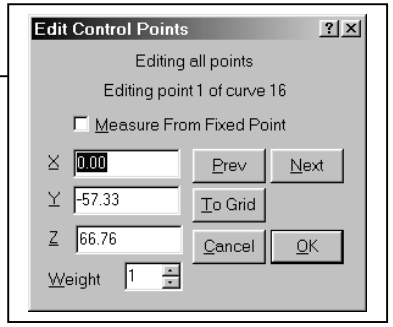




Curve Expert - Row 4&5 Buttons



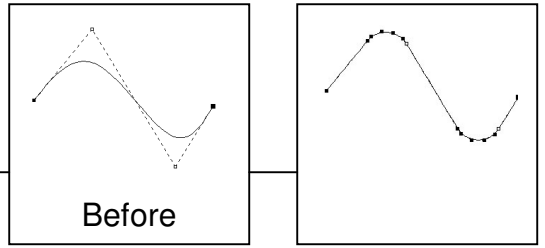
Creates a dialogue box which can be used to alter the position and weight of the selected control points by typed in values. If no points are selected the position of any editable control point can be altered.



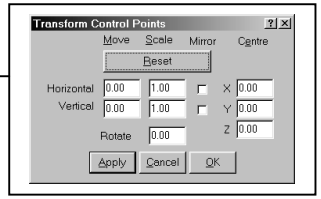
Creates a dialogue box which allows the weight of the selected control points to be set. Most points normally have a weight of 1. Increasing the weight of the point to 2 will have the effect of pulling the curve more closely to the control point. Increasing the weight of the point to 3 will have the effect of pulling the curve to pass directly through the control point forming a sharp corner. A weight of 3 is automatically assigned to the first and last points of an open curve so that the curve passes through these points.

The selected control points are deleted. If a point is deleted by mistake this command can be undone.

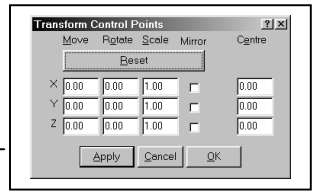
The dialogue box created by this command can be used to add fillet radii or round corner of a specified radius to the selected control points. The number of extra control points added to the curve to form the radii can also be entered.



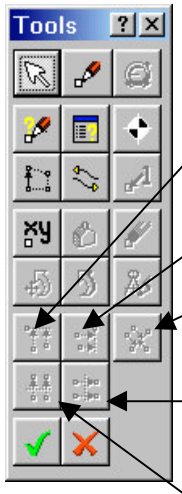
The dialogue box created by this command can be used to move, scale, rotate and mirror the selected control points by typed in values. These commands are optimized for use with 3D curves.



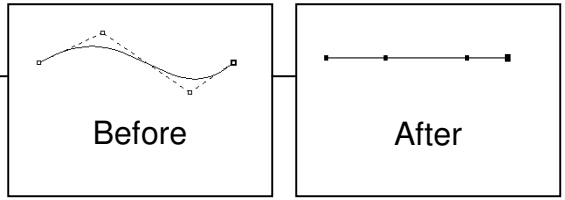
The dialogue box created by this command can be used to move, scale, rotate and mirror the selected control points by typed in values. These commands are optimized for use with planar 2D curves as all transformations occur relative to the screen.



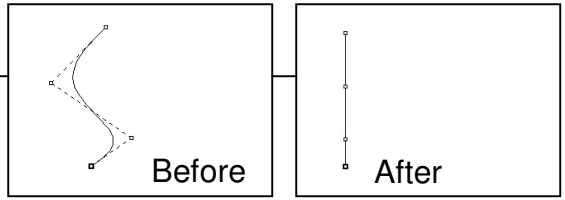
Curve Expert - Row 6&7 Buttons



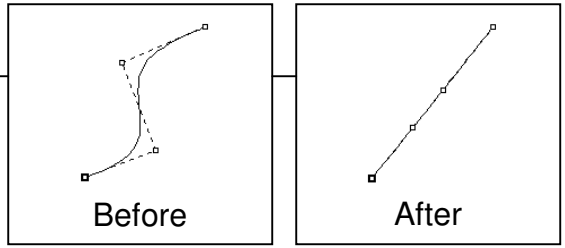
Used to align horizontally the selected control points. After pressing the button the cursor changes to a question mark at which point you should point and click near a selected control point. All selected control points are then aligned horizontally with this point.



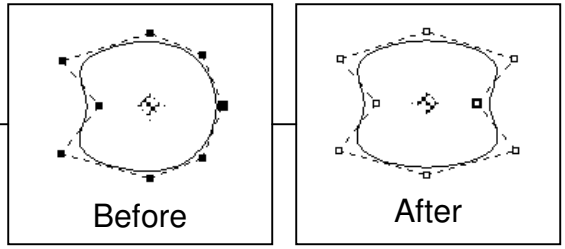
Used to align vertically the selected control points. After pressing the button the cursor changes to a question mark at which point you point and click near a selected control point. All selected control points are then aligned vertically with this point.



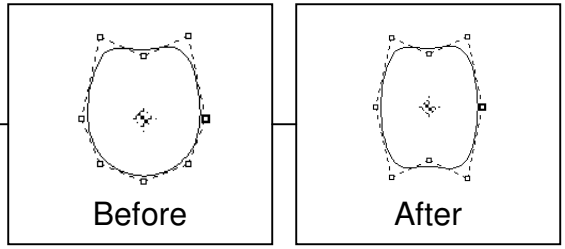
Used to align the selected control points. The control points are moved to a straight line defined by the first & last selected control point.



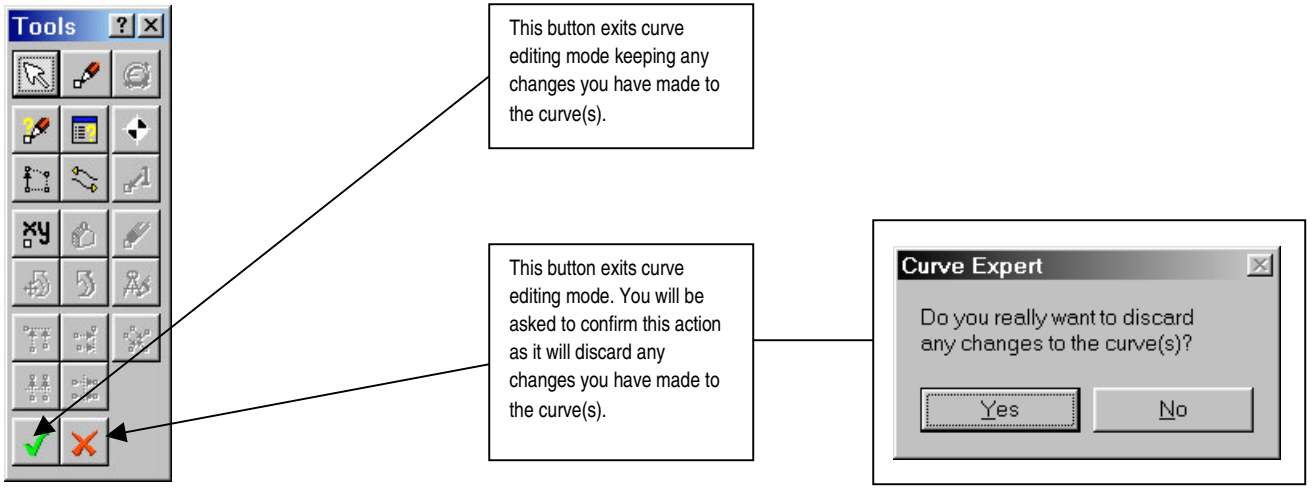
Used to mirror the selected control points about the vertical center line through the fixpoint. After pressing the button the cursor changes to a question mark at which point you point and click near a selected control point. This point becomes the target point. The control points on the opposite side of the center line to the target are moved to a position vertically opposite the nearest point on the target side. In other words the control points on the target side do not move.



Used to mirror the selected control points about the horizontal center line through the fixpoint After pressing the button the cursor changes to a question mark at which point you point and click near a selected control point. This point becomes the target point. The control points on the opposite side of the center line to the target are moved to a position horizontally opposite the nearest point on the target side. In other words the control points on the target side do not move.

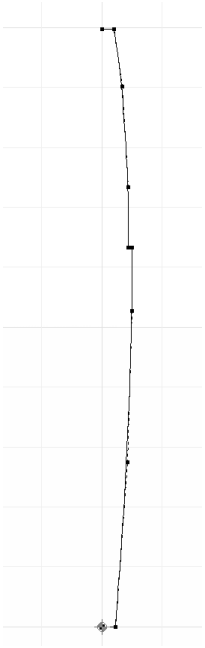


Curve Expert - Row 8 Buttons



The Body – Continued

Now you know how to use the functions in Curve Expert you can continue to design the pen using each of the functions as and when required. Start drawing from the origin marker, clicking on each point in turn. The first point must be vertically in line with the origin marker as this will be the point of rotation for the surface. If the curve does not start at this point the surface may end up with a hole in the middle of it. Use a 10mm grid to aid you in judging the size of the pen (about 100 long).



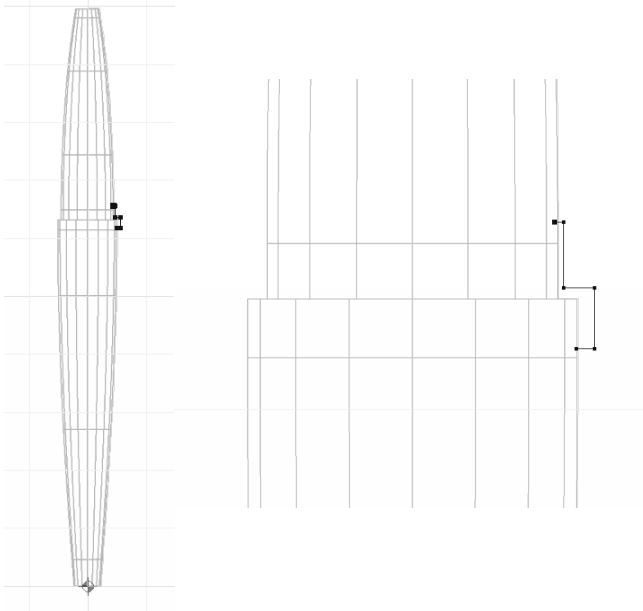
When you are happy with the shape of the curve click on the icon to leave Curve Expert. To create a surface from this curve via rotation choose ***Surface:Rotate*** and click on OK to accept the options in the dialog box.

A new surface is created.

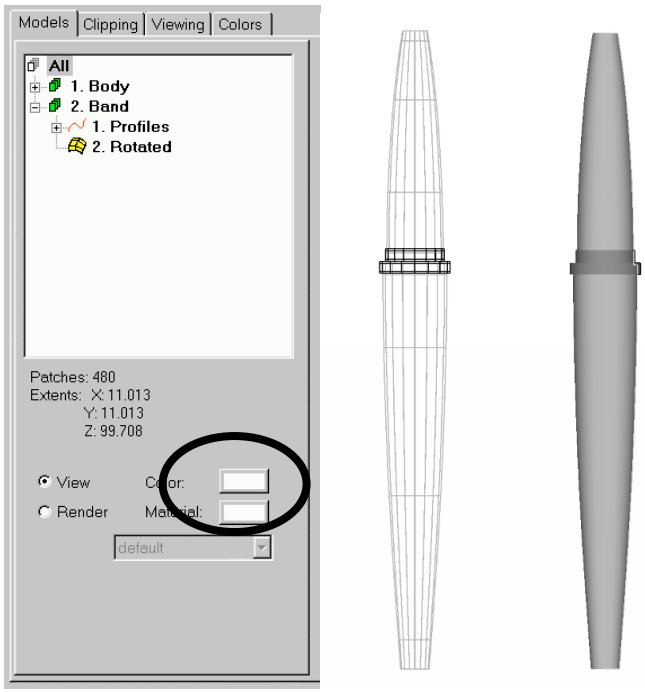
The Band

There is a metal band around the center of the pen. It is aesthetic feature. Also in the finished pen the body would be made in two halves and this band would join the two halves. This is an example of how in the early stages designs are simplified and only when the final designs are required is the full detail added.

The band is again circular so it is made in the same way as the body. First a new curve must be created. Since the shape of this curve means it can be made from a series of straight lines it is best to create a polygon using ***Curve:Create:Polygon***. In the subsequent dialog box choose the direction as X-Axis and ensure the curve type is Profile and New Part is ticked. You will see the screen change and notice that the other objects on the screen (the body surface and the body curve) remain on the screen. This is most important as it allows the band to be designed in the correct position relative to the body. Click on the points to define this curve now (six points).

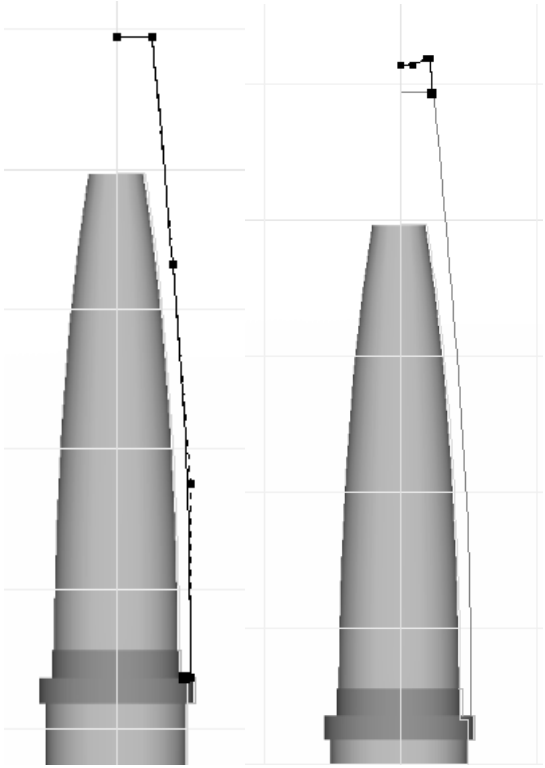


When you are happy with the shape of the curve click on the icon to leave Curve Expert. To create a surface from this curve via rotation choose ***Surface:Rotate*** and click on OK to accept the options in the dialog box. You can change the type of display by choosing one of the icons and by clicking on the color box in the model tab you can choose a different color for each part. These are colors to differentiate parts during modeling they do not represent the final colors or the model – that comes later,



The Cap and Top

The next two parts are made in an identical way to the body so here is a chance to show you can remember what you have learnt. Use the curves below as a guide to make **two** separate parts.

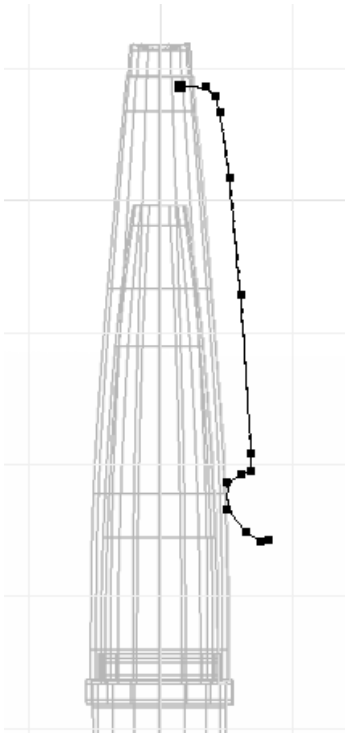


Your pen should now look like this…

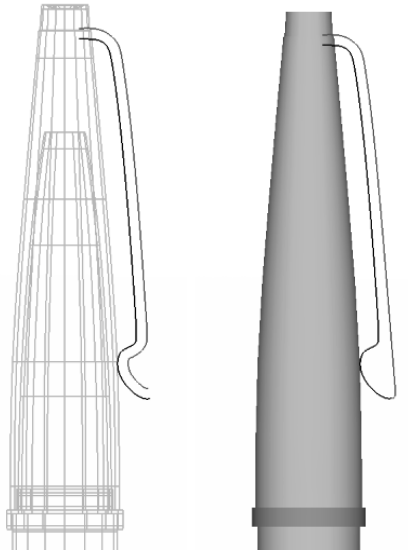
The Clip

The next part is the clip. It is clearly not circular so it cannot be made by rotation. We will use the most powerful and flexible shape creation method called building. The build command can accept a variety of curves as its input. The shape of the final surface depends on the number of curves you provide – in general you need to draw two curves to represent the opposite sides of the clip.

The first stage is to create a new curve using ***Curve:Create:Free Curve***. In the subsequent dialog box choose the direction as X-Axis and ensure the curve type is Profile. The new part option should be ticked. After clicking OK Curve Expert mode will be entered and you will be able to draw the shape required.



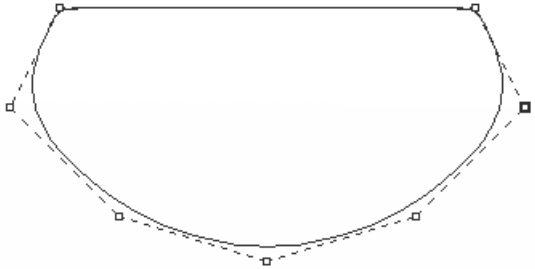
The opposite side of the clip can now be drawn. You could draw this in the same way as the last curve but an easier method is to create a second curve parallel to this a small distance away. Ensure the curve (Spline) is highlighted in the Models tab then choose ***Curve:Create:Offset***. You will be asked to type in an offset distance – if your model is to scale a value of 2 should be OK. You should see arrows drawn on the curve. If these are pointing to the outside of the clip then all is well as – if not choose the Opposite option for the offset direction. Click OK to create the offset curve. Look at the Models tab and you should see that below Profiles there are two Splines. You can edit the curves now to adjust their shape. It is important to remember that if you select the word Profiles in the model tab before choosing the command ***Curve:Edit*** then you will be able to edit both curves together.



When editing the curves you must remember an important rule. To use the build command with two curves they must contain an equal number of points. This means that if you add or delete a point in one curve you must do the same to the other.

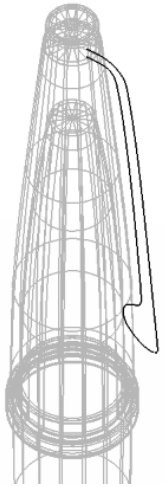
Also the curves at the end of clip must meet exactly otherwise you will end up with a hole in the clip.

There is one more piece of information required before we are ready to make the surface. We need to draw a third curve to describe the cross section of the clip. The shape of this curve means that the best starting point for the curve is to draw a circle made with ***Curve:Create:Circle***. Once again choose the direction as X-Axis but this time ensure the curve type is Section. Actually the choice of X-Axis does not matter it just makes it consistent with all the other curves. A suitable radius for the clip is 2 and the number of points should be 8 then click on OK. You can now use ***Curve:Edit*** to change the shape of this circle to a more interesting shape.

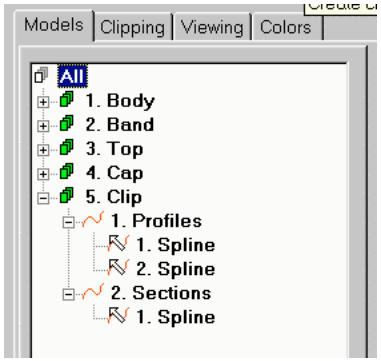


This shape was created by first deleting the top most point. The neighbors of this point were then moved and weighted to a weight of 2.

After exiting from the curve editor you are ready to create the surface. To enable you to see the surface as it is being created change the viewing direction to one similar to that shown below.

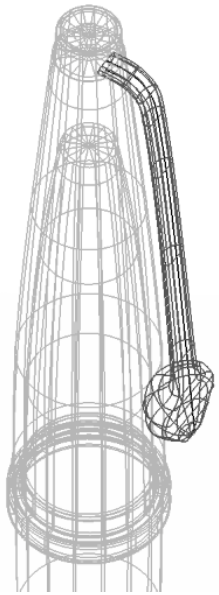


You are now ready to create the clip but before going any further check the data you have created is correct. The expanded Models tab should look like this…



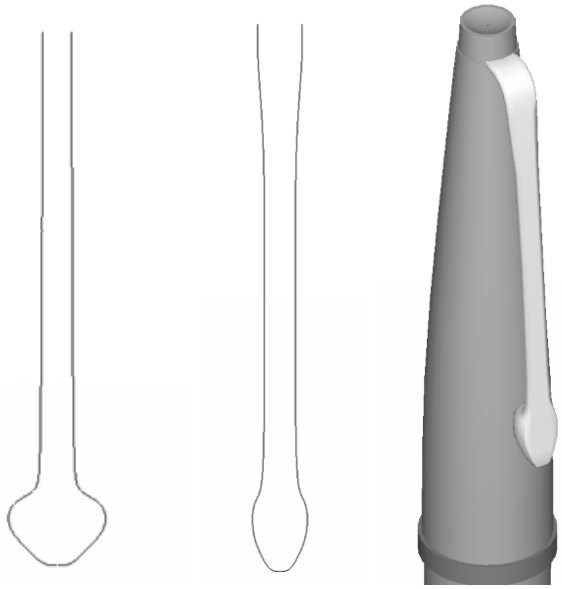
Now you can create the surface by choosing ***Surface:Build***. However you have the chance to change the orientation of the section on the surface. The flat

should be on the outside of the clip. If it is not, click on the rotation option then press Next>> until the orientation is correct.



The clip has been created but the end is rather bulbous. We need to modify the design a little. The build command offers a further refinement. You can create a second set of profiles viewed from another direction (known as silhouettes) that can be edited to adjust the shape.

Choose ***Surface:Create Silhouette*** and choose the silhouette direction as Y. Two curves are now created which describe and control the shape of the surface when viewed from the front.



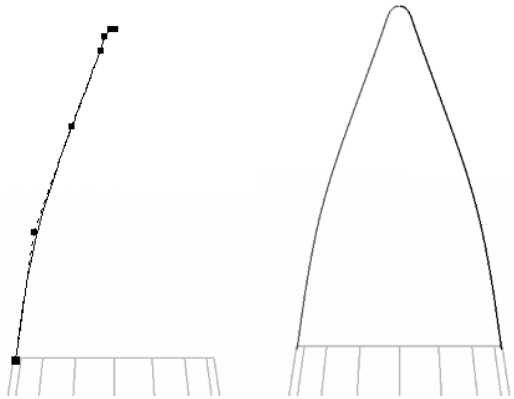
These curves can be edited to narrow the clip at the end (remember you can edit the two curves together by first clicking on Silhouettes in the Models tab).

After you have edited these curves the command ***Surface:Apply Silhouette*** should be used to affect the surface.

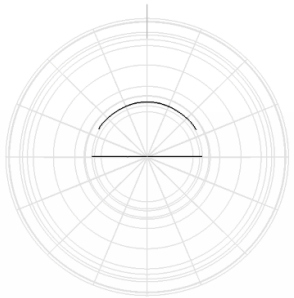
The Nib

The final part may look complicated but it is actually made exactly like the clip.

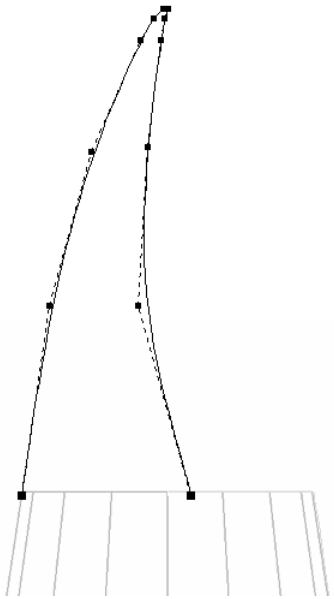
Draw this curve as a Y projection curve. Next ***Transform:Duplicate*** to make a copy of the curve then ***Transform:Mirror:Interactive*** to mirror the copy.

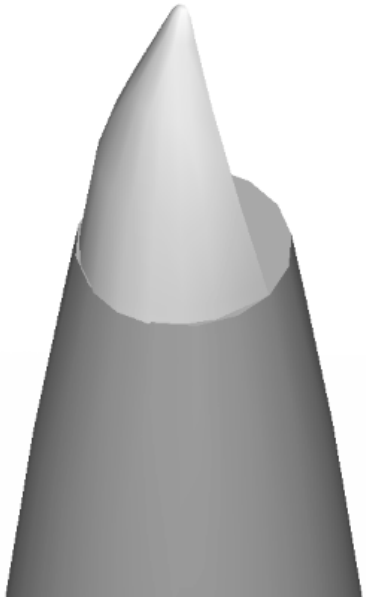


The section for this shape is an arc ***Curve:Create:Arc*** drawn from Z with 6 points, radius 2, start 30 and end 150.



Build the surface and create silhouettes from X - ***Surface:Create Silhouette.*** Edit the curves by moving the points as shown below then ***Surface:Apply Silhouette.***





Chapter 3 - More Complex Modeling

In this section we will introduce some of the more complex modeling techniques. This will allow you make more interesting models. In fact the techniques you learn here will form the foundation of all the modeling you undertake with Design Expert now and in the future.

The model chosen to illustrate the techniques is shown below. It is a modern kettle. The model you will create is a slightly simplified version of an actual product – after all you are not an expert modeler yet! Many of the models you will create as a designer will be simplified, since many of your designs will be sketch models which will never reach production. Sketch models are models that visually appear correct but lack details that are unseen. This is a sensible approach as time wasted on details that will never be seen means less time spent on design variation and iterations. When the design appearance has been finalized you can return to add the fine detail required for manufacture.

Many of the parts in this model are clearly going to be modeled separately. The body and the spout although both chrome material need to be made independently because there shapes are so different.

On the other hand the main body of the kettle and the base are both circular objects so you may be considering making them as a single surface but since they are to be made of different materials it makes more sense to separate them.

The handle is clearly another surface but what about the lid? The details of the indentations for the grips mean that it is impossible for the lid to be made from a single surface. You will see how many surfaces are actually used later.

The Base

The base seems the most logical place to start. The base is circular so the obvious method of creating this shape is by rotation. You may remember from earlier examples this means you need to draw a profile curve representing the shape of (half) of the base when viewed from the side.

You may be confident enough to attempt this without any further explanation. If you are start now referring to the next diagram for the shape of the curve that is required.

So you need some more guidance! OK start by creating a new model (***File:New***) if you have another model open. The window should go blank and the model list should be cleared. You are now ready to start the modeling process.

A new curve can be created. Since the shape of this curve means it can be made from a series of straight lines it is best to create a polygon using ***Curve:Create:Polygon***. In the subsequent dialog box choose the direction as X-Axis and ensure the curve type is Profile. Note that a new part will automatically be created for you since no parts exist yet. After clicking OK Curve Expert mode will be entered and you will be able to draw the shape required. Start drawing from the origin marker, clicking on each point in turn – a total of seven points. The first point must be vertically in line with the origin marker as this will be the point of rotation for the surface. If the curve does not start at this point the surface may end up with a hole in the middle of it.

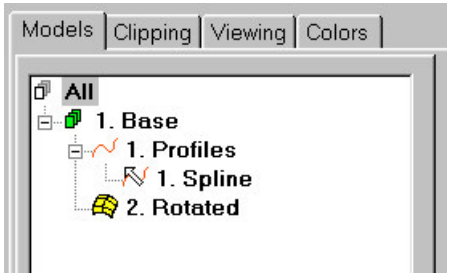
If your curve is a smooth curve and not a polygon you chose the wrong option to create the curve. Exit the curve editor and try again.

When you are happy with the shape of the curve click on the icon to leave Curve Expert. To create a surface

from this curve via rotation choose ***Surface:Rotate*** and click on OK to accept the options in the dialog box.

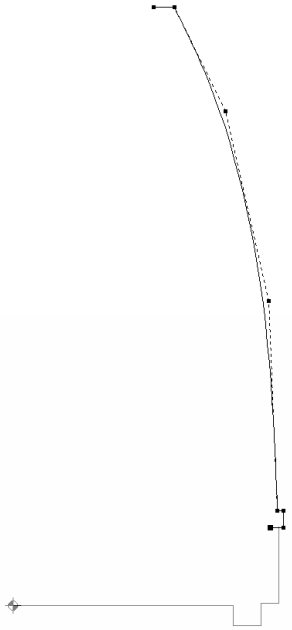
A new surface is created. As we said at the outset this is a simplification – none of the inside detail has been added as at this stage of the design it will not be seen.

It is always good practice to use meaningful names for each part in a model since as the number of parts increases it can be difficult to remember what each part was. Change the name of **Part1** now by clicking on the name in the Models tab and type in a new name such as **Base**. The models list should now look as follows.



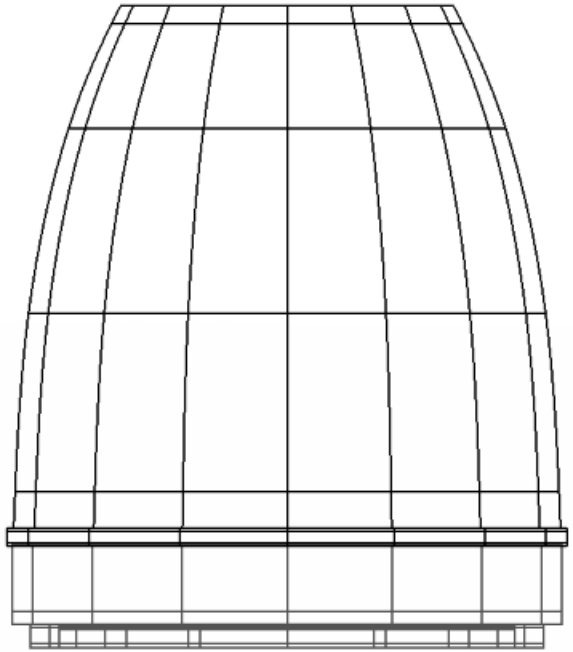
The Body

The Body should be created in an identical fashion to the Base. A new curve can be created. The smooth shape of this curve means it you should choose the option ***Curve:Create:Free Curve***. Once again choose the direction as X-Axis and ensure the curve type is Profile. Check the option to create a new part. After clicking OK Curve Expert mode will be entered and you will be able to draw the shape required. The curve for the base will automatically be shown in a non-editable fashion so that you have a guide. Draw the curve for the body ensuring that the curve crosses the base curve. You may need to zoom out by right clicking on the zoom icon to give you more room to draw.



When you are happy with the shape of the curve click on the icon to leave Curve Expert. To create a surface

from this curve via rotation choose ***Surface:Rotate*** and click on OK to accept the options in the dialog box.

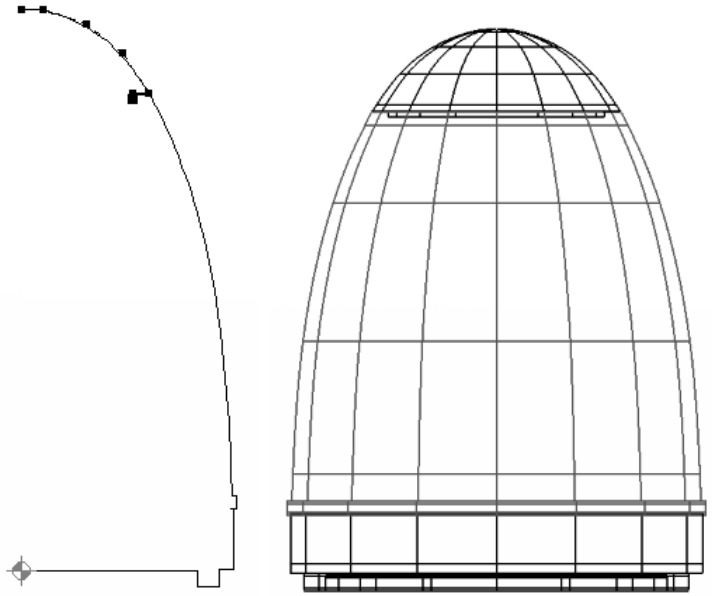


A new surface is created of the simplified body. Change the name of this part in the in the Models tab to **Body**.

The Lid

The basic shape of the lid (without the grip indentations) is made in the same way as the Base and the Body.

Follow the instructions for the Body but drawing the curve below to make the lid surface.



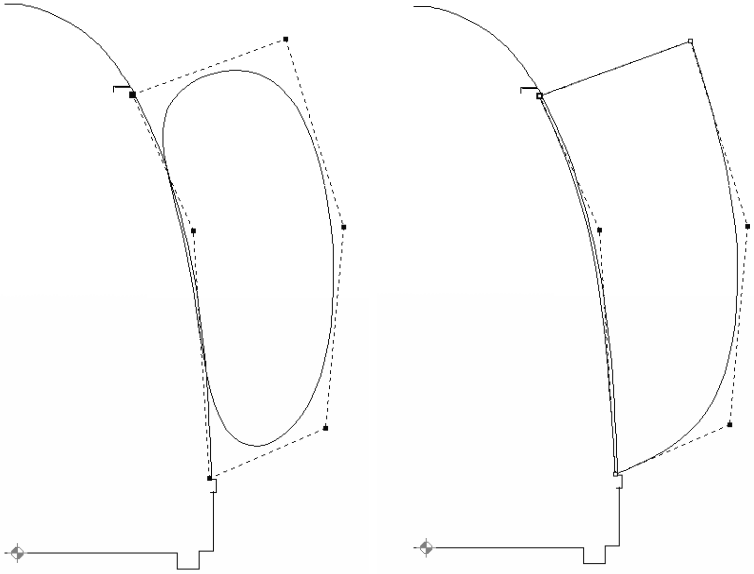
The model is now taking shape. The lid is not yet complete but we will return to complete it later. It is time to start producing a more complex shape.

The Handle

Clearly the handle cannot be made by a rotational surface. A more complex method is required. The most appropriate method for this shape is to build the surface. Here is how…

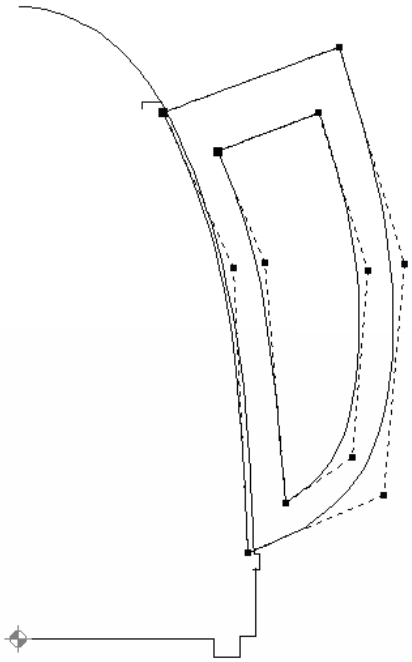
A new curve can be created. The shape of this curve means it you should choose the option ***Curve:Create:Free Curve***. Once again choose the direction as X-Axis and ensure the curve type is Profile. Check the option to create a new part. After clicking OK Curve Expert mode will be entered and you will be able to draw the shape required. The existing curves will automatically be shown in a non-editable fashion so that you have a guide. Draw the first curve for the outside of the handle. This curve is a closed curve (it forms a closed loop). This is achieved by clicking the last point directly on top of the first.

Note how the points on the left follow the shape of the body but are drawn just inside the body. This is important if the handle is to look as though it is attached to the body. To finish the shape the two points at the top of the handle and the point at the bottom need to be made in to sharp corners. This is easily achieved by selecting the points to be modified. Remember that you can select a number of points by holding down the control key during selection. After selection choose the Weight icon. Pick 3 in the dialog box to create sharp corners. The shape of the curve should now be as the right hand picture below.

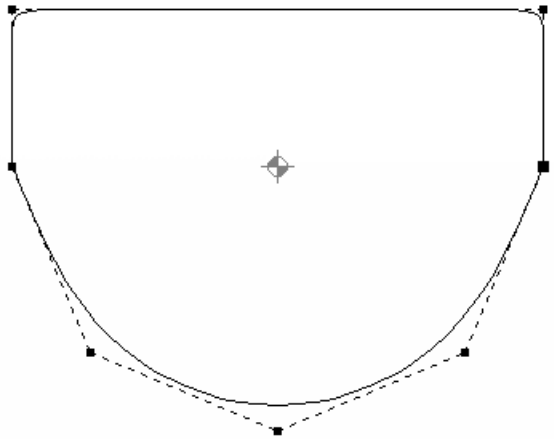


When you are happy with the shape of the curve click on the icon to leave Curve Expert. You are not yet ready to create the surface. Building requires more information than simple rotation surfaces. The next piece of information is a curve representing the inside of the handle. This is easily created by offsetting the first curve. Ensure the curve (Spline) is highlighted in the Models tab then choose ***Curve:Create:Offset***. You will be asked to type in an offset distance – if your model is to scale a value of 15 should be OK. You should see arrows drawn on the curve. If these are pointing to the inside of the handle then all is well as – if not choose the Opposite option for the offset direction. Click OK to create the

offset curve. Look at the Models tab and you should see that below Profiles there are two Splines. You can edit the curves now to adjust their shape (it looks better if you make the top of the handle thicker). It is important to remember that if you select the word Profiles in the model tab before choosing the command ***Curve:Edit*** then you will be able to edit both curves together.

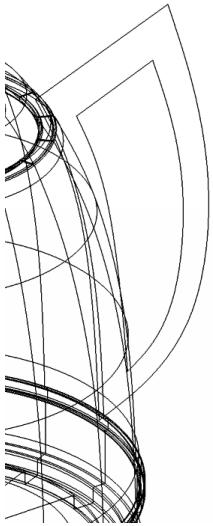


There is one more piece of information required before we are ready to make the surface. We need to draw a third curve to describe the cross section of the handle. The shape of this curve means that the best starting point for the curve is to draw a circle made with ***Curve:Create:Circle***. Once again choose the direction as X-Axis but this time ensure the curve type is Section. Actually the choice of X-Axis does not matter it just makes it consistent with all the other curves. A suitable radius for the handle is 10 and the number of points should be 8 then click on OK. You can now use ***Curve:Edit*** to change the shape of this circle to a more interesting shape.

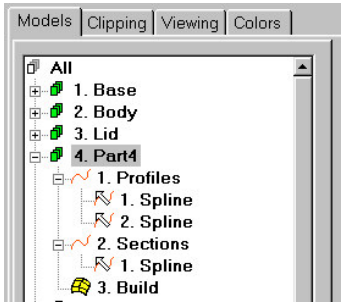


This shape was created by first deleting the top most point. The neighbors of this point were then moved so that they are directly above the centerline points. The top four points were then weighted to a weight of 2.

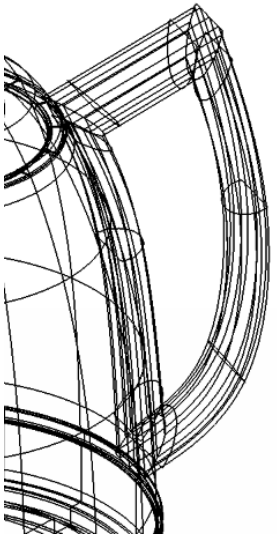
After exiting from the curve editor you are ready to create the surface. To enable you to see the surface as it is being created change the viewing direction to one similar to that shown below.



You are now ready to create the Handle but before going any further check the data you have created is correct. The expanded Models tab should look like this…



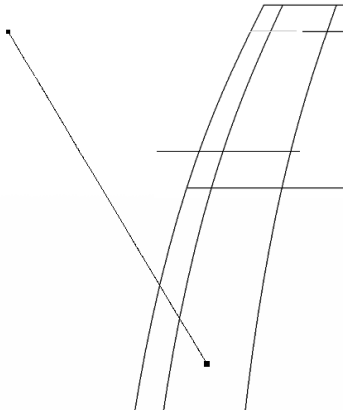
Now you can create the surface by choosing Surface:Build. Choose the one direction option and a surface will be drawn. However the option is offered to change the orientation of the section on the surface. The flat should be on the outside of the handle. If it is not, click on the rotation option and then press Next>> until the orientation is correct.



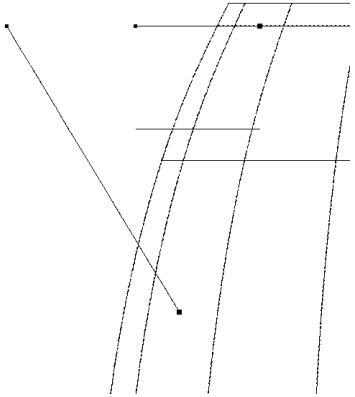
The Spout

You may be surprised to find that the spout is made in an identical fashion to the handle. The only difference is the shape of the curves involved. Follow the handle description along with these brief hints.

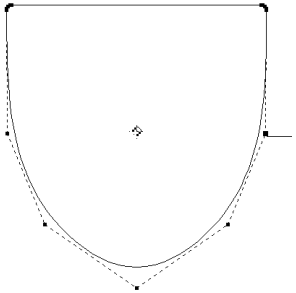
A new curve should be created with the option ***Curve:Create:Free Curve***. Once again choose the direction as X-Axis and ensure the curve type is Profile. Check the option to create a new part. Draw the curve below…



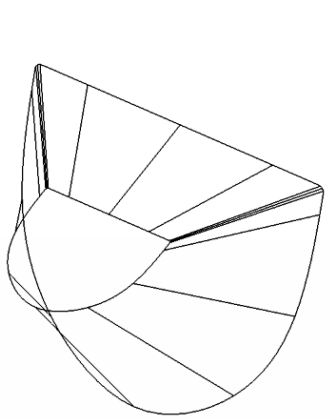
Create a second curve as an offset (***Curve:Create:Offset and Edit***) at a distance of 10mm above the first curve. Edit as below…

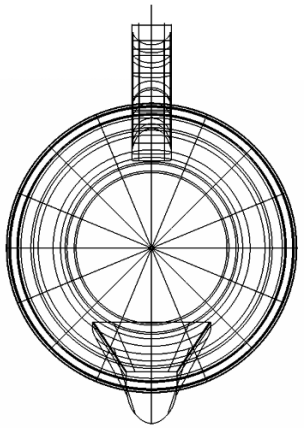


Create an 8 point, 10mm radius circle with the option ***Curve:Create:Circle***. Once again choose the direction as X-Axis but this time ensure the curve type is Section. Use ***Curve:Edit*** to change the shape of this circle. A small radius has been added to the top two corners...



You should now be able to create this surface. Choose the Section Scaling: Two Directions option.





If you change the viewing direction to above (***View:Move Viewpoint:Ortho View: +Z***). You will see

that the spout tapers so that it is very wide where it meets the body. This is easily adjusted. Choose ***Surface:Create Silhouette*** and choose the silhouette direction as Z. Two curves are now created which describe and control the shape of the surface when viewed from above. These curves can be edited to narrow the spout near the body (remember you can edit the two curves together by first clicking on Silhouettes in the Models tab).

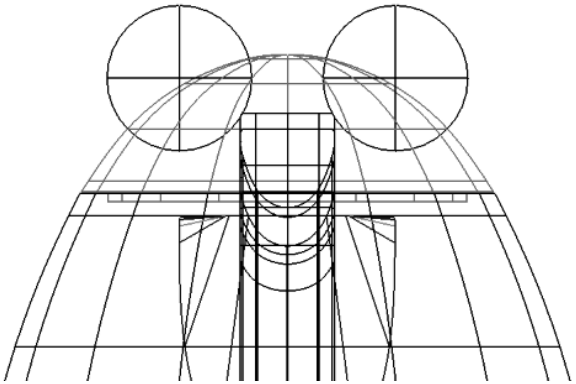
After you have edited these curves the command ***Surface:Apply Silhouette*** should be used to affect the surface.

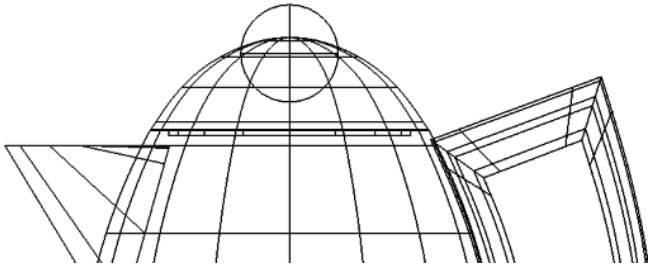
The Finger Grips

Now it is time to return to the lid to add the finger grips. These are cut into the surface we have already created. This is a new concept – using one surface to cut into another – but it is fundamental to the modeling process for all but the simplest models.

The first step is to create the surfaces for the grips. These can be created using any of the surface creation techniques you have already learnt. We will use a simple spherical surface primitive. Create a new part ***Edit:New Part*** then create a ***Surface:Primitive***. Choose a Sphere with a radius of 10.

Now use the Transformation techniques you learnt in Chapter 1 to move the sphere to the correct position. Then use ***Transform:Duplicate*** to create a copy of the correctly positioned sphere. Set the fixed pointback to 0,0,0 using ***Transform:Fix Point:Numeric.*** Finally, mirror the copy of the sphere in the Y direction.





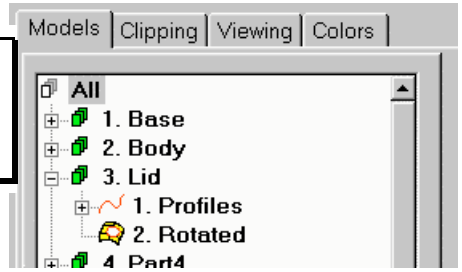
If the Rolling Ball Blend Option is grayed out check you have selected the surface not the part name.

With the grips correctly positioned it is time to make the cuts. To improve the ergonomics of the design we will create a smooth transition between the two surfaces. This is known as a rolling ball blend. Adjust the view position and turn on shading so that you can see the creation process then select the Lid surface.

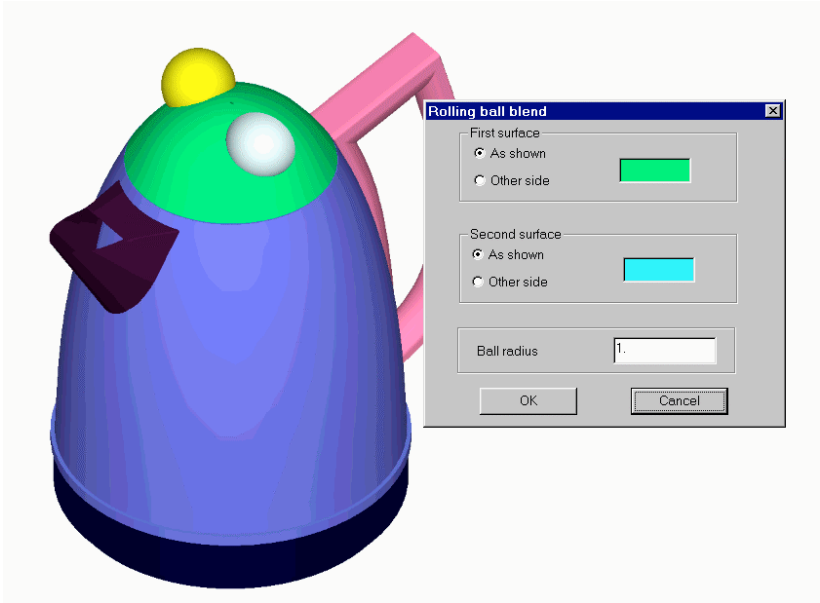
The two grips will need to be cut one at a time. To initiate the process choose ***Combine:Rolling Ball Blend*** then

click on one of the spheres (the picked surface will be cut with the surface selected in the Models tab). The view will now change and part of the sphere and the lid will be cut away. At this point you have to decide whether each of the two surfaces has been cut correctly. They should look like the following picture. If they do not, you can choose the Other Side option for either surface (the surfaces are colored so you can tell them apart). When you are confident that the options are correct type in a radius of 1 and choose OK. Looking in the Models tab you should see that another new surface is created below the sphere called a Blend surface.

Note that the small icon next to the lid surface has changed to indicate that it has been trimmed.



If something goes wrong in the creation of the blend remember that you can always ***Edit:Undo*** the operation and try again.



This series of operations should be repeated for the second sphere. On completion you should note that the lid is actually made up of a total of 5 surfaces (lid, 2xsphere and 2xblend).

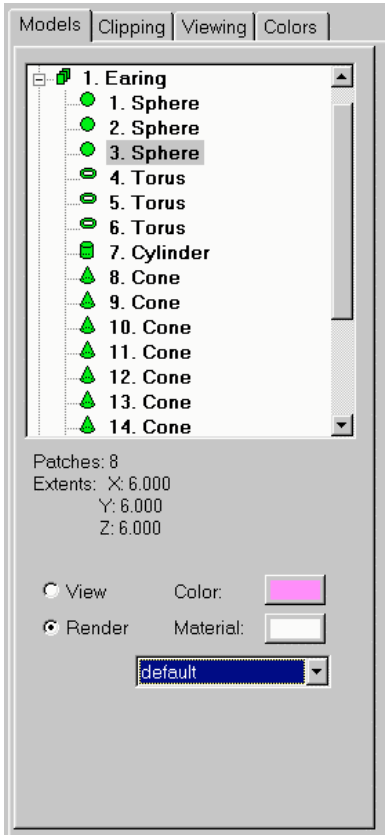
That completes the model. You may like to experiment by adding a blend between the spout and the body. Another combine option is ***Combine:Trim Surface/Surface*** which works in similar way to blend but does not create a smooth transition surface – it leaves a sharp corner. You could try this between the handle and the body but remember it will not look any different unless you view from the inside of the body.

Chapter 4 – Rendering

As you have been working through this manual you have no doubt become accustomed to using the two display options of Wireframe and Shaded. These are the two options that are used for the majority of modeling operations as they are the simplest and therefore quickest drawing methods available in Design Expert.

When it is time to create design presentation graphics an additional option is provided called rendering. Highest level of presentation graphics can be generated with ExTrace ray tracing which is explained in the on-line Help.

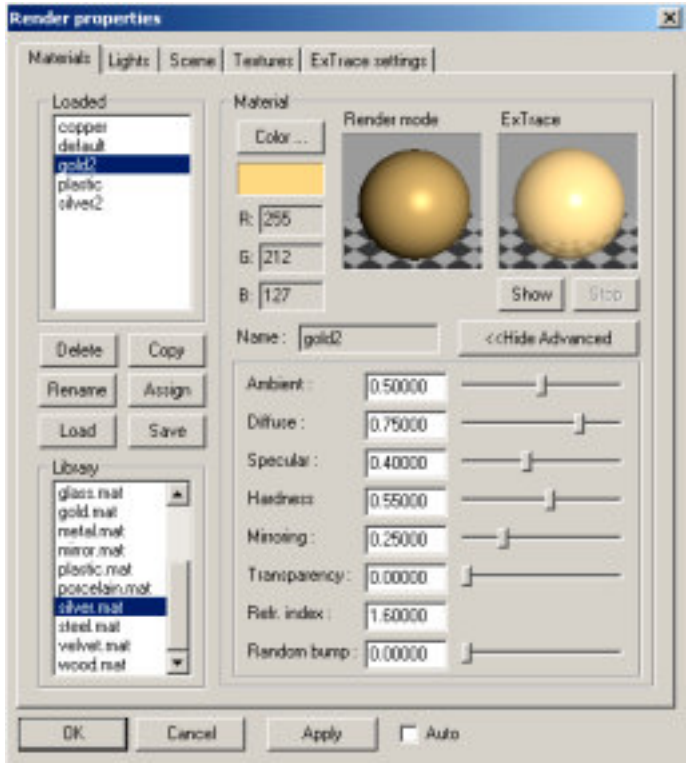
To show a model rendered first load the model (if one is not already loaded). This tutorial will describe the steps involved in creating an improved picture of the earring, which you created in the first chapter so it would be good to load this now. After loading click on the Render option, which can be found at the bottom of the Models tab. The display should change – you are likely to see the background change to black graduating up the screen to gray and the model will be white – the default color.



The controls for altering the appearance of the model can be found in the Render menu, which offers options including – Define Scene and Materials. We will examine both of these options starting with materials.

Define Materials

The ***Render:Define Materials*** option produce a large dialog box with many options to alter the appearance of objects. In this box you will define the visual material properties of different materials (how light interacts with the surface of an object). These materials can then be assigned to different parts of the design.



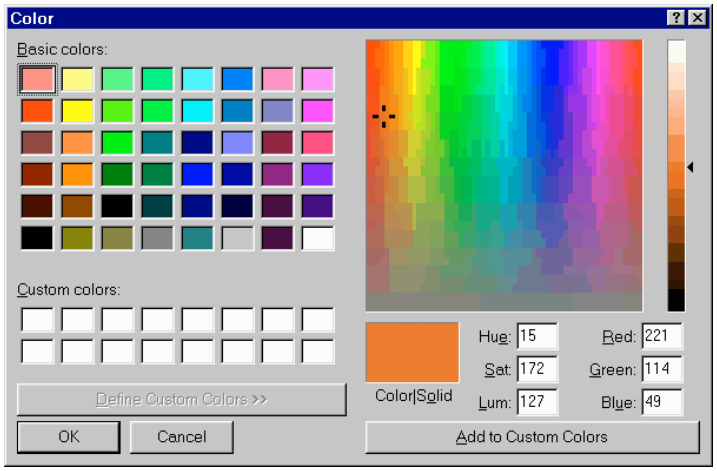
The easiest way to use this dialog is to load in predefined materials from the Design Expert library. At the bottom left of the box you will see an area titled library, which lists the materials available. Double clicking on one of these names loads a copy of the material from the library into the current design file. Try loading in gold and silver now, as we will need these for the earring.

Of course not all the materials you want will be stored here so Design Expert provides the tools to change the appearance of the material in a design file. Say for example we wanted to make the sun part of the earring a more copper color we could create a new material called copper – here is how.

Most new materials can be based on an existing similar material. Copper bears some similarities to gold (if you exclude price!). We will need gold as well so click on the name gold in the 'Loaded' section of the dialog box. Now press the Copy button and a duplicate material called Copy of gold will be created. Click on this name and press Rename and type Copper. We have now created a material called Copper that is currently an exact duplicate of gold.

To change the appearance of Copper click on the Color button and the familiar Windows color chooser box is displayed in which you can select a new color. Find one that looks like copper which is much redder than gold.

ExTrace material preview widow shows slightly different material appearance compared to Render mode window due to different image generation method.



Click OK to return to material editor. Further refinement of the material properties can be made with the 'Advanced' properties which are described later but these can be left alone for now.

Although materials can be associated with a particular object from this box there is an easier way, so for now press OK on the Materials dialog.

Just below the Render option in the models tab you will see a chooser list, which is only active when in Render mode. This chooser lists all of the materials available in this design file. It will probably show Default at the moment. First select an object in the models tab – say one of the cones and then select from this list copper. The cone should change to a copper appearance. You can repeat this with all of the parts in the design – look back to chapter one to see a picture of the earring and try an replicate the appearance of that model.

Advanced Material Properties

Ambient

Ambient light describes a constant brightness for the whole surface. You can think of it as *"day light"* which comes from all directions, independently of the user-defined light points.

Usually, give a material some ambient light: otherwise, light points result in completely black shadows. However, *avoid using it with glass and mirror materials* - they easily become over lit if you add the ambient component.

Diffuse

The diffuse reflectivity value describes the general reflectivity of the surface, *i.e.*, the overall effect of light points. A low value means a dark object, a high value, a light one.

Diffuse reflectivity does not affect highlights. They are controlled separately, with the following two sliders.

Specular

The specular reflectivity factor describes the polish of a surface, *i.e.*, *brightness of highlights*. The greater the value, the brighter the highlights and the shinier the surface appears.

Hardness

Hardness describes the *highlight sharpness* (size) on the surface. For soft materials, use values near to zero, and for hard materials larger values.

With very large hardness values (close to one), the highlights may become so small they don't show on the test ball at all. They will show on a larger picture resolution, though.

Mirroring

Mirroring defines the reflection factor, i.e. how much the object mirrors other objects. Value 1.0 describes a perfect mirror, while smaller values make the material reflect the environment only dimly.

Remember to *use a low ambient value* with mirroring materials.

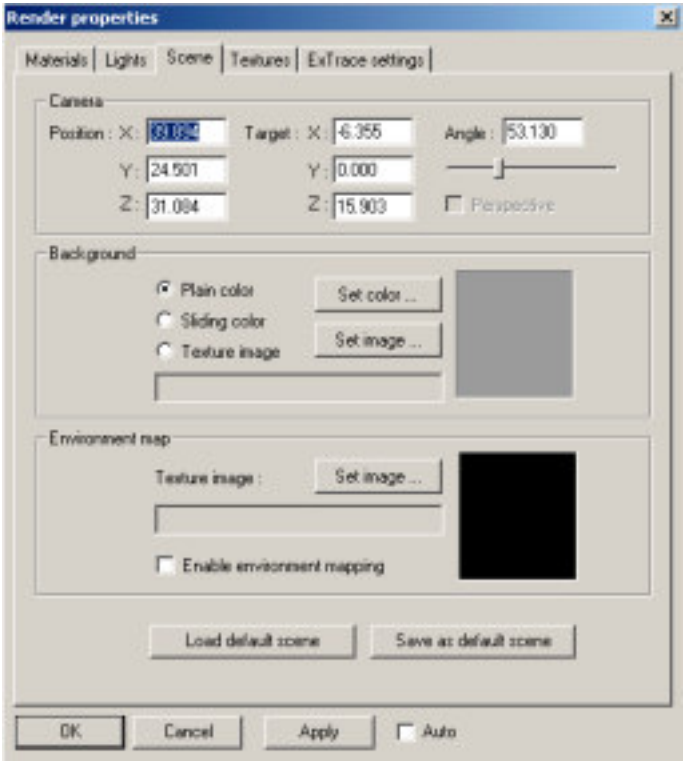
Transparency

The glass behavior is controlled by a property that controls transparency - *how much* the object shows through.

Mirroring does not have much impact on glass, so you should usually set it off. If you have mirroring on, however, use a *low reflection value* – an object cannot simultaneously show through and be a perfect mirror.

Define Scene

You may have noticed when you apply materials such as gold and silver to the design their appearance is not very realistic. These materials are examples of reflective surfaces and when such surfaces have nothing to reflect they can appear 'flat'. The ***Render:Define Scene*** command gives an option to correct this. In the dialog box choose a suitable texture image and tick the enable environment mapping option to see the results. You can also set the background color and type here – a sliding color background can often be effective.

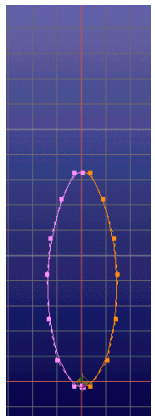
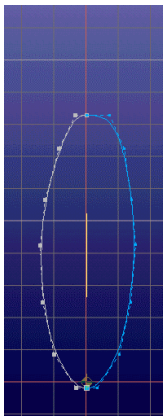
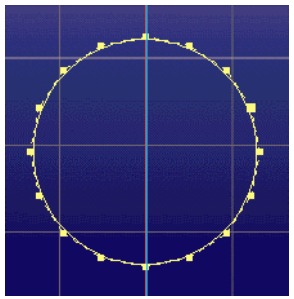
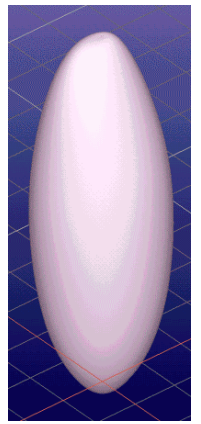
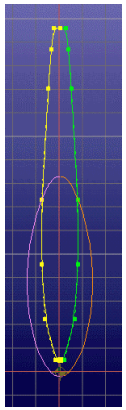
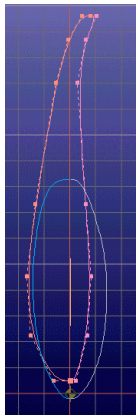
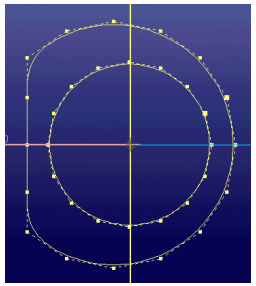
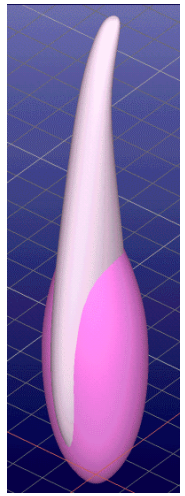


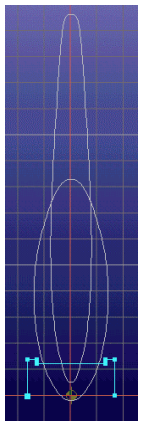
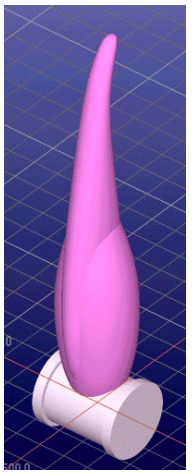
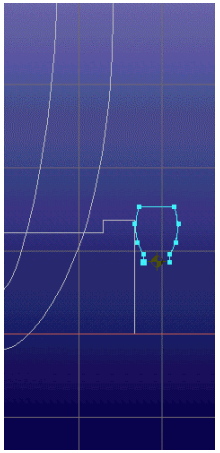
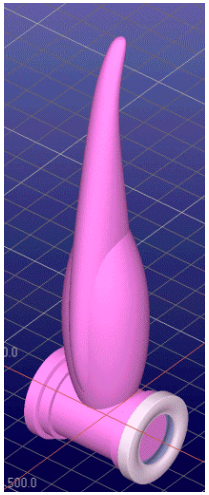
Chapter 5 – Advanced Modeling

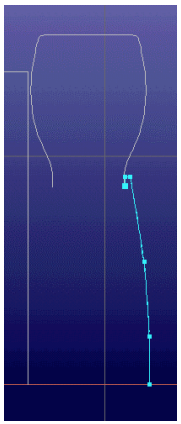
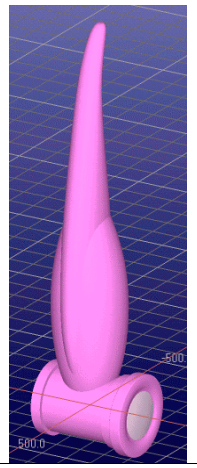
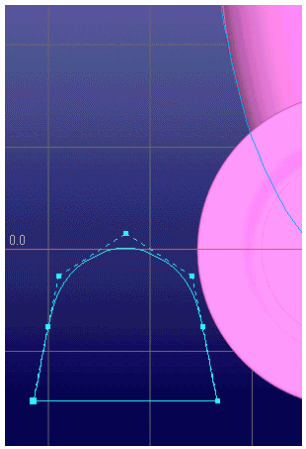
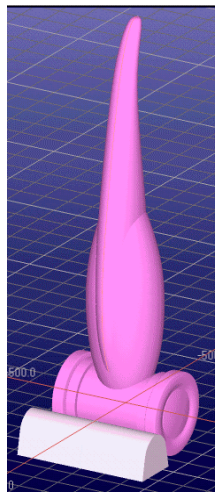
In this section we will return to the process of creating models using the more complex modeling techniques which began to be introduced in Chapter 3 - More Complex Modeling.

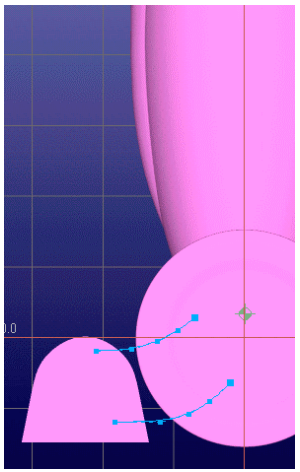
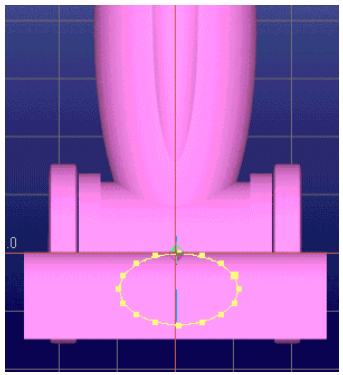
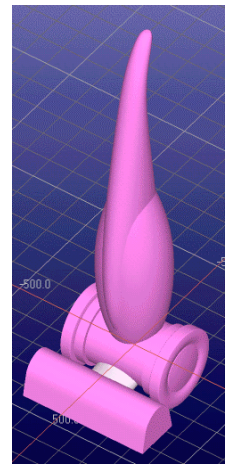
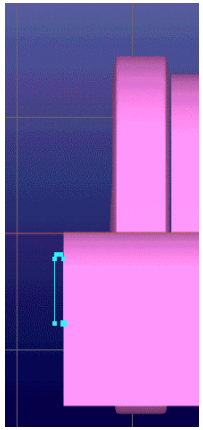
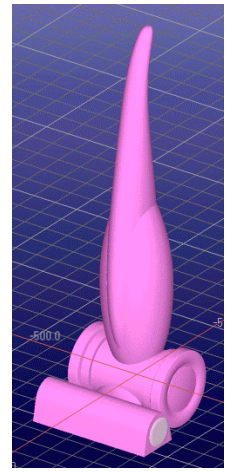
The model chosen to illustrate the techniques is shown here. It is a vacuum cleaner. The model you will create is a complete model of the external appearance of the product. This model will be used in the next chapter as a basis for experimenting with the creation of high quality images using the technique called ray tracing.

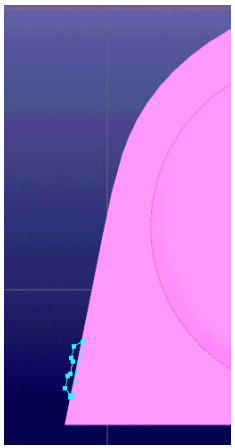
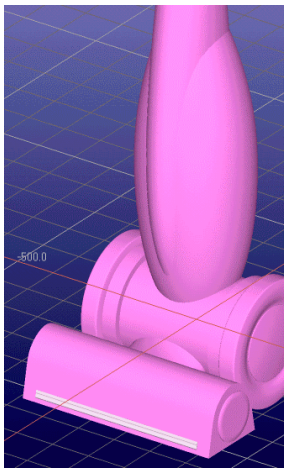
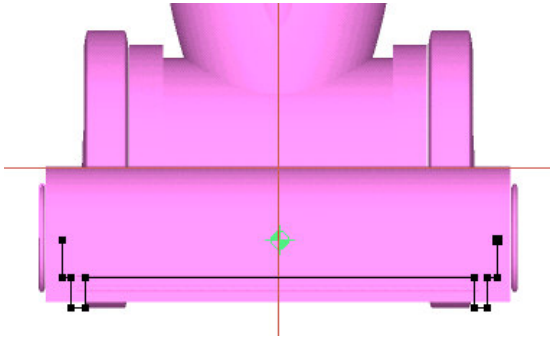
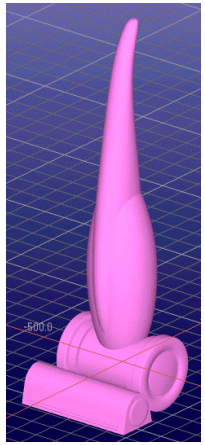
Now that you have had some experience of using Design Expert you will not be given a word for word explanation of the whole model. Instead we will use a comic strip technique for each stage of the model. This will diagrammatically describe the each of the curves and the surface created from them along with a few words of explanation. When new techniques are introduced then you will be given a fuller explanation.

	Profile	Silhouette	Sections	Comments	Result
Body Surface	 Draw 1 side then copy and mirror to make 2 curves	 Create silhouette curves from Y	 A single circular section with 16 points	Use the **Surface:Build** command to create the surface. Don't forget to use the Apply option to apply the secondary projections to the surface.	
Handle Surface	 Draw 1 side then copy and mirror to make 2 curves	 Create silhouette curves from Y and edit. Note the left side of these curves are just outside the body	 Start with a single circular section with 16 points. Duplicate this and edit the second section second section to flatten one side without adding or deleting points. All sections in a surface must have the same number of points.	Because you have defined 2 section curves you must tell Design Expert where these sections are to be positioned. First select one of the section curves then press . You will see numbers along the profile curves. Point to one of these numbers and the section will be applied there. Repeat this for every section curve. Use the **Surface:Build** command to create the surface. Use the Rotate button to make sure the flat on the cross sections are at the rear of the surface. Don't forget to use the Apply option to apply the secondary projections to the surface.	

	Profile	Silhouette	Sections	Comments	Result
Spindle Surface	 Weight all points in the curve to 3 - sharp corners	None required	None required	Use the **Surface:Rotate** command to create the surface. You do not need a silhouette or a section because the spindle is circular. Use the rotate around Y axis option.	
Tire Surface	 Weight the top edges of the tire to 2 –soft corners	None required	None required	When editing the curve move the origin marker to the center of the tire and then use the horizontal mirror icon to ensure that both sides of the tire are the same. Use the **Surface:Rotate** command to create the surface. You do not need a silhouette or a section because the tire is circular. Use the rotate around Y axis option. Duplicate the surface to make a copy then mirror this in the Y axis to make the second tire.	

	Profile	Silhouette	Sections	Comments	Result
Wheel Surface		None required	None required	Use the **Surface:Rotate** command to create the surface. You do not need a silhouette or a section because the wheel is circular. Use the rotate around Y axis option. Duplicate the surface to make a copy then mirror this in the Y axis to make the second wheel.	
Head Surface	 Draw this shape as a profile curve	None required	None required	Use the **Surface:Extrude** command to create the surface. Use the Through Curve option as the surface is extruded both sides of the current position of the curve. Also choose the Create End Caps option. Specify the length of extrusion numerically to be a bit wider than the wheels.	

	Profile	Silhouette	Sections	Comments	Result
TUBE Surface		None required		Create a circular section then scale it in one direction to form an ellipse. Make sure the width of the ellipse is the correct width for the tube. Use the **Surface:Build** command to create the surface. Choose the Section Scaling in one direction option then the width of the ellipse will determine the width of the tube. Use the Rotate button to make sure the ellipse is built across the tube.	
End Surface	 Draw this shape as a profile curve	None required	None required	Use **Transform:Fix Point:Interactive** command to set the fix point to the bottom of this curve. Use the **Surface:Rotate** command to create the surface rotating around the fix point. You do not need a silhouette or a section because the wheel is circular. Use the rotate around Y axis option. Now transform the object in X so that it sits on the end of the HEAD surface. Duplicate the surface to make a copy then mirror this in the Y axis to make the second end.	

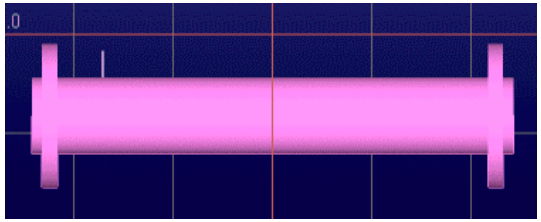
	Profile	Silhouette	Sections	Comments	Result
BUMPER Surface	 Draw this shape as a profile curve	None required	None required	Use the **Surface:Extrude** command to create the surface. Use the Through Curve option as the surface is extruded both sides of the current position of the curve. Also choose the Create End Caps option. Specify the length of extrusion numerically to be a bit narrower than the HEAD surface.	
Roller Surface	 Draw this shape as a profile curve	None required	None required	Use **Transform:Fix Point:Interactive** command to set the fix point to the bottom of this curve. Use the **Surface:Rotate** command to create the surface rotating around the fix point. You do not need a silhouette or a section because the roller is circular. Use the rotate around Y axis option. Now transform the object in X so that it sits **inside** the HEAD surface.	

That concludes the creation of surfaces that do not need any new techniques. The rest of the tutorial will introduce new techniques that require more explanation so the tutorial will revert to its narrative style.

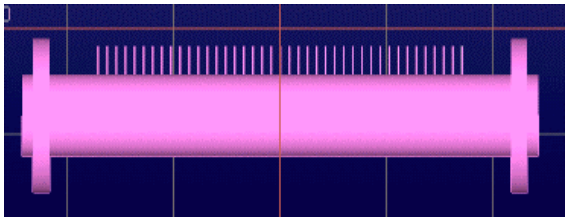
Bristles

Since the roller is the last surface you created lets finish it off by adding bristles. The bristles are in rows along the brush, which are repeated around the brush – but with a twist literally! To reduce the vibrations the bristles on a vacuum cleaner are always twisted around the brush. How can Design Expert help us achieve this easily?

Creation of a single bristle is easily achieved it is just a simple cylinder created using the **Surface:Primitive** commands you met in Chapter 1 – Basic Concepts. Create a cylinder in a new part called Bristles choosing a suitable length and diameter then transform it to sit on top of the roller.

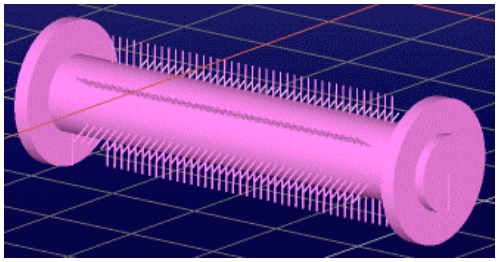


This is the first bristle. Now we need a long row of these. Rather than making lots of new bristles it is easier to copy this first bristle. **Transform:Duplicate** the surface then use **Transform:Move:Interactive** to move the copy along the roller. Now all that is need is to repeat this process – **Transform:Duplicate Transform:Repeat Trans.** Notice the short cut keys which make this much quicker <CTRL>D and <CTRL>R. Just repeat these key presses until you have a row of bristles along the brush.

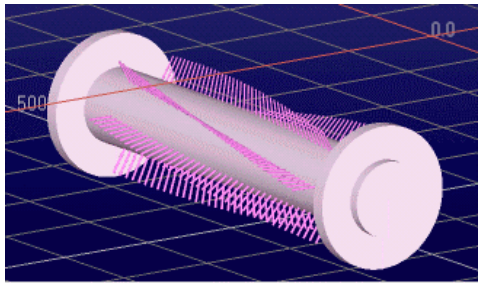


To create the other rows use the same technique but this time they need to be rotated around the center of the Brush part. To achieve this select the Brush then use **Transform:Fix Point:Interactive** and point near the center of the brush. Now select the part called Bristles and use **Transform:Fix Point:Copy** to set the fix point of the bristles to the same as the brush.

Select the part name Bristles and Duplicate the whole part. Now **Transform:Rotate:Numeric** and enter 60 as the angle and Y as the axis. Duplicate and repeat the transformation to create 6 rows around the brush.



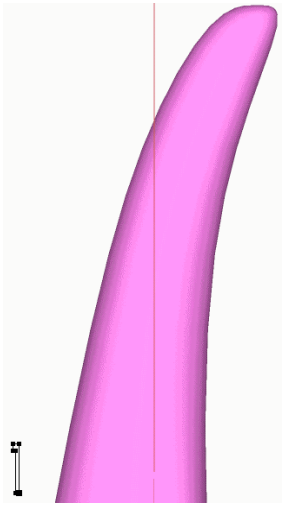
Now the twist! Select the first part called Bristles and pick **Transform:Deform:Twist**. The twist will occur along the Y axis by 60 degrees around the fix point enter this information and the first row of bristles should be shown twisted. Repeat the process for the other 5 rows.



Since all this is inside the head of the vacuum cleaner you wont be able to see it if you display the whole model. Later we will add transparency to the model so your hard work does not remain hidden!

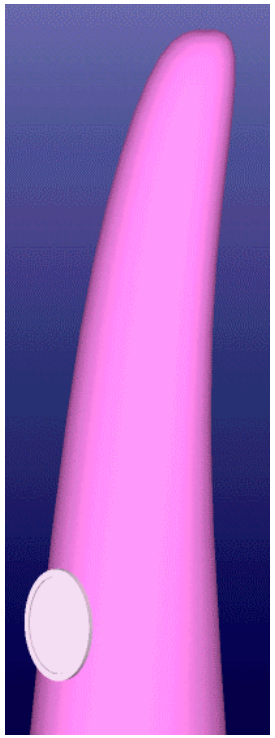
Logo

Lets make a disk for the manufacturers logo. Create a new part and draw the L shaped curve shown below.



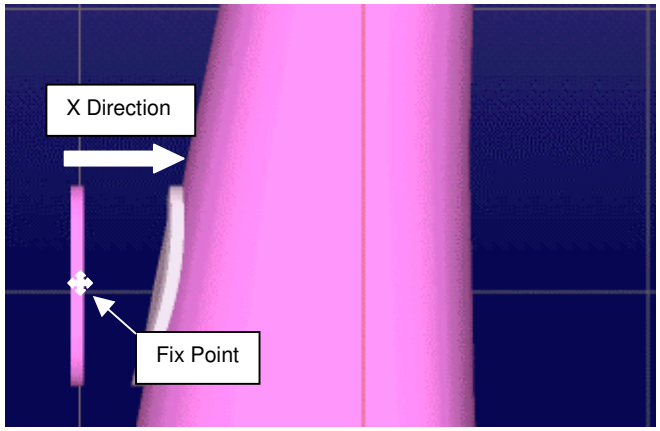
Notice that the curve is not touching the handle surface. Because the handle is curved the Logo surface needs to be wrapped around the handle.

Create an ellipse as a section curve (scale a circle in one direction). Set the fix point to the bottom of the profile curve and create a surface by Rotation.



You can use one of the Transformation commands to wrap the logo around the handle. Select the logo surface and set the fix point to the back of the logo. Now choose **Transform:Project To Surface** command. Point to the

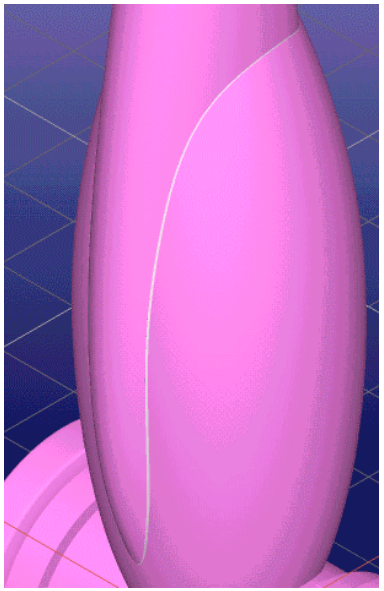
handle surface on the screen as the surface around which the logo will be wrapped. The projection direction is X and the method should be orthogonal.



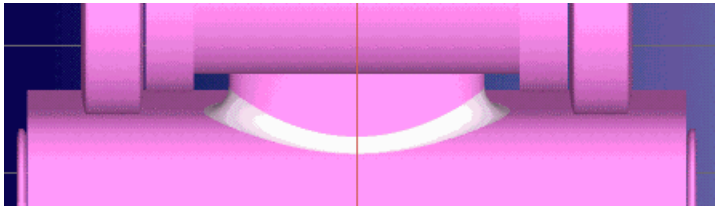
More Cutting and Blending

The basic principles of cutting and blending were introduced in Chapter 3 - More Complex Modeling with the creation of the finger grips in the lid. It would be good to go back and review that section now before continuing.

There are many surfaces in this model that have been deliberately designed to pass through each other – for example the handle surface goes right inside the body. Before we investigate some new features of the Combine menu you should add a small rolling ball blend between the Handle and the Body surfaces. Select the Body surface, **Combine:Rolling Ball Blend**, choose the correct sides of the surfaces and a suitable radius value.



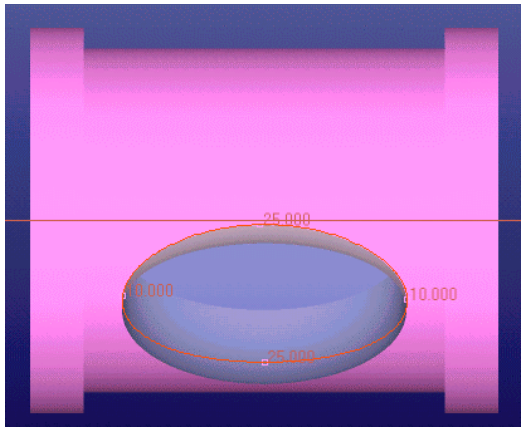
Also use a similar technique to add a large blend between the Tube and Head surfaces.



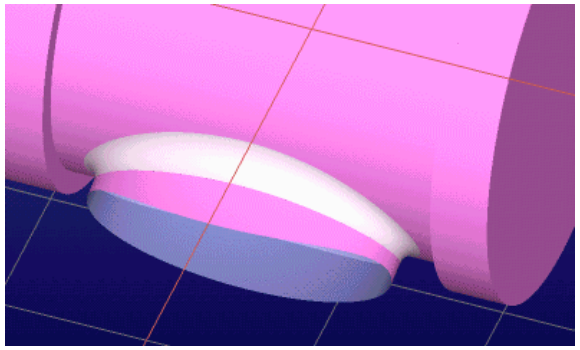
You could create a similar blend between the Tube and the Spindle but instead we will use the opportunity to introduce another blending technique – variable radius rolling ball blends.

Variable radius blends are very similar in execution to normal rolling ball blends. To better enable you to see

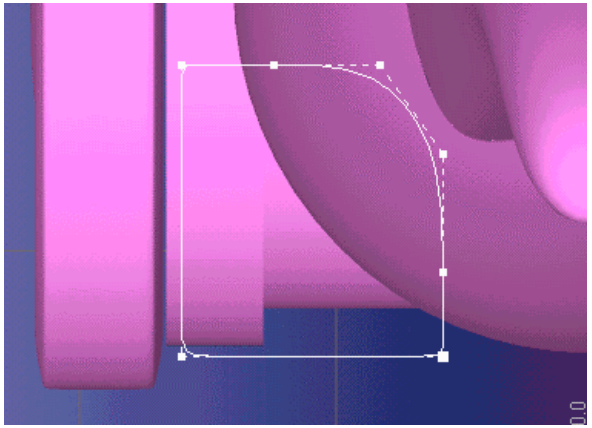
what is happening clear the screen and display only the Tube and Spindle surfaces. Look from the X direction with the Tube surface selected. Issue the **Combine:Variable Radius Blend** command and point to the Spindle surface. You will be presented with the Rolling Ball dialog box with which you should by now be familiar. Use it to choose the correct surface directions after which a red line will be drawn around the intersection of the two surfaces. Design Expert is now waiting for you to define the radius of the blend at different points around the intersection. Look at the diagram below, which shows four positions and the radii associated with them. Point to each of these positions in turn and enter the radii. You may need to adjust the radius values you enter if you model is to a different scale (use the value you entered for the previous rolling ball blend as a guide). When you enter the fourth radius value remember to tick the Ready to Blend option to start the blend creation process.



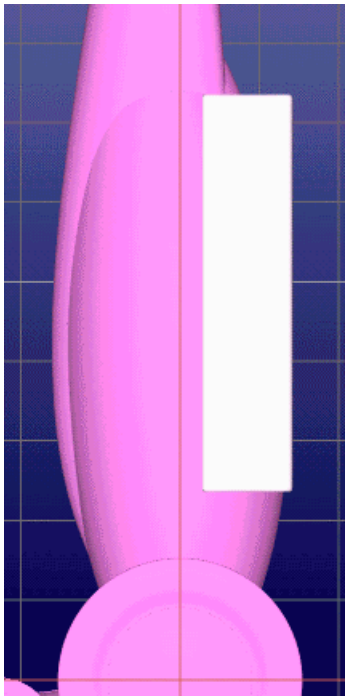
You should see the blend created after a few seconds. If you are warned that the blend was unable to be created the probable reason is that you entered too large radii and the blend would not fit onto the tube surface.



As another example of blending lets create some cutouts for the vacuums tools to sit in. Create and draw the curve shown below from the Z direction in a new part called Attachment Cutter.



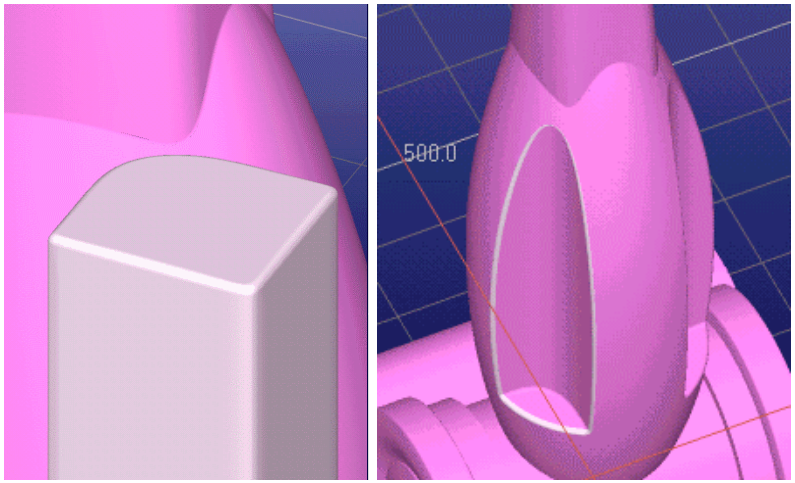
Extrude this curve to make a surface and then translate it in Z to the correct position. Choose a suitable extrusion distance and add End Caps to the extrusion.



To make the cutout we will add a rolling ball blend around the intersection with the Body surface. With your experience of blending you should be able to attempt this now – be careful to choose the correct sides of the surfaces.

Did it work? No you probably got an error message about a difficult surface or tight corner – why? The reason is that end caps of the Cutter surface are created with a sharp corner and Design Expert is not able to create blends around these sharp corners. We could change our plans and simply trim the two surfaces but this would leave sharp edges when what we intended was a smooth join.

There is a simple solution. Design Expert has a command that rounds off all sharp corners of a surface. If we did this it would mean that we could create the blend surface as we originally intended. Select the surface and use the command **Surface:Reshape:Fillet** choosing the same radius value as you would like to use for the blend. After the command if you look carefully at the surface you will see the ends are rounded off. Now the rolling ball blend can be created successfully – but first duplicate and mirror the surface for the other side.



Chapter 6 – Advanced Rendering

Chapter 4 – Rendering introduced some of the options available to you to define materials and the scene so that models can be rendered in a realistic manner. Rendering is a 'half-way house' – it gives reasonable quality appearance and is still quick to generate.

Design Expert offers an additional optional module for creating VERY realistic pictures but at a price – the pictures take longer to generate – usually a few minutes but for the most complex models this could stretch to hours.

The technique used to generate the pictures is called Ray-Tracing. The name is derived from the fact that the program simulates the physics of light by tracing rays of light from the computer screen back to the light source. Ray-tracing is a technique used by many modeling programs but what is unique about Design Expert is that, in recognition of the time a picture can take to generate, a picture refining technique is used so that you quickly get to see the results.

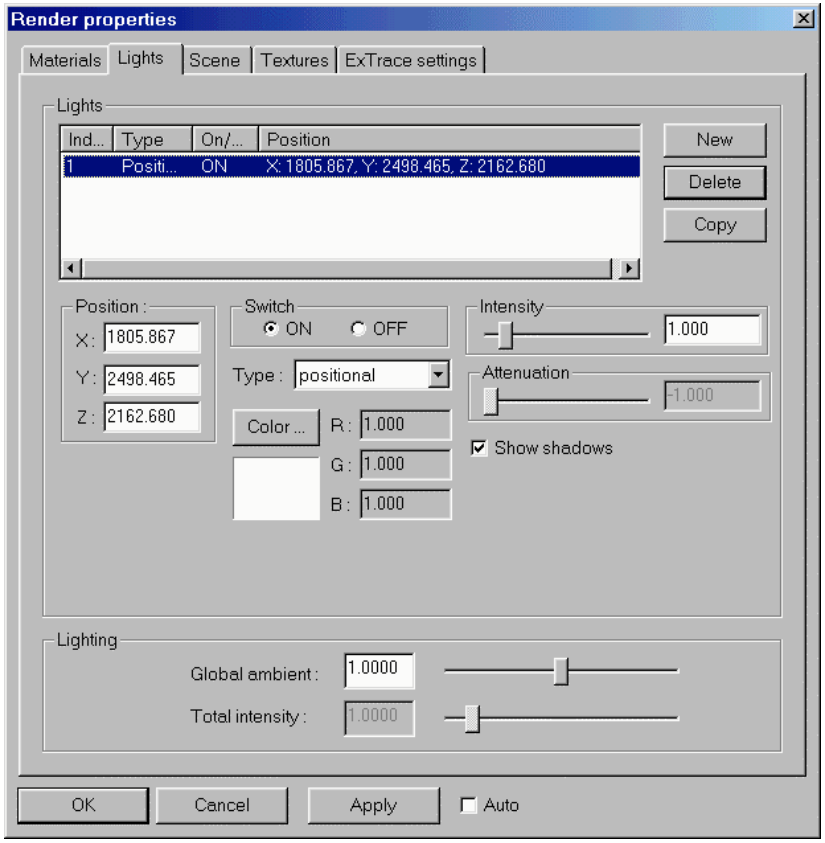
Try and reproduce the next picture using the rendering techniques you have already learnt. Load the plastic material and change the color to black, copy it and change the color to light gray. Make a third white material that has diffuse and ambient light set to 0.7 and the rest of the parameters set to 0.

This is the rendered image lets quickly create a raytrace to compare. Choose the command **Render:ExTrace Settings** and in the dialog box remove the tick from the Camera View option (so the view in the render window - known as the Eye View - will be drawn) and Apply. A new window called ExTrace will be created and the picture will

start to generate – very blurred at first but refining as you watch. After you have watched for a while, press the escape key (ESC), which stops the rendering process. The ExTrace window can be resized by dragging a corner to make it more in proportion to the vacuum cleaner. Right click the mouse in the ExTrace window to pop up a menu and choose Redraw to start the rendering starts again. As the picture gets to a high quality you can see how it differs from the render window. ExTrace generates shadows and also the reflections are true reflections not simulated.

Define Lights

Lighting on the model is adjusted by the command **Render:Define Lights** or the icon . Choosing this command will create a large dialog box. At the top of the dialog the lights are listed – click on the first (only) light in the list and the parameters for that light will be displayed.



Most areas of this dialog are self-explanatory.

Position shows the coordinates of the light, which can be changed by entering new values but there is a more interactive way of doing this, which will be explained later.

Switch turns the light on and off.

Color allows a colored light bulb to be defined. The standard windows color picker is used.

Intensity varies the brightness of the light.

Type allows you to choose between a positional or spotlight. If you choose spotlight you will also have options to define where the light is shining towards and the angle of the spot although these can be changed graphically.

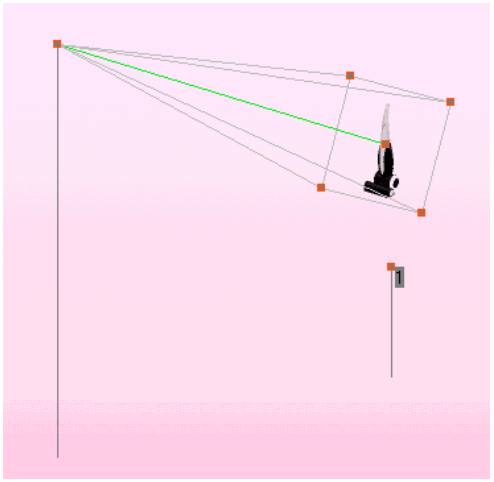
At the bottom of the box are two sliders. Global Ambient and Total Intensity. These sliders are very useful. As you add more and more lights to the scene it is possible that the picture may become very bright causing white-out. You can use these sliders to reduce the overall brightness of the picture.

The New, Delete and Copy buttons work on the currently selected light in the list.

Scene Edit

Graphical position of the camera and lights is achieved using the **Render:Scene Edit** command . Press this button now and zoom out to see more of the scene. You should see a camera and a light around the model.

Both the camera and the light have red 'grab' handles on them, which you can use to move them around. You can use either the left or the right mouse button. The left button moves the light/camera round the model but keeps the height of the light/camera above/below the floor constant. The right button lifts the light/camera up and down. The four handles on the end of the camera lens can be used to adjust the view angle.



The right button can also be used to bring up a context sensitive menu through which you can create a new light.

The ExTrace window should still be open. If it is not press to start rendering again.

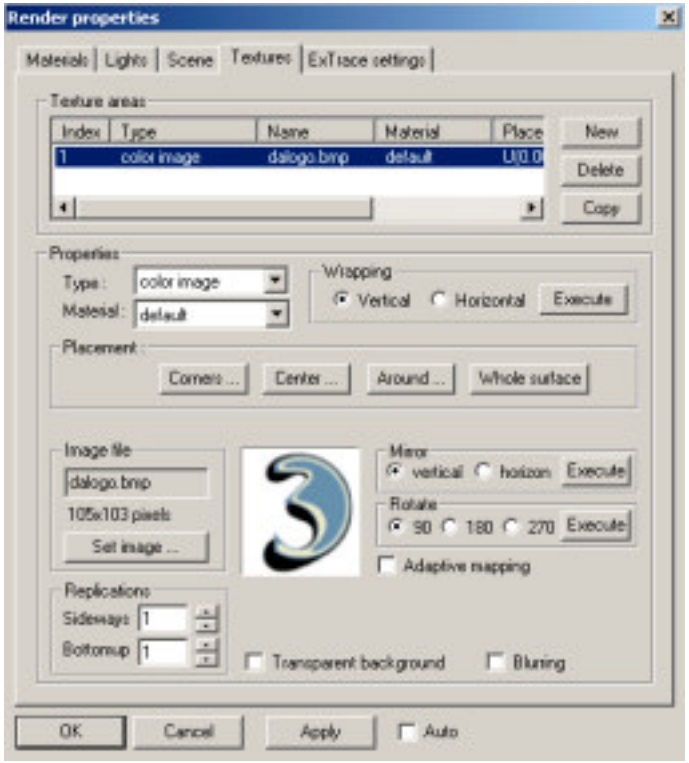
In the ExTrace window right click and make sure there is a tick next to the Camera View option.

Experiment with camera and light settings – after each change choose Redraw from the menu in the ExTrace window and when you have seen enough press the ESC key to stop rendering.

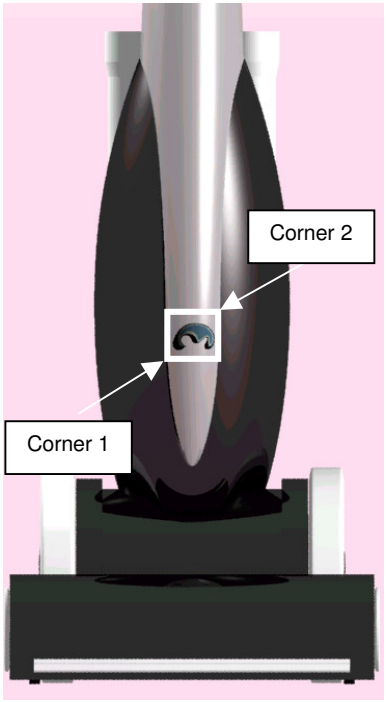
Note: There are Zoom and pan commands in the Extrace menu which are very useful for making fine adjustments to the camera position.

Define Texture

Another area that can add realism to a scene is to add decals or textures to the model. Lets add a simple logo texture to the vacuum cleaner. Select the Handle surface then issue the command **Render:Define Texture** . The texture dialog will appear – press the New button to create a new texture and select it in list. The parameters for this texture are displayed.



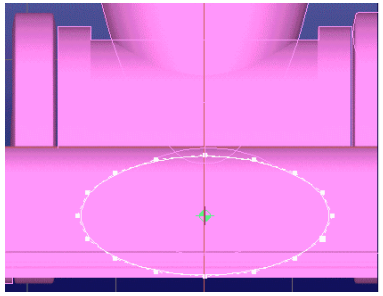
The default image is the dalogo bitmap, which will make a good logo for the vacuum. The first job is to position the logo onto the surface. Press the Corners button and point to two opposite corners of logo position on the Handle surface. Also tick the Transparent Background and Blurring options then press the Apply button to see the decal in the ExTrace window.

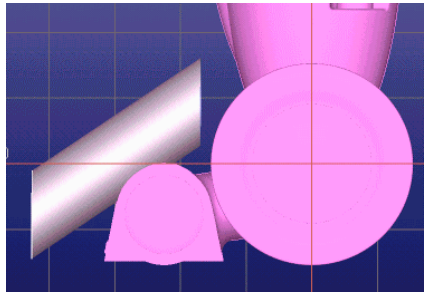


3D Building

The model is nearly complete but remember the brush hidden inside the vacuum head. We said we would modify the model so that it is visible.

Create a circle X profile in a new part called HeadCutter. Scale this into an ellipse and then translate so that it is placed in front of the vacuum head. Extrude a surface from this curve using the graphical option. The new surface should intersect the Head – if it does not Translate it to the correct position.





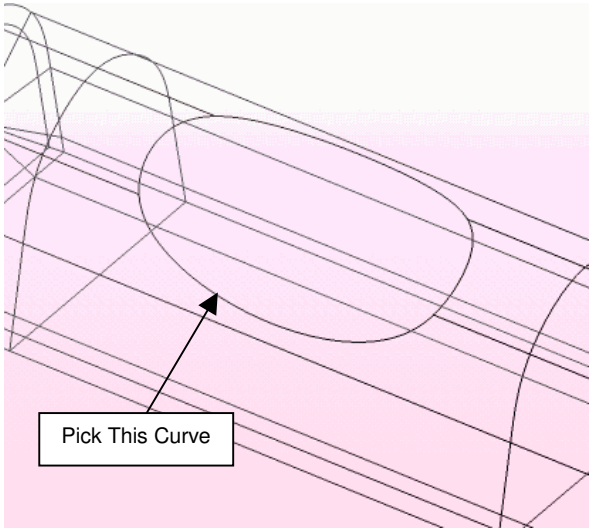
We are going to use this new surface to cut a hole in the head. Use **Combine:Trim Surface/Surface** command to cut this hole – make sure you keep the correct part of the Head surface. Afterwards right click on the HeadCutter part in the model list and choose **Make Passive** as this is not really part of the model.

The hole means we can see the brush but it also means the dust has an escape route! What we need is a transparent window. This is surprisingly easy. Make a Duplicate of the Head surface and issue the command **Combine:Invert Cut Part**. This changes the surface so

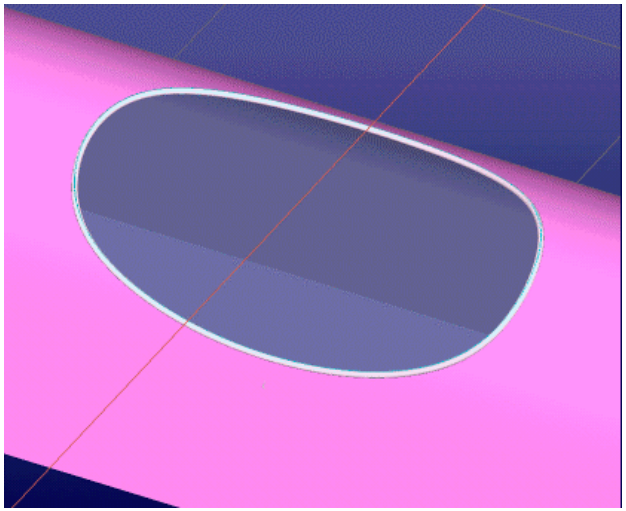
that the hole becomes solid and vice versa. But we can't see the brush again! Easy just create a new glass material and assign it to inverted surface (transparency is only simulated in Render or ExTrace not in View).

To finish the model we just need to add a surface to cover the join around the transparent surface. This is a very complex shape so we need to use a new technique to create it. The technique is called 3D Building.

Erase the screen and show just one of the head surfaces also turn on wireframe display . Create a new part called Band and select this new part. Now issue the command **Curve:Pick Curves** and point to the curve bounding the hole press the left button to pick the curve then the right button to finish curve picking.



You should see 3D Curves in the new part and this should contain one polyline. Select this polyline and issue the command **Curve:Representation**. Choose Spline and Number of Points 100 in the dialog and the polyline will be converted into a 3D spline. Now click on 3D Curves and choose **Curve:Change Type** then Profile. The final step in preparation is to draw a small circle as a cross section. **Surface:Build** should create the surface shown below.



Reviewing this complex process we have…

- Used a wireframe curve on the screen to produce a 3D Polyline.
- Converted the polyline to a Spline.
- Converted the 3D Curves to a Profile
- Added a circular section curve.
- Built the surface.

That completes the tutorial. There is still lots more to learn so feel free to experiment with any new command you find. In particular you may want to investigate further options in the Deform menu, further methods of creating surfaces such as skinning and textures, which we only covered briefly.