HEXAGON
MANUFACTURING INTELLIGENCE

ALPHACAM 2020.1
SURFACE CREATION

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Contents

Table of Images

Conventions used in this manual

To enable you to use the information in this guide effectively, you need to understand the conventions used in the guide to represent differing types of information.

- Buttons on the screen are represented as the button text in square brackets.
  For example: Click on **[OK]**.

- Keys on the keyboard are represented as bold lettering in between **< >** characters.
  For example: Press **<Enter>**.

- Ribbon Tab options are represented as a path with the Ribbon Tab in **UPPER** case with sub menus Capitalised and separated with an arrow
  For example: Select **FILE > Open.**

- Field names are represented as bold text. And the value to be entered will be represented by Bold Text.
  For example: Enter the value **50** in the **Offset** field. Or
  When prompted for the X & Y values type **100,50 <Enter>**


Denotes a **<LClick>** or Primary Mouse Button Click.


Denotes a **<RClick>** or Secondary Mouse Button Click.


This is a note. It contains useful or additional information.


This is a reference. It directs you to another part of the user guide.


This is a thought box. It is generally used in exercises and contains a question for you to consider.


This is a highlighted note to emphasise information


This is a warning; it contains information that you must not ignore.


This is a tip. It is generally used in exercises and offers further advice.

1. This is the first line of a number list item
2. This is the second item of the numbered instructions, which you must
3. Follow in sequence.

- This is a list
- of items, in which
- The order is not important.

Recommended Operating Systems and Hardware for Alphacam

Supported Operating Systems

- **Operating System**
- **64bit** operating systems of the following list are supported,
  - Windows 7 (Professional, Enterprise or Ultimate) SP1 required,
  - Windows 8.1 Professional and Enterprise,
  - Windows 10 Professional and Enterprise.
- Alphacam will install and run on the 'Home' editions of the above operating systems. However, this is not recommended, and we cannot guarantee to fix any Alphacam issues specifically related to these operating systems.
- Nvidia or ATI Open GL Graphics Card with 1Gb dedicated memory

We recommend you keep up to date with the with the latest Software Updates for the supported operating systems and drivers for your hardware base.

Any Windows Operating system (OS) prior to and including Vista, is not a supported operating system.

ALPHACAM Minimum Specifications

The latest minimum specification can be found at http://www.alphacam.com/systemrequirements

This minimum specification is to run any **ALPHACAM Essential** module, you will need to considerably increase the specification if you are working with solid models and producing the NC code for 3D machining and 3, 4, or 5 axis simultaneous machining strategies.

Your minimum specifications should be the fastest processor with the most memory and the highest specification video card that your budget will allow.

If using Autodesk Inventor Files, please check the current Inventor View requirements at autodesk inventor view

Hexagon Customer Portal

At Hexagon, we strive to provide you and your business with first class technical support and services. The Customer Support Portal allows you the tools you need to receive the best from your software. In addition to generating new and updating existing support cases, the portal allows you:

- Unlimited user logins for your company.
- Access to all your licenses for easy reference.
- Get the latest releases and software update at the touch of a button.
- View what is available on your support and maintenance schedule.
- Reference to the Customer Support Charter at any time.
- View the status of your Technical Support cases.
- View all purchased Professional Services like Training and Consultancy.

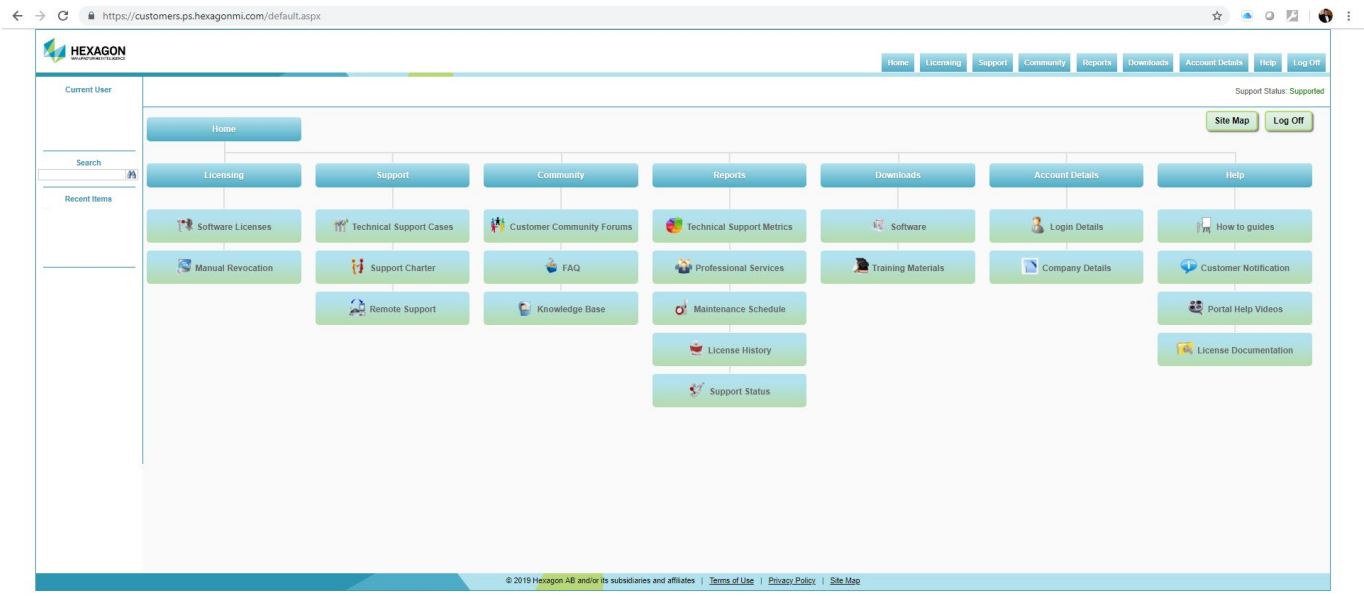


Figure 1 - Customer Support Portal

For the Hexagon Customer Portal visit customers.ps.hexagonmi.com

ALPHACAM esupport

Another location to gain valuable information about using the software or asking other experienced users for assistance are the esupport forums.

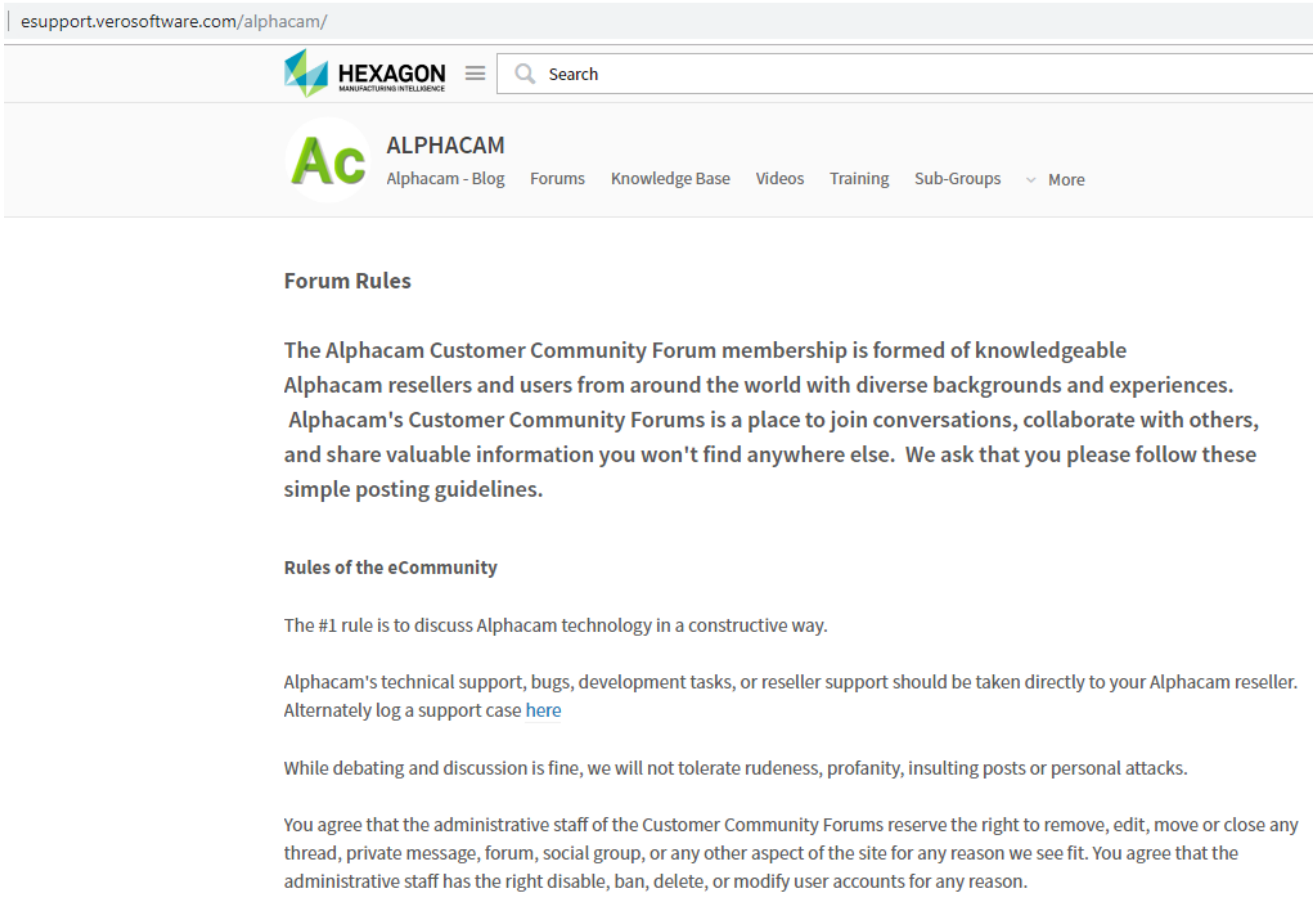


Figure 2 - esupport page

Asking a question of the community, using the knowledge base or other available information links could save you time if you have a problem that someone else may already have supplied a solution for.

Objective

The purpose of this training course is to enable you to create, manipulate and machine Surfaces.
It is the capabilities of the machine tool that places restrictions on the position and orientation of the surface and the type of machining strategies that can be used.
In order to create surfaces, you have to be able to create the surface definition edge geometries in their respective work planes.

What are work Planes

Work planes are 2 Dimensional sections through 3D work space. The 2 D section represents the working Z Zero plane.

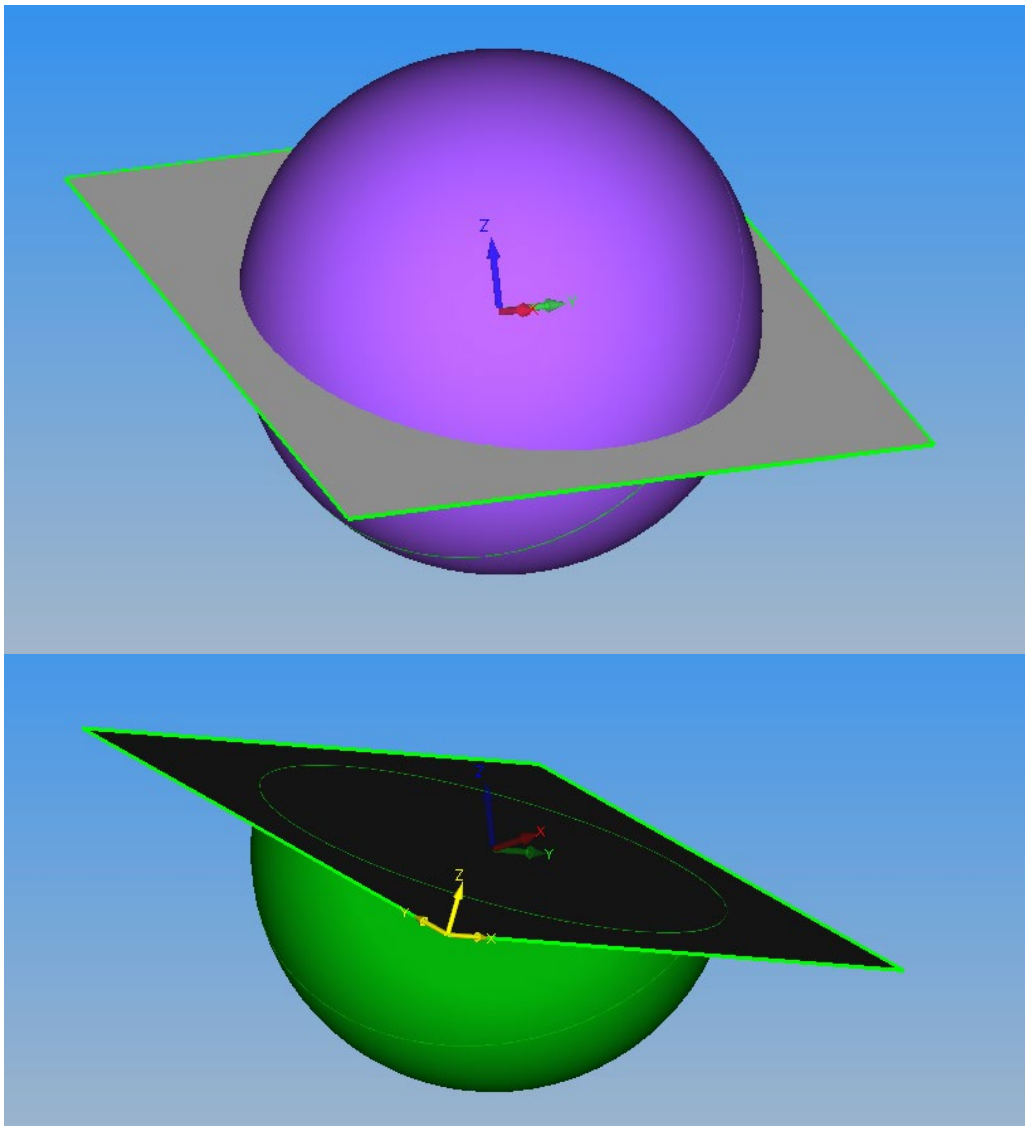


Figure 3 - Visual example in 3D of the concept of Work Planes

Work Plane Terminology

Global Datum

Global X0, Y0, Z0, shown in Red(X), Green(Y) and Blue (Z) also known as the World Coordinate System WCS. This is the equivalent on the machine datum (G92).

Local Datum

Local X0, Y0, Z0, shown in YELLOW, also known as the User Coordinate System UCS. This is the equivalent of machine datum shift (G54, G55, G56).

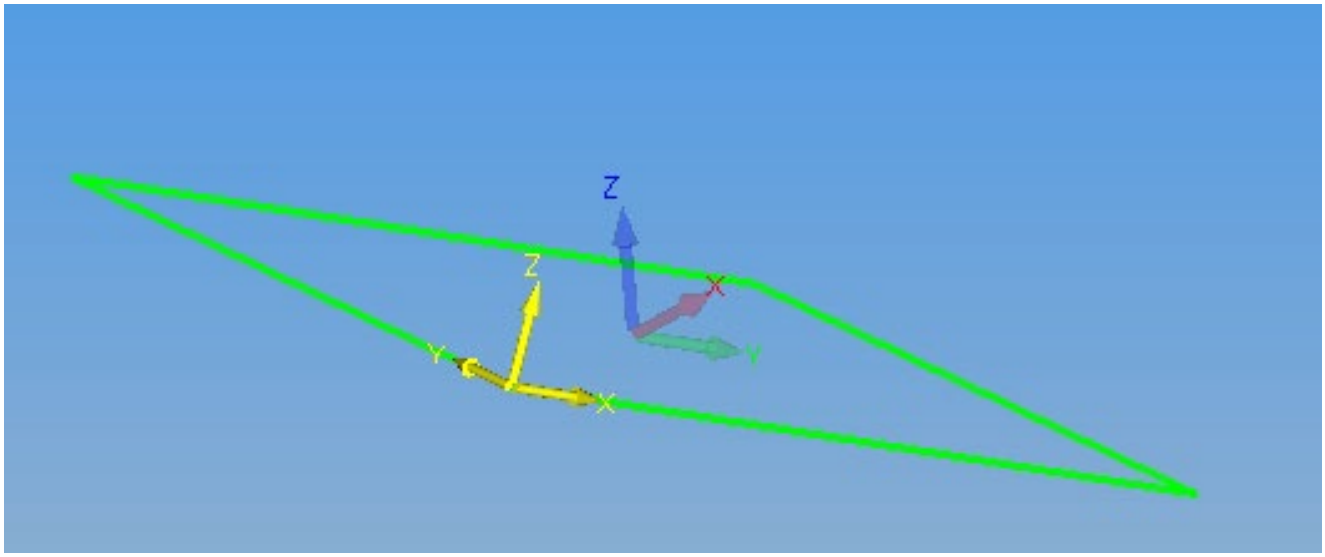
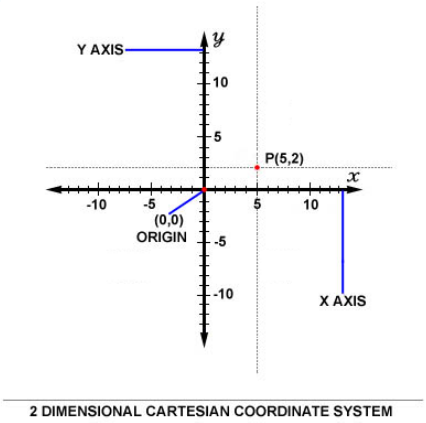
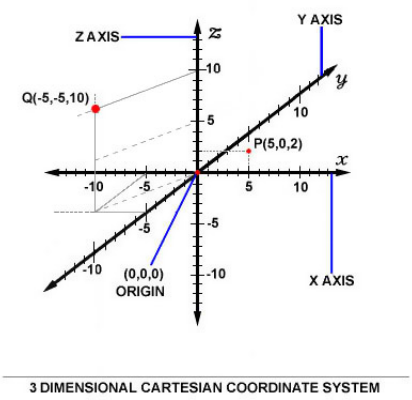


Figure 4 - Work Plane origin colour

Alphacam works using a Cartesian coordinate system.



2 DIMENSIONAL CARTESIAN COORDINATE SYSTEM



3 DIMENSIONAL CARTESIAN COORDINATE SYSTEM

Orientation of the Coordinate System obeys the Left hand rule.
This has implications for the use of the Rotate command, where the CW and CCW directions are always set when looking down the axis back towards the origin point around which the rotation will take place.

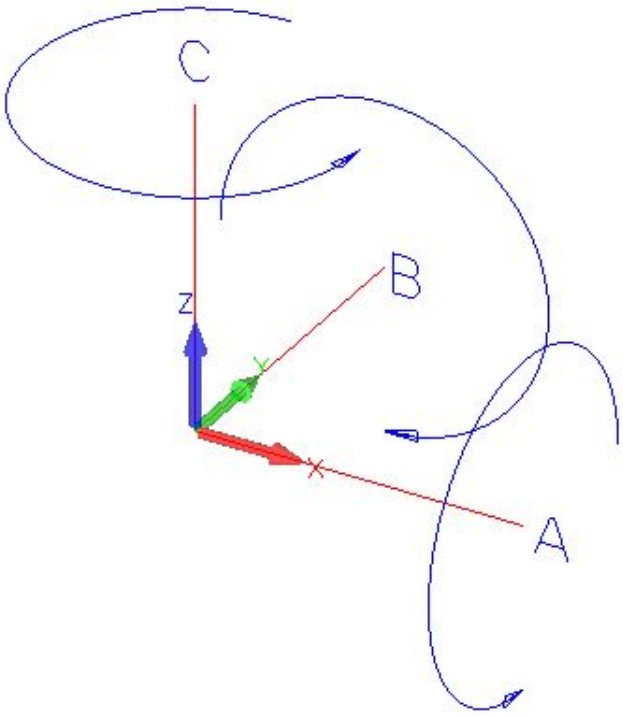


Figure 5 - Left hand rule for axis rotation

Flat Land

The term **Flat-Land** is used to represent the default 2D working environment. It is similar to Paper Space a piece of paper on the ground.
You cannot remove **Flat-Land** it is ALPHACAM's default drawing sheet.
Flat-Land is the default drawing status of ALPHACAM; it is the same as an infinite sheet of paper laid flat on the ground, upon which normal 2D geometry and tool paths are created. Flat-Land's coordinate dimensions are with respect to the global X0, Y0, Z0 datum WCS.

Work Volume

The work volume is a rectangular bounding area enclosing the 3D working envelope. The purpose of the work volume is to give a visual perspective of the bounding area.

The creation of a work volume is not mandatory however it can be beneficial to inexperienced users. (the panel macros create a work volume).

Work Volume Creation

The work volume is created by converting a 2D Rectangle into a work volume and adding top and bottom Z values. The work volume is created with respect to global X0, Y0, Z0, datum WCS.

The command **3D > Set Work Volume** converts a 2D rectangle into a work volume.
Create a rectangle from **X0, Y0** to **X200, Y100.**

Select 3D > Set Work Volume and pick the rectangle.
The define work volume dialogue will be displayed, set the Top Z to **0** and the Bottom Z to **-100.**

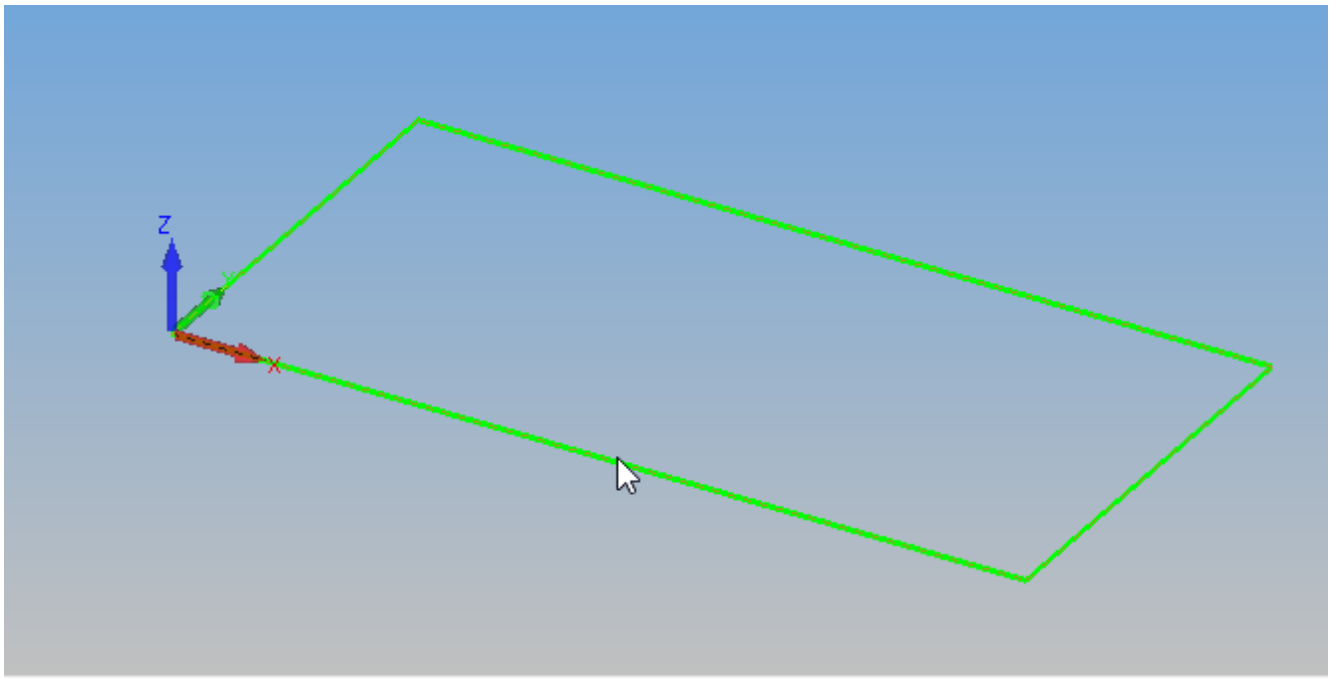


Figure 6 - Created Work Volume

The Define Work Volume dialogue will be displayed set the Top Z to **0** and the Bottom Z to **-100**.

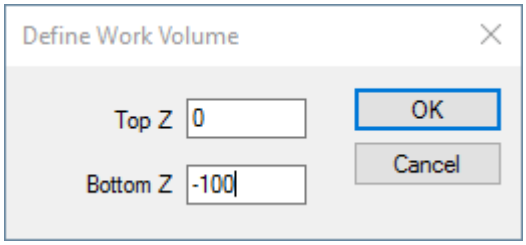


Figure 7 - Work Volume Z values

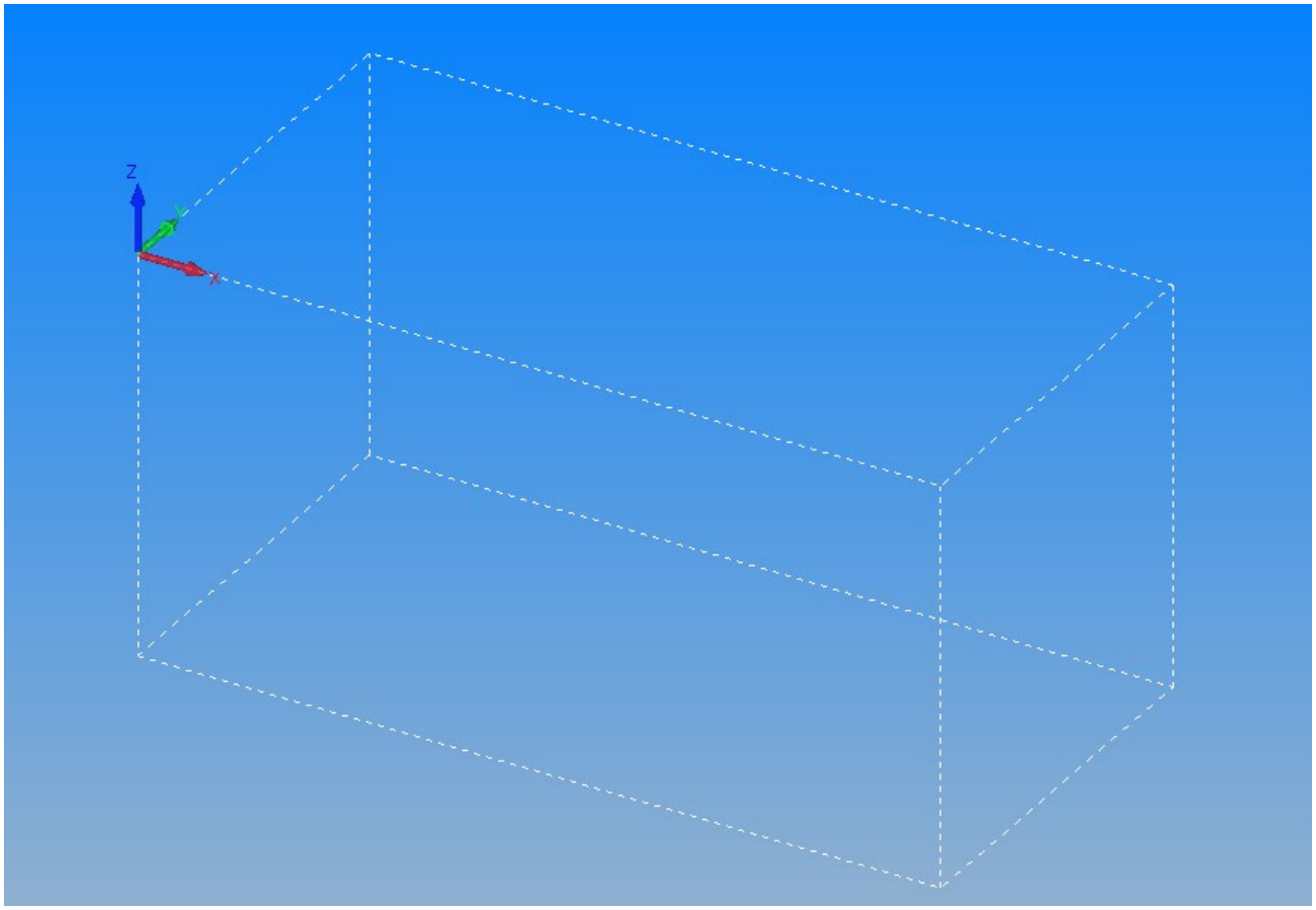


Figure 8 - Created Work volume

Select **WORK PLANES > Slice through Work Volume**  pick the X axis top edge of the work volume.

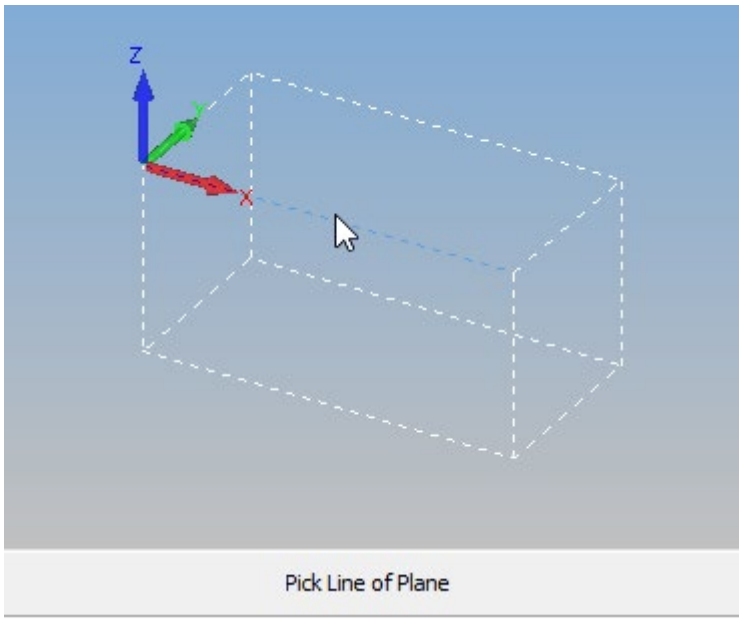


Figure 9 - Selecting a line to Slice Through

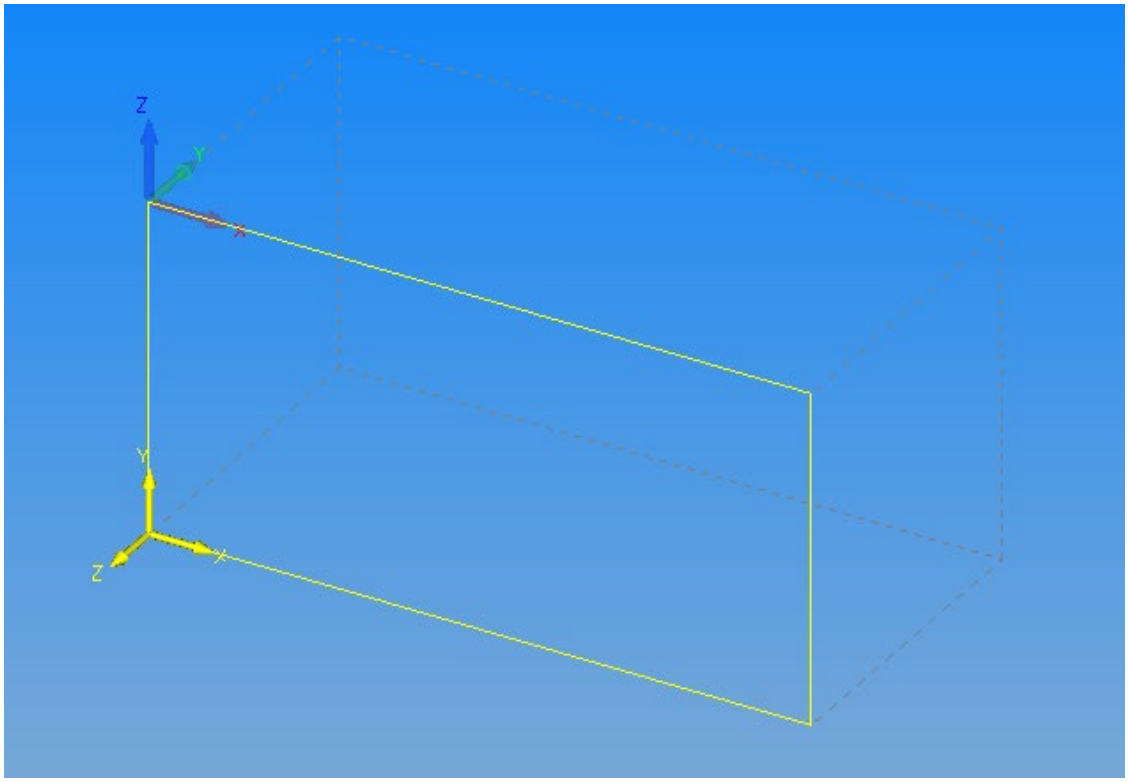


Figure 10 - Generated Work Plane

This example shows how using a Work Volume can allow simple side and end face work planes to be created.

Select **VIEW > Work Plane**  this changes the view to only show the current work plane.

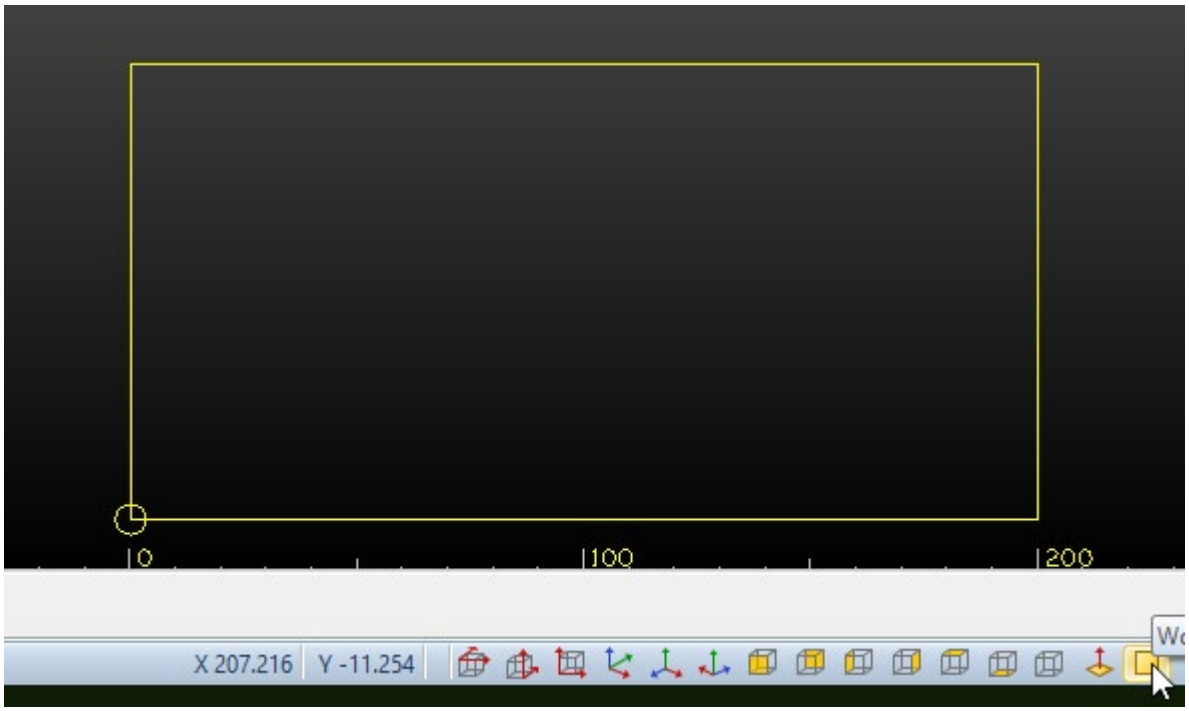


Figure 11 - Work Plane view option

Use the View options to see how they change the display.

Figure 12 - View options from the prompt bar

Work Plane Commands

Set Work Volume 

This allows you to select an existing rectangle to convert to a work volume by setting the top and bottom Z values as shown.
There are some additional Add-ins that automate the work volume creation.

Auto Set Work Volume 

This will automatically create a work volume bounding the extremities of, geometries (3D or with Z levels), or the toolpaths (including rapids).

Auto Set Work Volume on CAD Input 

This will automatically create a work volume bounding the extremities of, geometries (3D or with Z levels). Auto set work volume does not work on 2D geometries.

> It is worth considering the orientation of the part when using the Auto Set option. If the item is not aligned to suit manufacture when created then the created work volume on import will not be correct once the item is set correctly.

Set Work Plane Origin 

This allows you to change the position of the local plane UCS origin point with respect to the global WCS origin position. The position set MUST lie on the active work plane.

Work Plane Properties 

This allows you to set: The reference name/number of the plane as it is displayed in the project manager page. The Offset Number that is used by the post processor to reference the plane in the NC Code.

Order Work Planes 

This option allows for the ordering of work planes to suit the shortest movement for toolpaths on differing work planes when using 4 and 5 axis moves.

Cancel Work Plane 

This returns the status of Alphacam to **Flat-Land**. It is the same as picking “**Flat-Land**” name on the Work Plane Project Manager Page.

Make Local Axis Horizontal 

This command will automatically manipulate existing work planes so that either the X or Y axis is maintained Horizontal with respect to the Global Axis. This allows you to re-orientate planes to suit the machine configuration.

Work Plane ribbon section

This provides access to the commands that allow work planes to be created and activated. A work volume is not required to define work planes.
Activation of work planes can also be done from the **Work Plane** Project Manager page.

Slice through Work Volume 

This allows any line geometry to be used as a knife which cuts through 3D space to create the work plane (2D section through 3D space).

The Slice command only works in one of the Ortho Views (XY, XZ, ZY) and any Slice in the ISO view will always be in the global Z direction.

When using the Slice command, you do not have control over the orientation of the local axes, it is important that you ensure the axes directions are suitable for your machine tool.

2 Lines for X and Y axes 

This allows you to use two lines whose ends are coincident, that lay in different work planes, to define a work plane. The first line selected defines the new work planes X axis orientation. The second line selected indicates the new work planes Y direction. The lines meeting point is used for the planes origin position.

Existing Geometry 

This allows you to pick geometry on screen and activate the plane on which the geometry is associated. If the command does nothing, then the geometry picked is in “Flat-Land”. No new work plane is created.

Parallel to Current Plane 

This will create a new work plane that is parallel to the currently active plane. The new plane is created the specified distance from the current plane in the Z axis direction. Positive or negative direction is allowed.

Perpendicular to Current Plane 

This command will create a new plane that is perpendicular (90°) to the current plane.

Reverse Current Plane 

This command will flip the current planes axes through the 4 possible orientations.

As Current Plane, Different Origin 

This command will duplicate the current plane and allow you to set a new local origin.

By Name 

This will display a dialogue showing the names of all the existing planes. The dialogue allows you to activate a plane from the list, (same as picking the name in the Project Manger page), or Clear, (delete all), Empty work planes. This is very useful if you create a work plane by mistake.

Best Fit Plane through Points. 

This will activate a work plane that fits evenly through the points in a 3D spline or polyline the spline or polyline must contain at least 6 points. This option is used primarily when there are no natural 2D options or planar model faces available.

> Polylines and 3D splines exist in 3D space and NOT on work planes.

By Line Element Vector 

This will create a work plane perpendicular to one end or the other of a selected 2D geometry line or 3D polyline. The direction and location of the UCS origin is determined by the direction in which the original element has been drawn and the selected answers to the creation dialogue options.

Normal to view 

This will create a work plane in the exact direction that the user is viewing the drawing or solid model. This method of creating a work plane can prove very useful when no reference geometry is available, no 2D or 3D items or the solid model does not have a flat face.

From Solid Model Face 

This will create a work plane that lies on the flat model face, or perpendicular to a model cylinder hole, selected.

Create 3D Work Plane 

This method of work plane creation uses direct input rather than item selection.

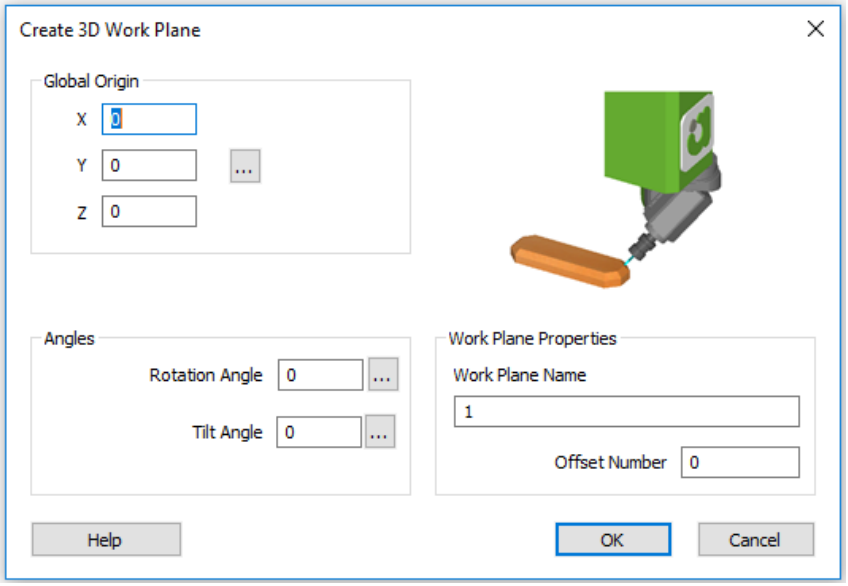


Figure 13 - Create 3D Work Plane dialogue

Using the X, Y & Z options, you can directly state where the work plane origin is to be located.

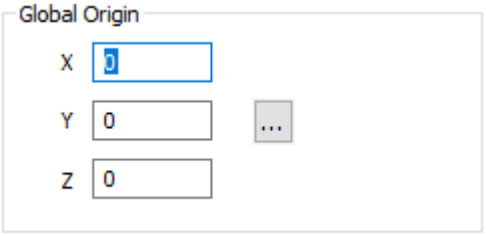


Figure 14 - Origin location

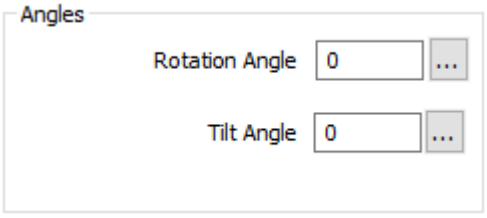


Figure 15 - Angles options

The **Rotation** and **Tilt** angles allow you to specifically set the required values, useful if you have a limited amount of movement on a machine head or table.

Using the browse option  will allow you to set any to these options directly from the part.

What are Surfaces?

A Surface is the mathematical description of the outer skin of an area. A surface is created by effectively stretching a skin between the edges defining the area.

Surfaces have no thickness and cannot be seen when looking perpendicular to the edge.

Surface Creation Methods

3D Surfaces can be created by one of eleven methods depending upon the edge categories.

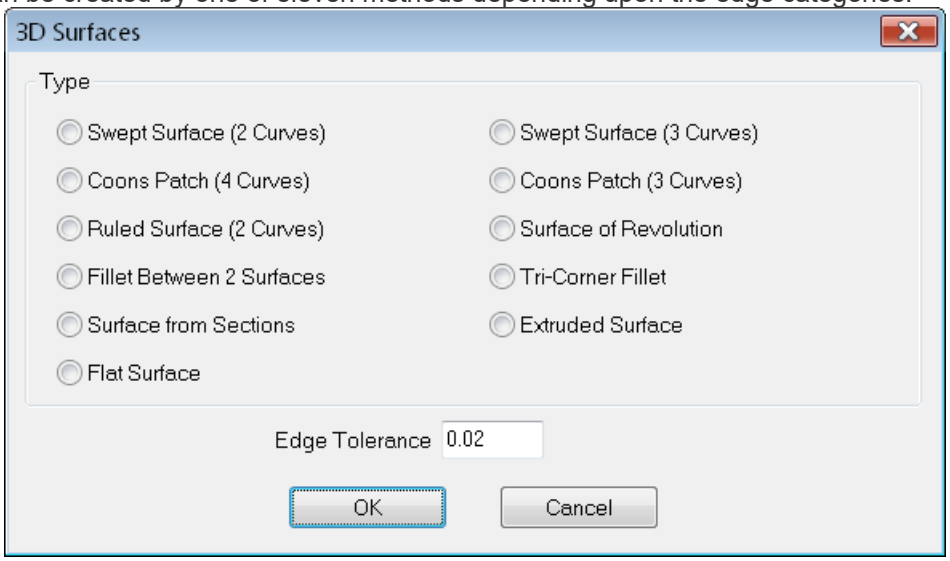


Figure 16 - Surface creation dialogue options

When creating a surface, you have to specify an edge tolerance, the smaller the tolerance the more detail is defined in the surface; however, this increases the storage size of the surface.
The edge tolerance should be set with respect to the size and detail of the surface.

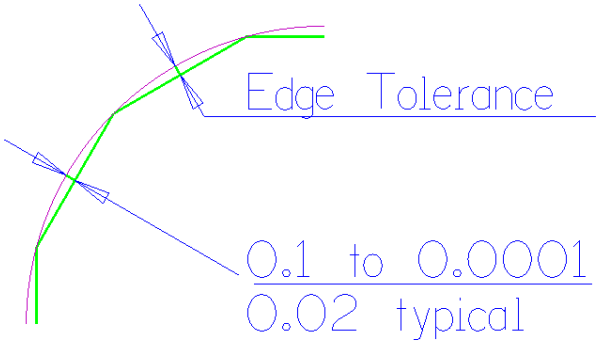


Figure 17 - Edge tolerance explanation

Setting the edge tolerance to 0.1 would be ok for something like a car roof whereas 0.005 would be required for the detail on the end of a 20mm diameter electrode.

Surfaces should not contain sharp corners; if a sharp corner is required then it must be created using two surfaces.

Surface Types

Flat Surface-Definition

A **Flat Surface** is created within a closed boundary that lies in one plane. Its main purpose is to easily create a blanking surface. Flat surfaces should not be used if it is required to be trimmed or filleted.

> The closed boundary must be a polyline or line it cannot be a spline.

Flat Surface Example

Draw a closed shape on screen in flat land.

Select **GEOMETRY > 3D Surfaces** and the surface creation dialogue is displayed. Set the Tolerance to **0.02** and select Flat Surface then **[OK]**.
The system prompts **Pick Closed Geometry to Make Flat Surface <LClick>** on the boundary you have drawn and the surface is created.

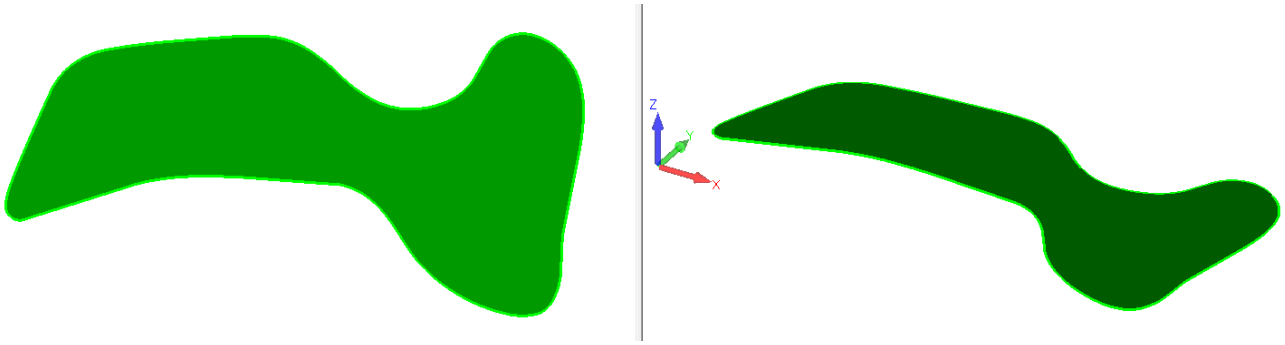


Figure 18 - Flat surface example in shaded mode

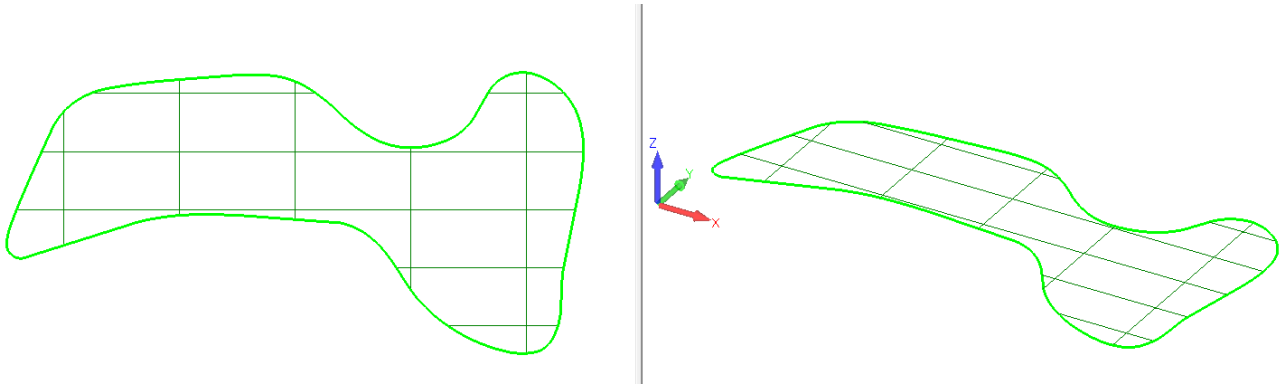


Figure 19 - Flat surface example in wire frame

When the surface is not shaded, the parameter lines are displayed.

Extruded Surface-Definition

The **Extruded Surface** command will create a flat surface for each element in the profile selected. Each surface is created with respect to the Z axis of the plane on which the geometry profile is created.

> The closed boundary must be a polyline or line it cannot be a spline.

Extruded Surface-Example

Using the same profile as use in the flat surface creation Select **GEOMETRY > 3D Surfaces** and the surface creation dialogue is displayed. Set the Tolerance to **0.02** and select ⊙ **Flat Surface** then **[OK]**.

The Extrude Surface Distance dialogue is displayed set the distance to **-20** then **[OK]**.
The system prompts **Pick Geometry to Extrude <LClick>** on the profile and the extruded surfaces are created.

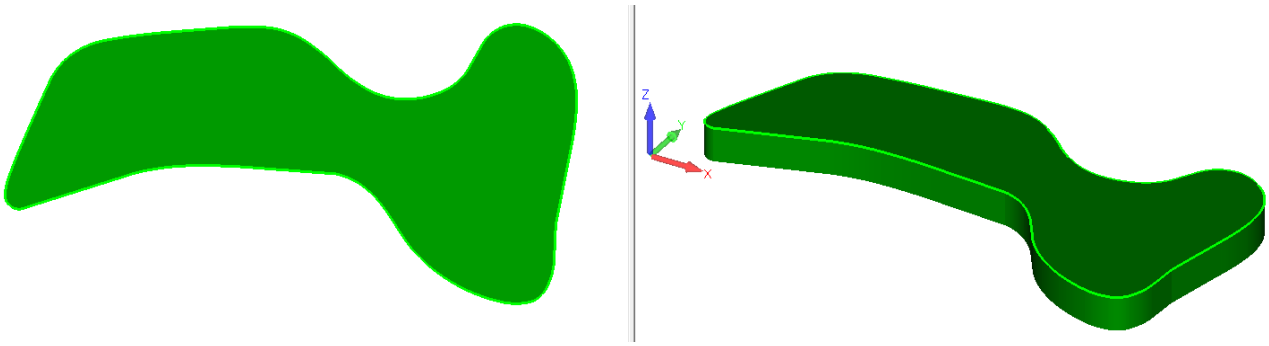


Figure 20 - Extruded surface example in shaded mode

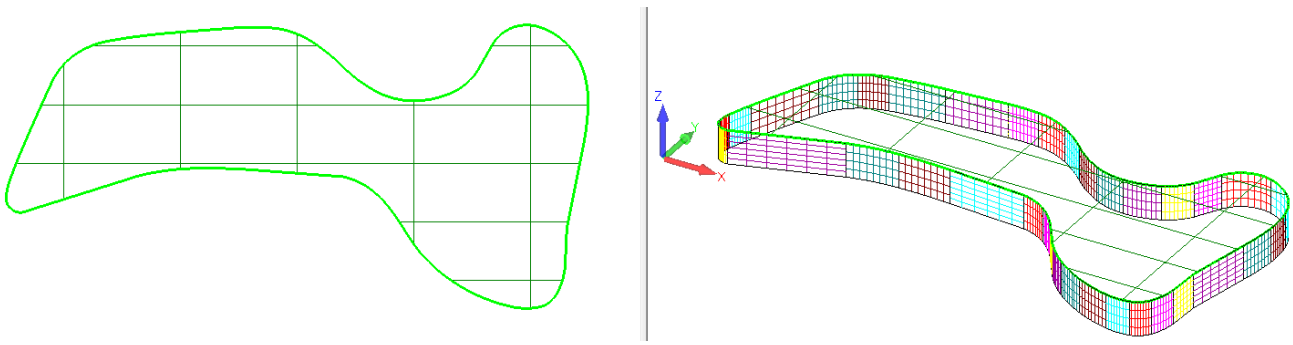


Figure 21 - Extruded surface example in wireframe

When the surface is not shaded, the parameter lines are displayed.

> The extrude edge is made up of as many surfaces as there are elements in the whole profile.

Ruled Surface-Definition

The **Ruled Surface** command will create one surface which is suspended between two edges.
The edge curves can be open or closed; they can lie in different work planes, and be lines, polylines, splines or other another surface edge. The surface is created by connecting the start points of each curve together and the end points of each curve together with a straight line (hence the term Ruled Surface).
If when using open geometries, they are created in different directions that would result in a twisted surface a warning dialogue is displayed which will automatically reverse one of the edges.
When using closed geometries, you need to ensure the start points are perpendicular otherwise the surface will be twisted.

Ruled Surface-Example1

Use the current geometry as the previous examples.

Select **EDIT > Auto 2D 3D > 3D** 

this causes any manipulation to be a 3D manipulation.

Select **EDIT > Copy**  then select the geometry profile.
For the Base Point use **X0, Y0, Z0** then for the new position use **X0, Y0, Z20** and the geometry profile will be created on a work plane 20mm up, **<RClick>** or **[Esc]** the copy command.

Select **GEOMETRY > 3D Surfaces**  and the surface creation dialogue is displayed. Set the Tolerance to **0.02** and select Ruled Surface (2 curves) then **[OK]**.
The system prompts **Pick Start Section <LClick>** on the original profile.
The system prompts **Pick Finish Section <LClick>** on the new profile.
The **Generate Surface** dialogue is displayed requesting the number of steps along first edge; this defines the number of control points on the surface, **[OK]** and use the default value, and the surface is created.

Select **EDIT > Auto 2D 3D > Auto**  to set the manipulation method back to default.

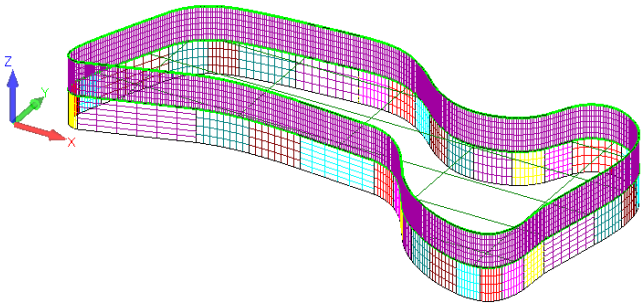


Figure 22 - Ruled surface compared to an Extruded surface

Notice the difference between the extruded surfaces and ruled surface.

Ruled surface Tutored examples

Open the example drawing from the
"…….\ALP TRG 206 Surface Creation 2020\Examples\Drawings\Ruled Surface Geometries"
Set the view to ISO.

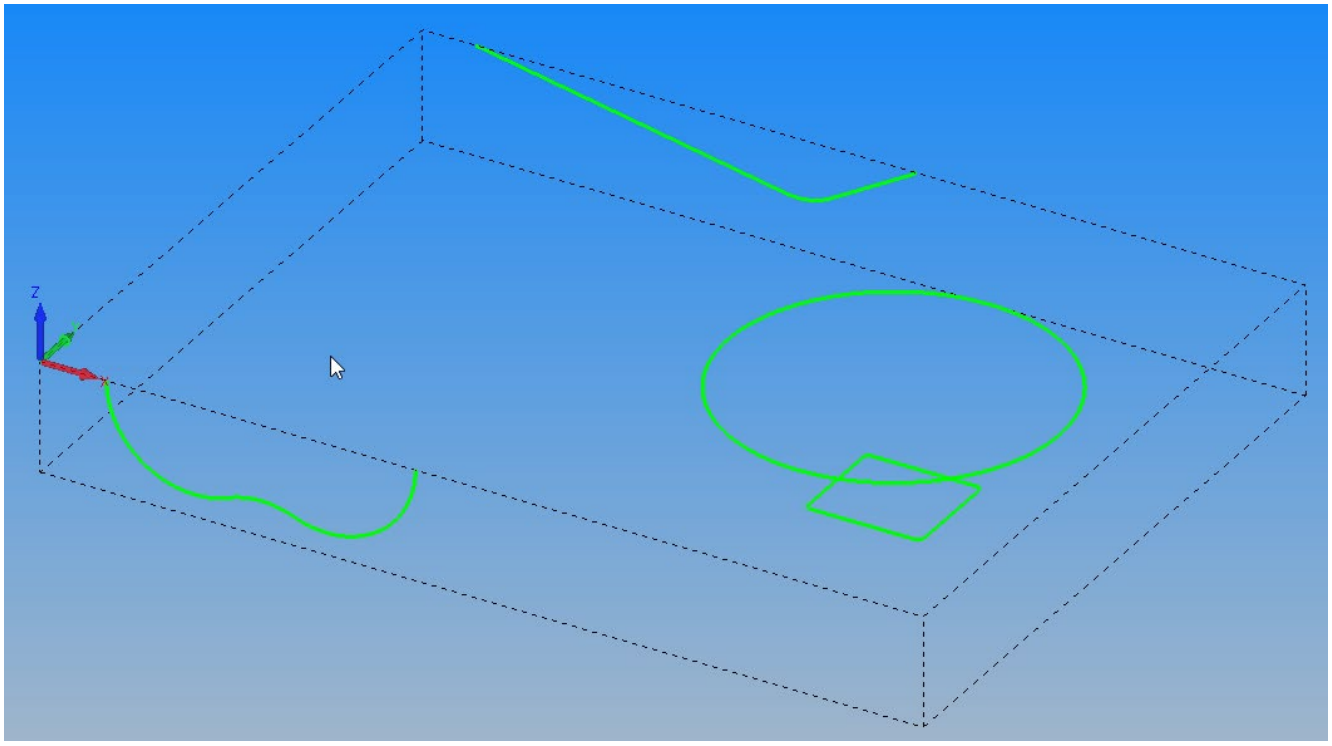
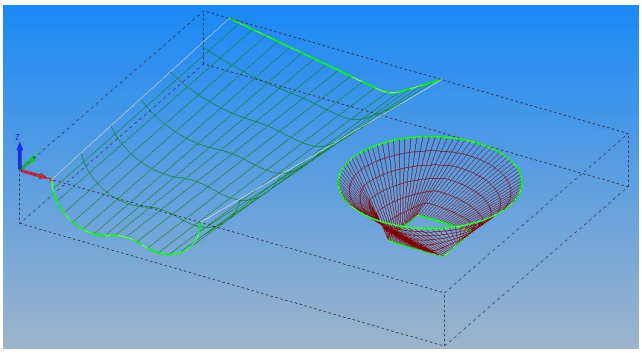


Figure 23 - Ruled surface example geometries

These edges are created in different work planes and will be used to create the surfaces below.



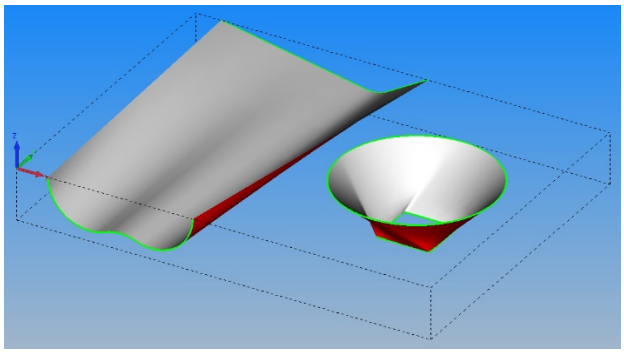


Figure 24 - Ruled surfaces created and viewed in wireframe and shaded

Swept Surface 2 Curves-Definition

The **Swept Surface (2 curves)** command will create a surface by sweeping one curve along and in the direction of the second curve.
A dialogue asks if the curve being swept is to be rotated to keep the section element normal to the curve of sweep. The result will be different depending upon the option selected.

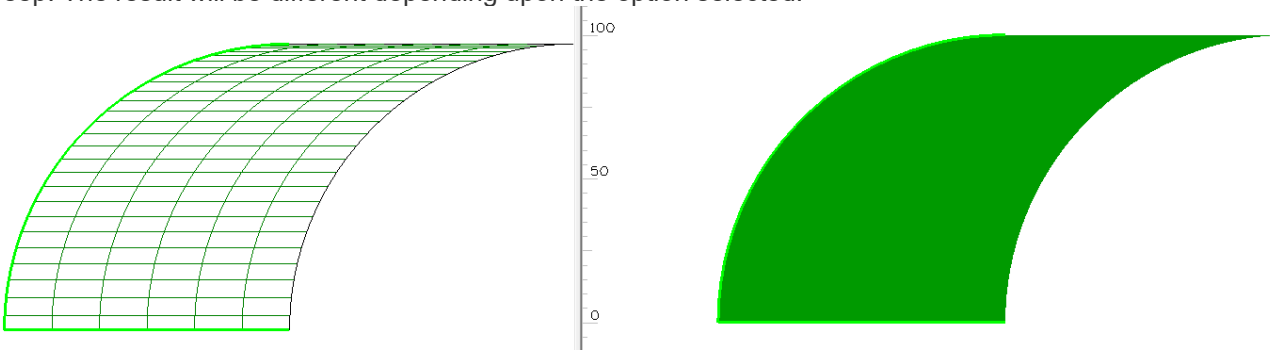


Figure 25 - Straight line sweep without rotation

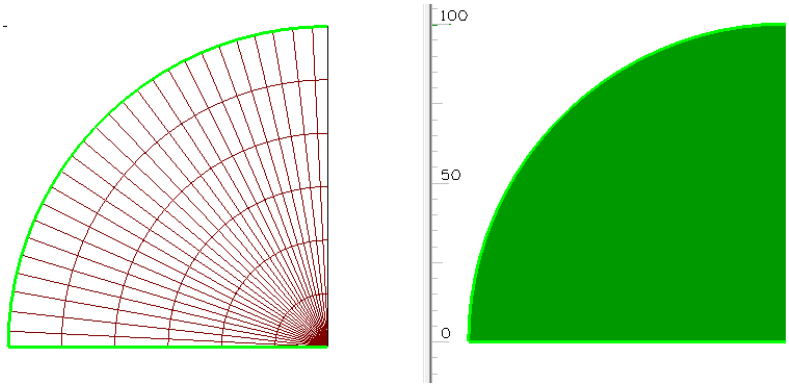


Figure 26 - Straight line sweep with rotation

Care is required with tight bends in the geometry path to ensure they do not cause the edge of the swept surface to turn inside out. (To travel back on itself).

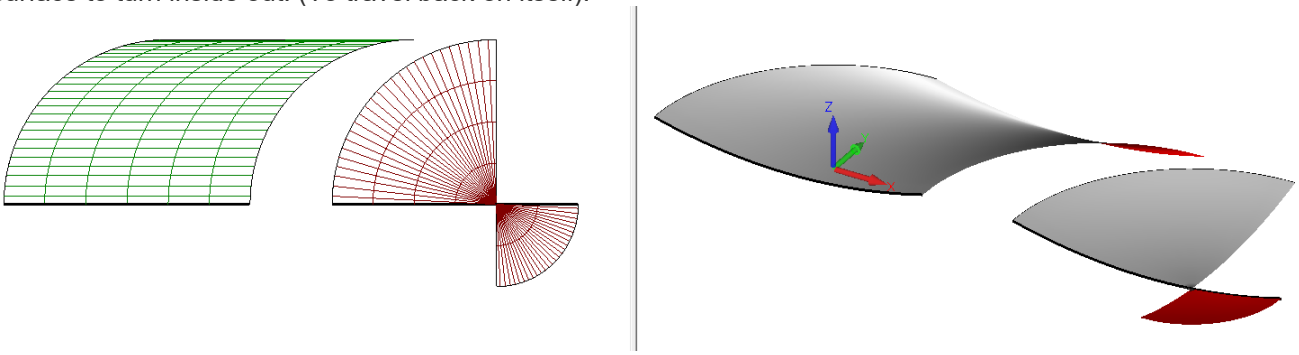


Figure 27 - Swept surface possible errors

Swept Surface 2 Curves Tutored Example

Open the example drawing from the
“…….\ALP TRG 206 Surface Creation 2020\Examples\Drawings\Swept surface 2 Curves”

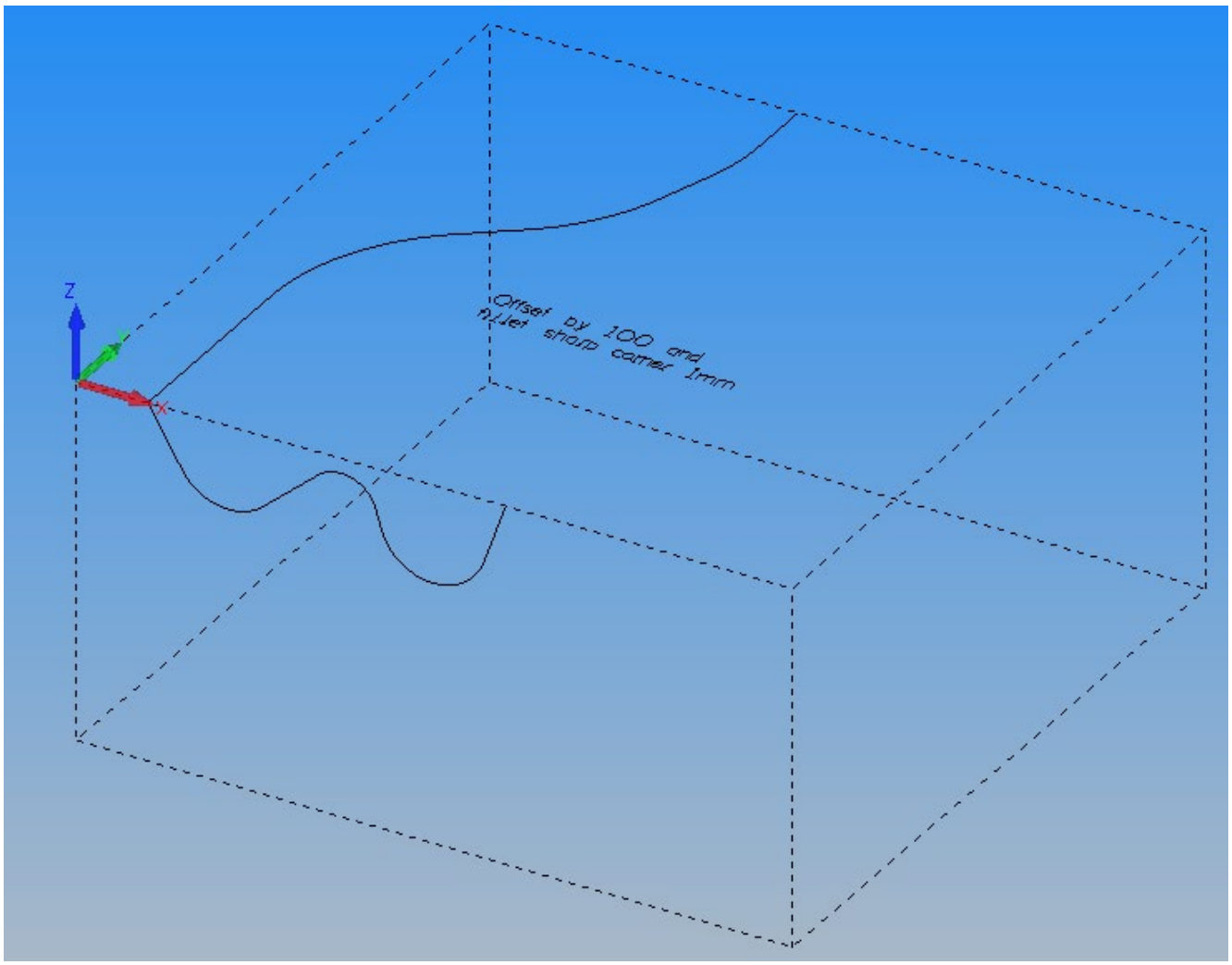


Figure 28 - Swept Surface 2 Curves base geometry

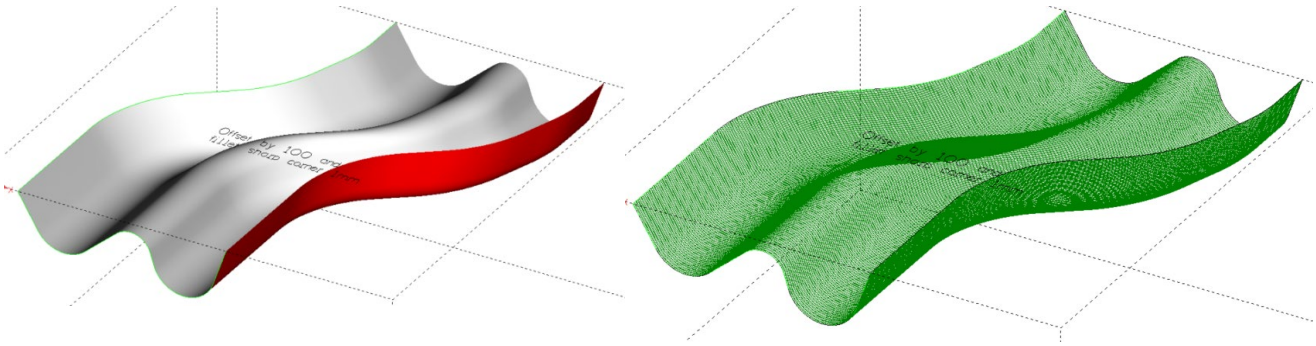


Figure 29 - Swept Surface 2 Curves completed examples

Swept Surface 3 Curves-Definition

A defined edge (Curve1) is swept along a defined geometry path (Curve2) changing form into another defined edge (Curve 3).
Care is required with tight bends in the geometry path to ensure they do not cause the edge of the swept surface to turn inside out, (To travel back on itself).

Swept Surface 3 Curves tutored example

Open the example drawing from the
“…….\ALP TRG 206 Surface Creation 2020\Examples\Drawings\Swept surface 3 Curves”

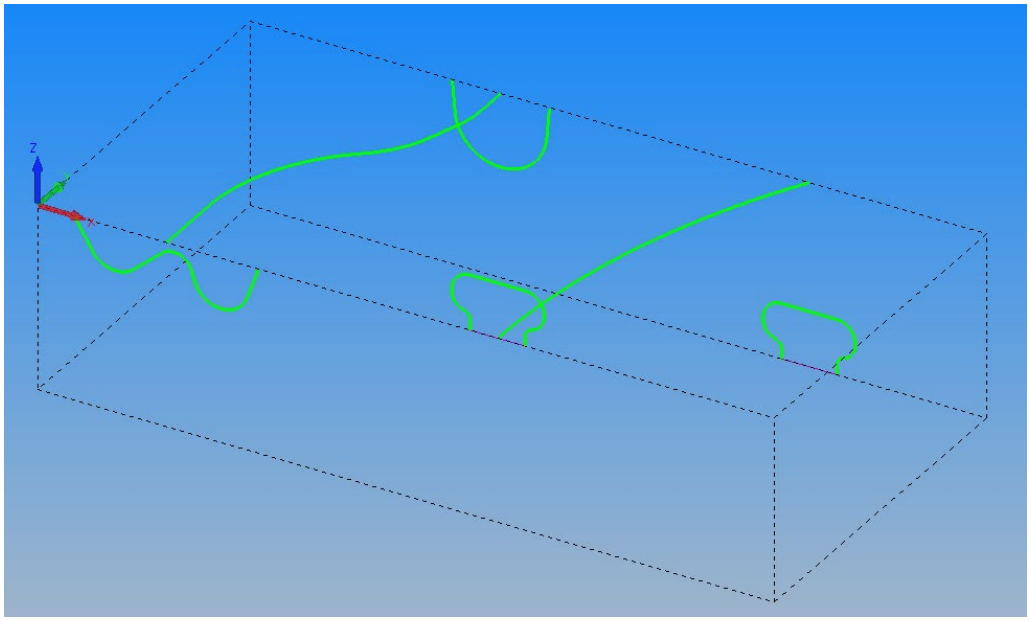


Figure 30 - Swept Surface 3 Curves base drawing

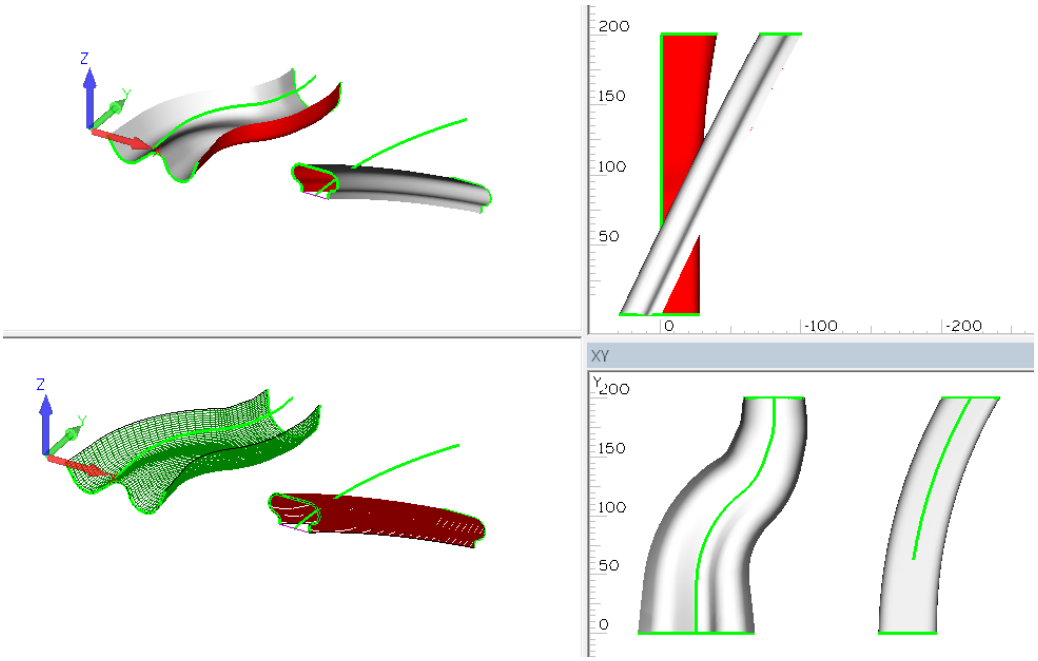


Figure 31 - Swept Surface 3 Curves completed examples

Surface of Revolution-Definition

A Revolved surface is the surface generated when a Profile is revolved about an axis.

> The profile MUST NOT cross the axis line.

Revolve Surface tutored example

Open the example drawing from the
“…….\ALP TRG 206 Surface Creation 2020\Examples\Drawings\Revolved Geometries”

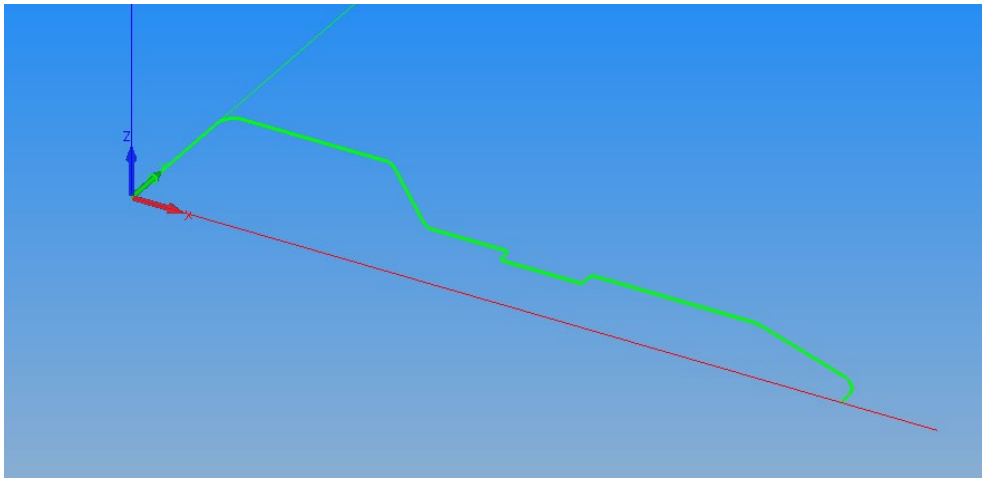


Figure 32 - Surface of Revolution base drawing

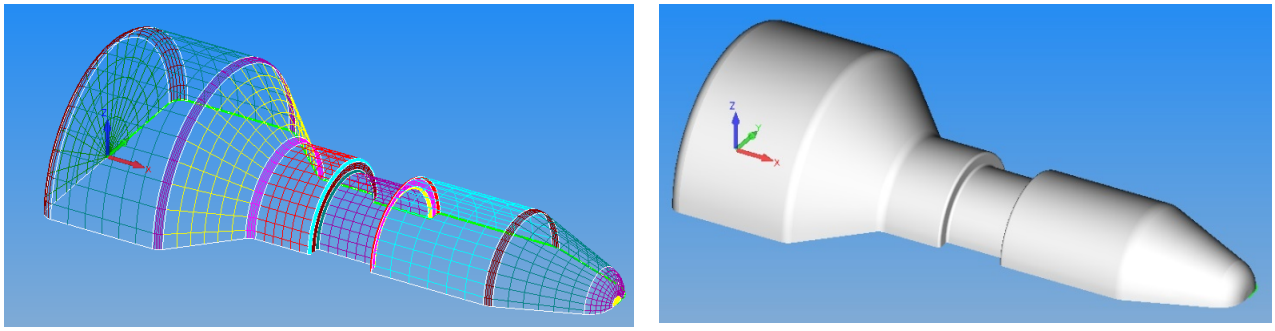


Figure 33 - Surface of Revolution completed examples

Rotation directions

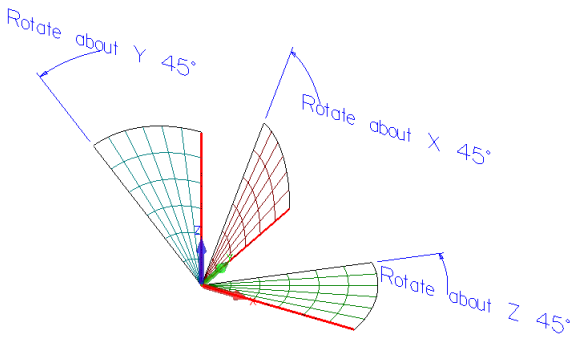


Figure 34 - Example of rotation angles

Coons Patch-Definition

A Coons Patch is the surface generated from 3 or 4 edges.

> The edges can lie in different work planes, but they **MUST** be coincident at the ends.

Coons Patch tutored example

Open the example drawing from the

“…….\ALP TRG 206 Surface Creation 2020\Examples\Drawings\Coons Patch Edges”

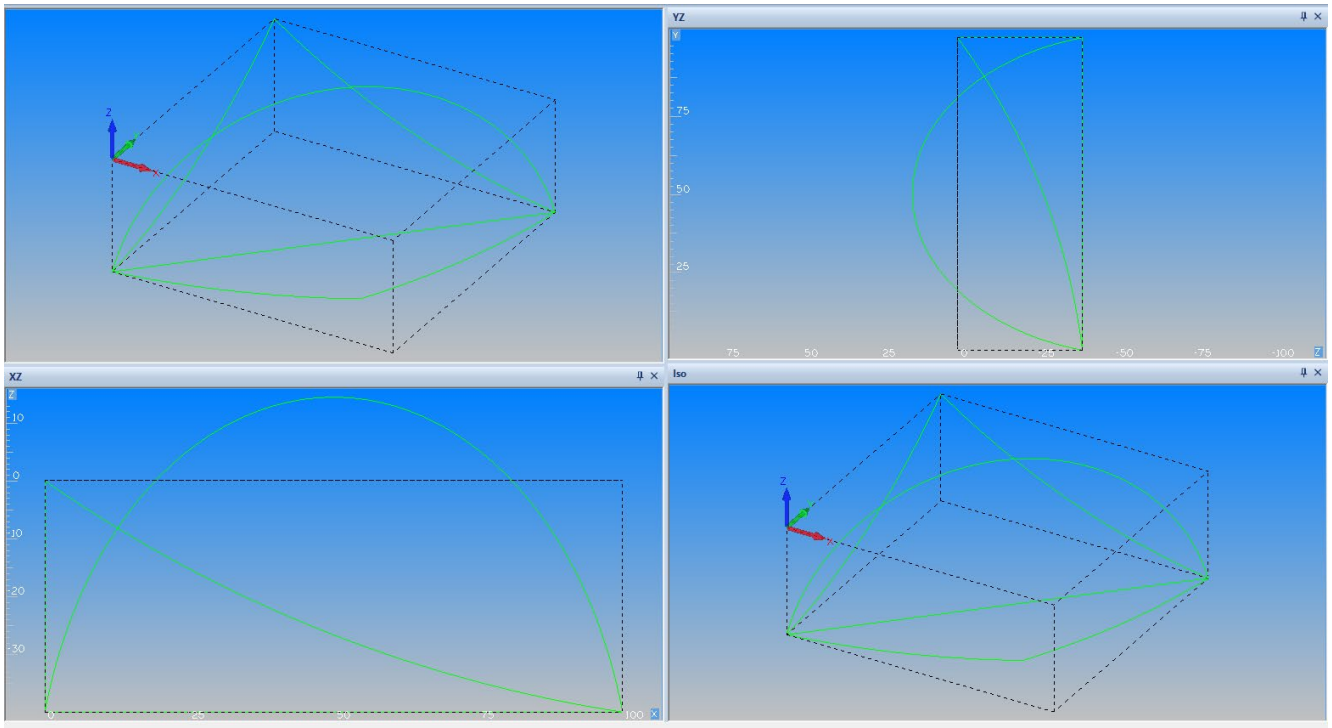


Figure 35 - Coons Patch example drawing

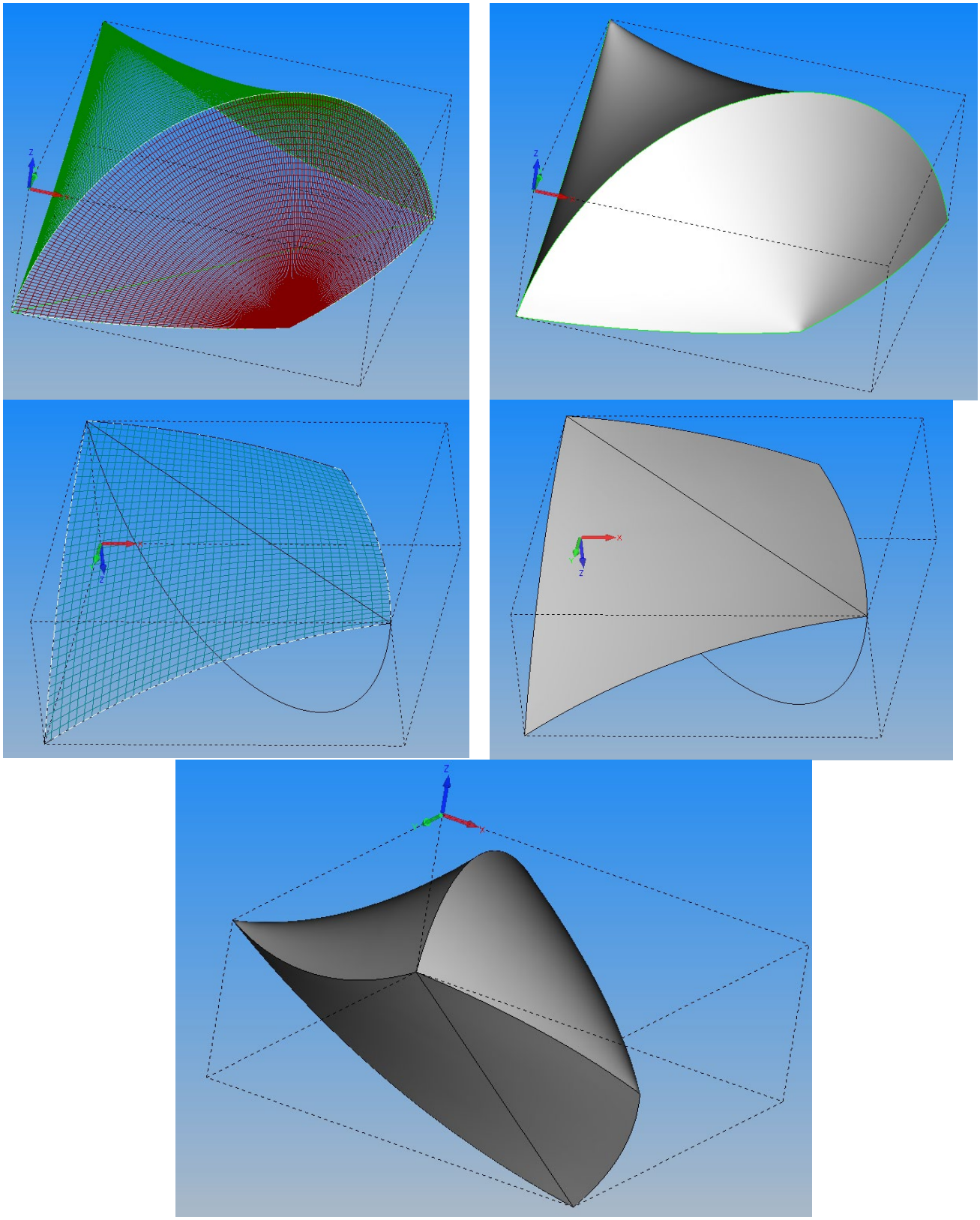


Figure 36 - Coons patch completed examples

Fillet and Trim Surface-Definition

It is possible to fillet and trim intersecting surfaces.
When fillets are applied to open ended surfaces, the fillet radius can be different at the start and end. If the fillet is a closed fillet the start and end radius MUST be set the same.
Fillets are always produced between the tool sides (GREY) of the surface.

Fillet and Trim Surface Tutored Example

Open the example drawing from the
“…….\ALP TRG 206 Surface Creation 2020\Examples\Drawings\Fillet surfaces 1”

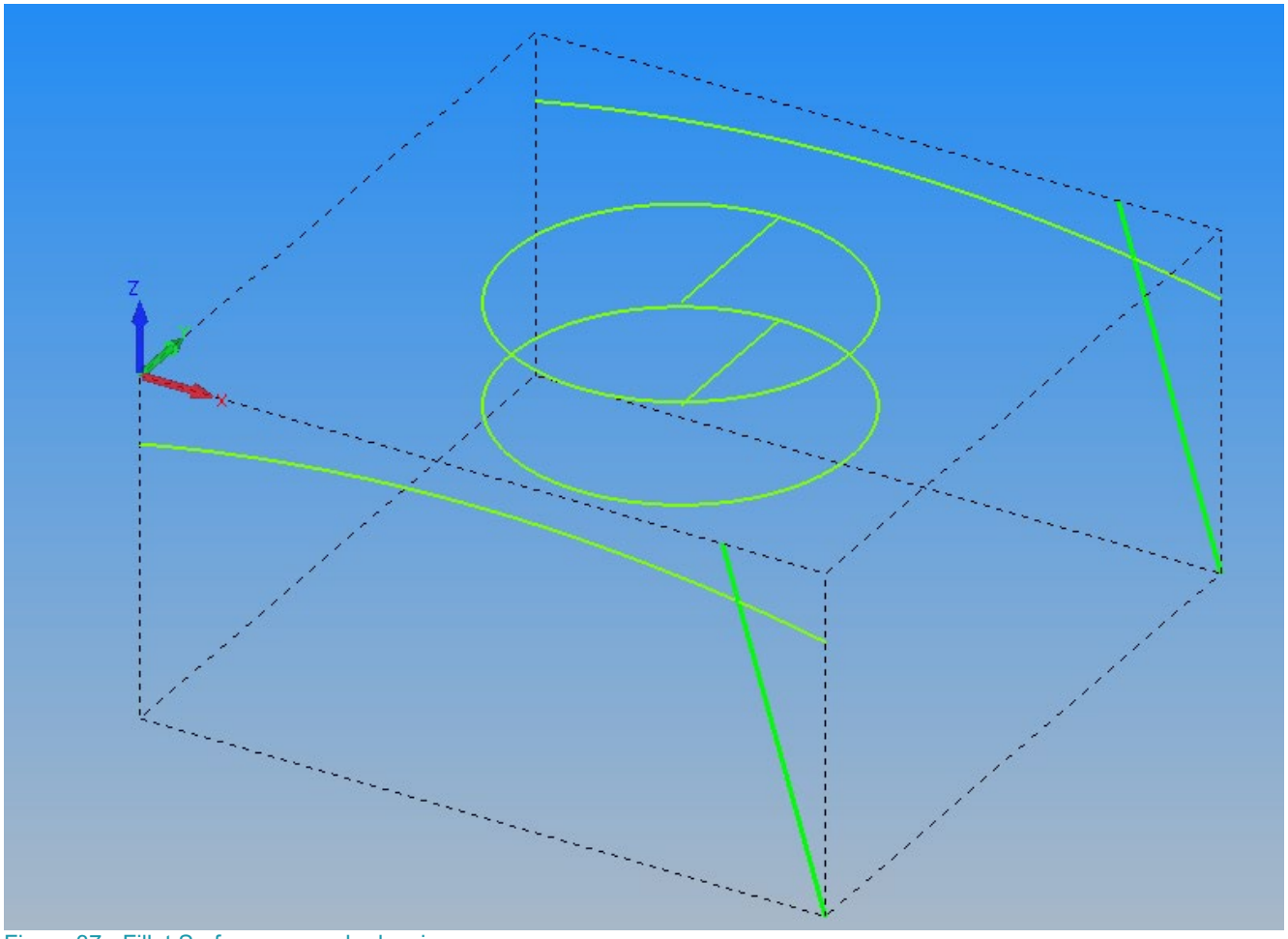


Figure 37 - Fillet Surfaces example drawing

Introduction to surfacing Jobs

To create the first example, it is first necessary to recognise the types of surfaces required. This will depend upon the machining strategy that is to be used. For example, if a surface were to be machined on the top of a block it would be practical to define the surface such that it overlapped the edges of the actual material. In some cases, where the surface has to be created much bigger than the area to be machined, the machining can be restricted within a boundary.

When working with work volumes and work planes, remember that the geometry manipulation commands **Move, Copy, Rotate, and Mirror** … work differently.

When a work plane is active the manipulation commands work as normal 2D commands. The manipulation takes place within the work plane.

When a work plane is NOT active the manipulation commands work as 3D commands. The manipulation takes place within the work volume.

Example 1 Tutorial

Reference Drawing

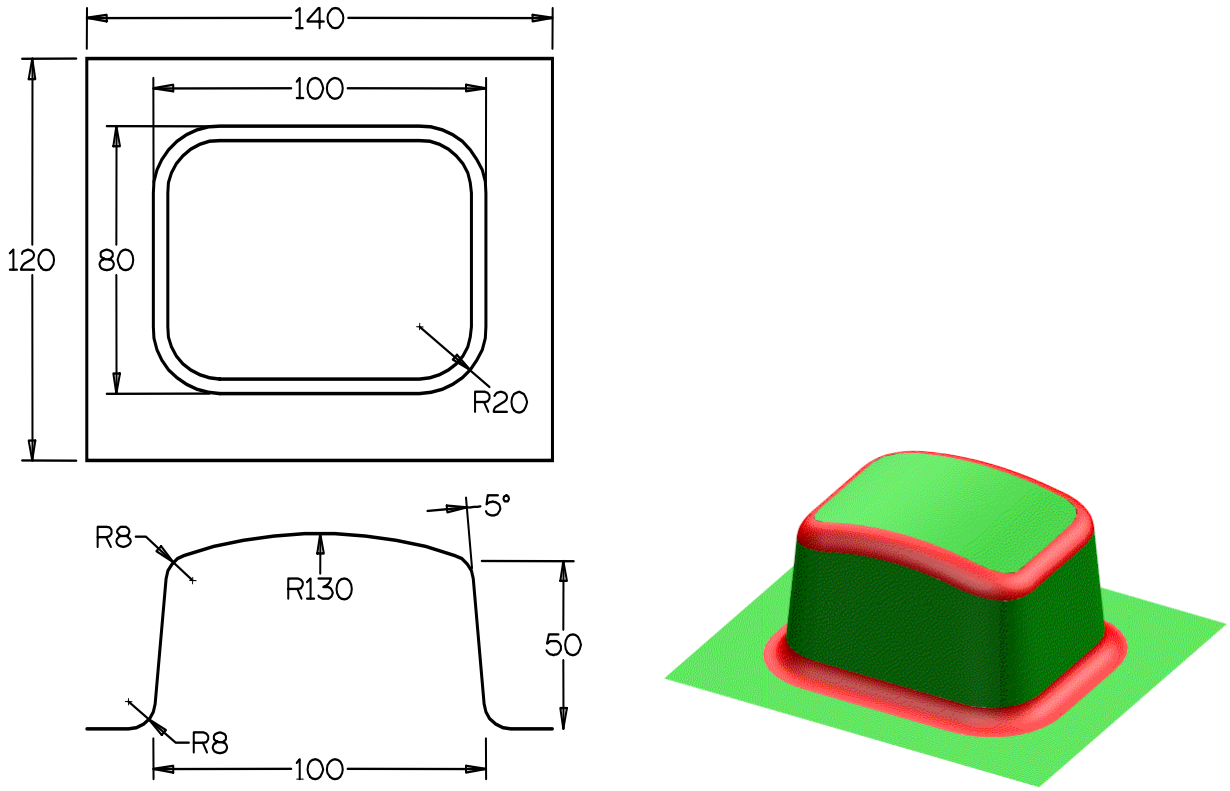


Figure 38 - Worked example reference drawing and completed example

From the drawing you have to decide how many and what type of surfaces need to be defined in order to generate the surface model.

This can be made up from either 3 ruled surfaces, 3 swept surface 2 curves, or a combination of ruled and swept together with 2 fillets.

These notes will create the surfaces as ruled surfaces.

Note that in this image the surfaces are shown in green to make it easier to see on the page. In Alphacam the surfaces are coloured either Silver (external face) or Red (internal face.)

Base Geometry Creation

Using **GEOMETRY > Rectangle** draw a rectangle from **X0, Y0** to **X140, Y120.**

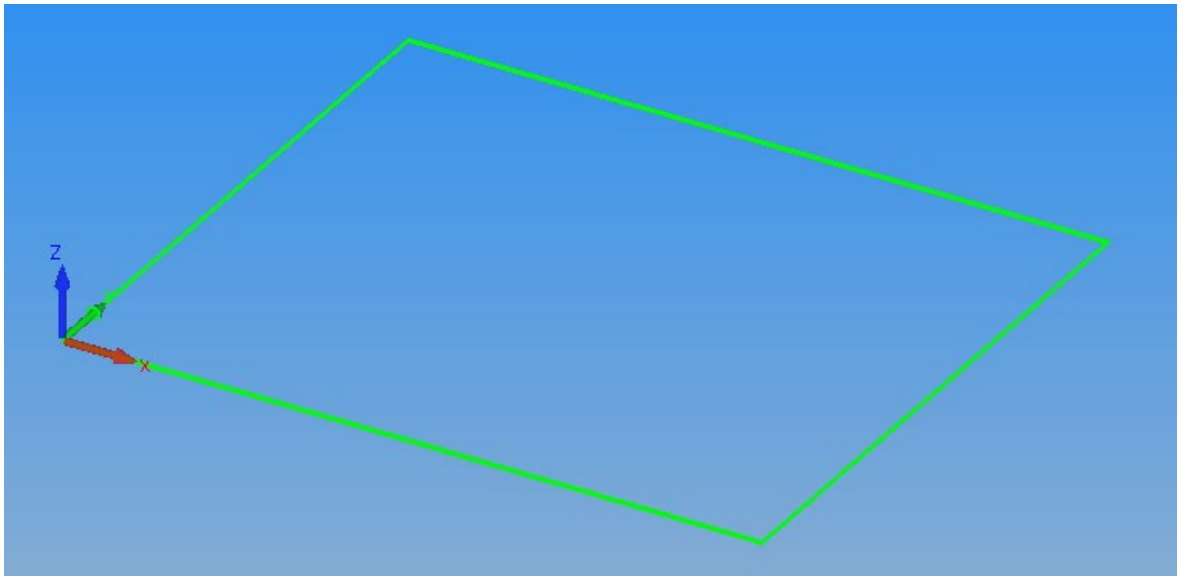


Figure 39 - Example 1 starting rectangle

Select **3D > Set Work Volume**

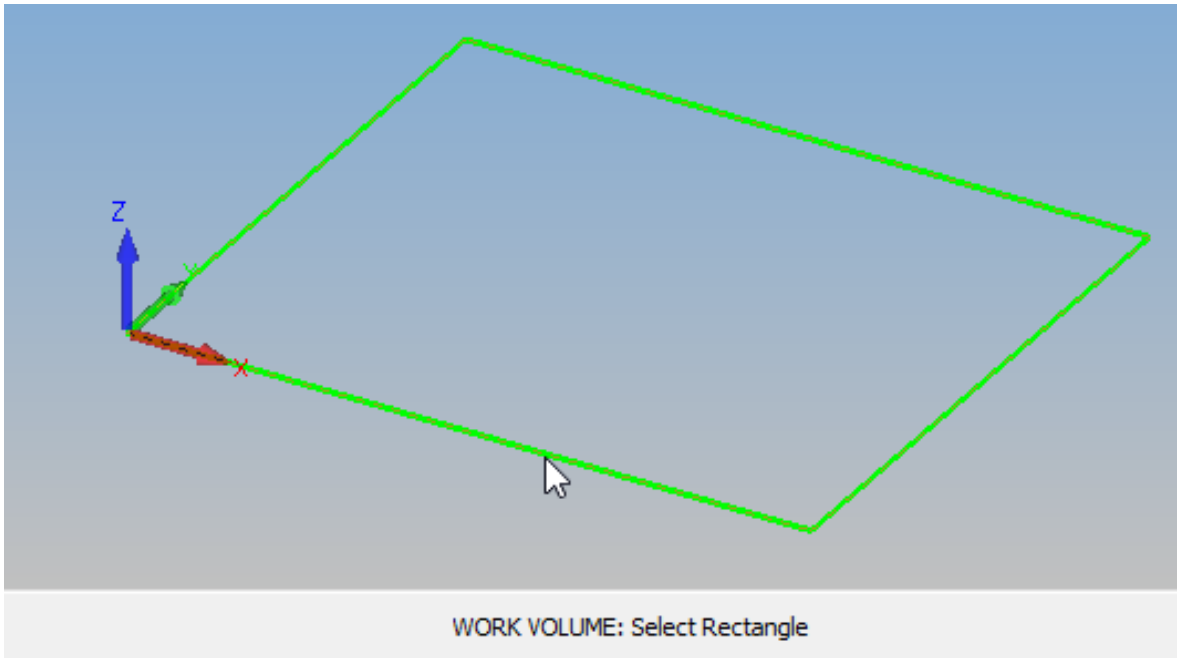


Figure 40 - Work Volume selection

Using the rectangle set the work volume top to **0** Bottom to **-70.**

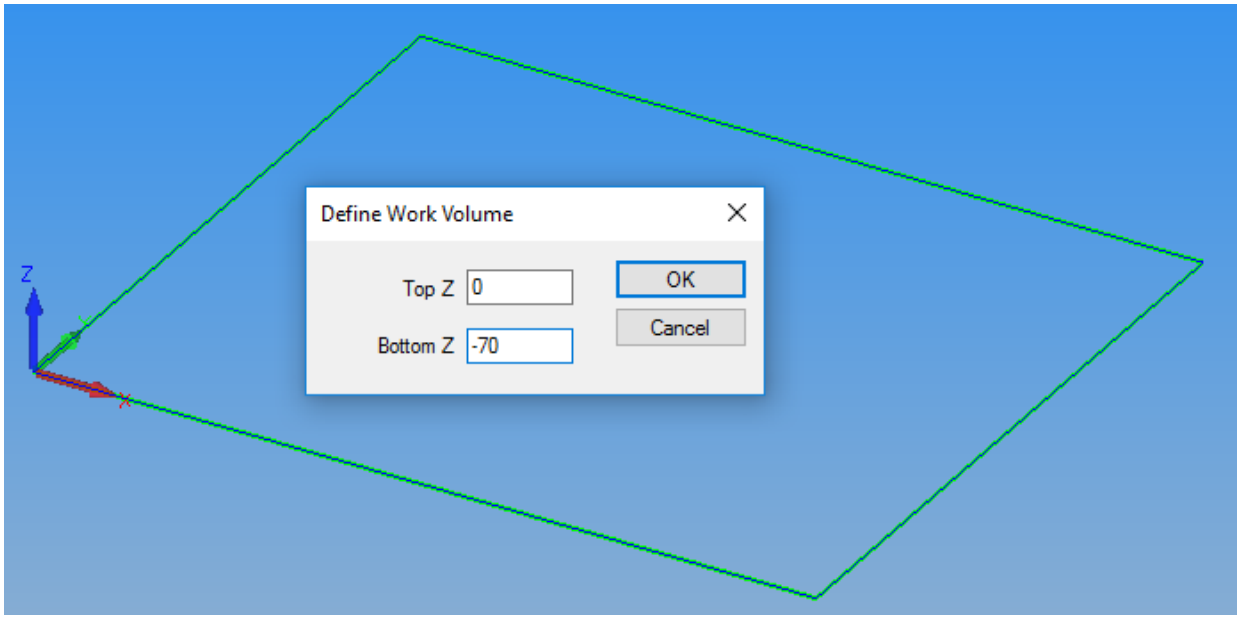


Figure 41 - Setting the Work Volume levels

<LClick> [OK] to finish.

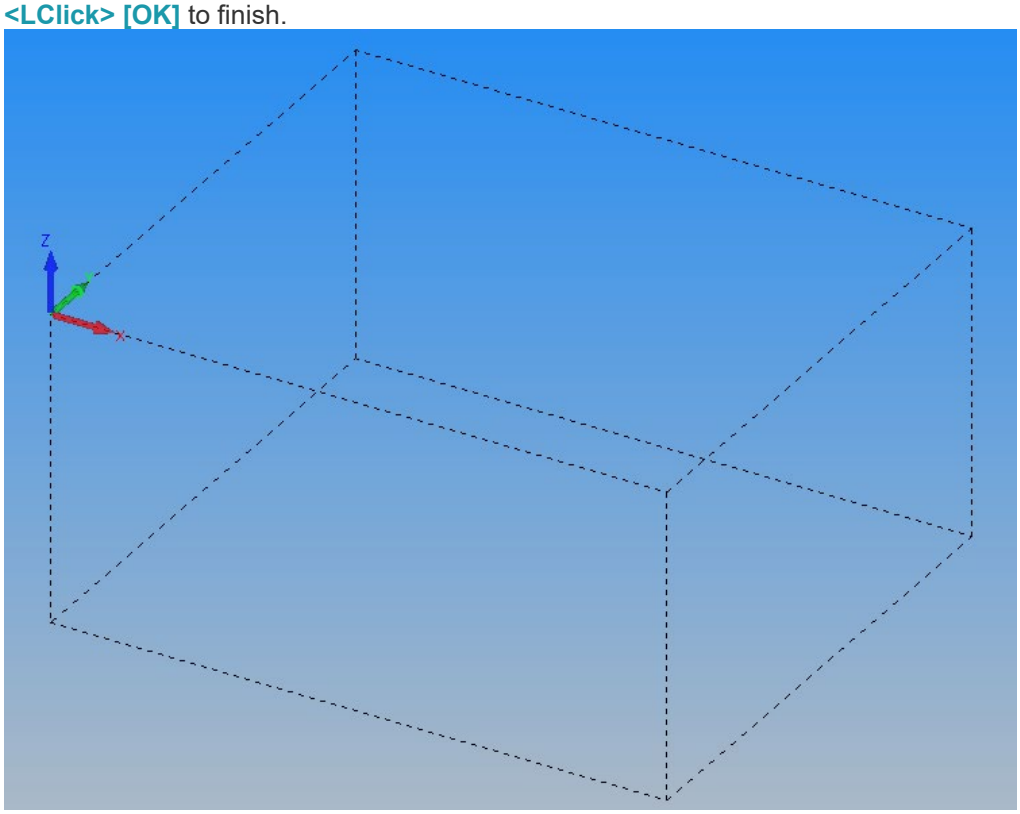


Figure 42 - Completed Work Volume

Select **VIEW > XZ**  .

Figure 43 - Work Volume viewed in XZ

Base Geometry for The Taper

To set the bottom of the work volume as the current plane.

Use **WORK PLANES > Slice through Work Volume** and pick the bottom line of the Work Volume.

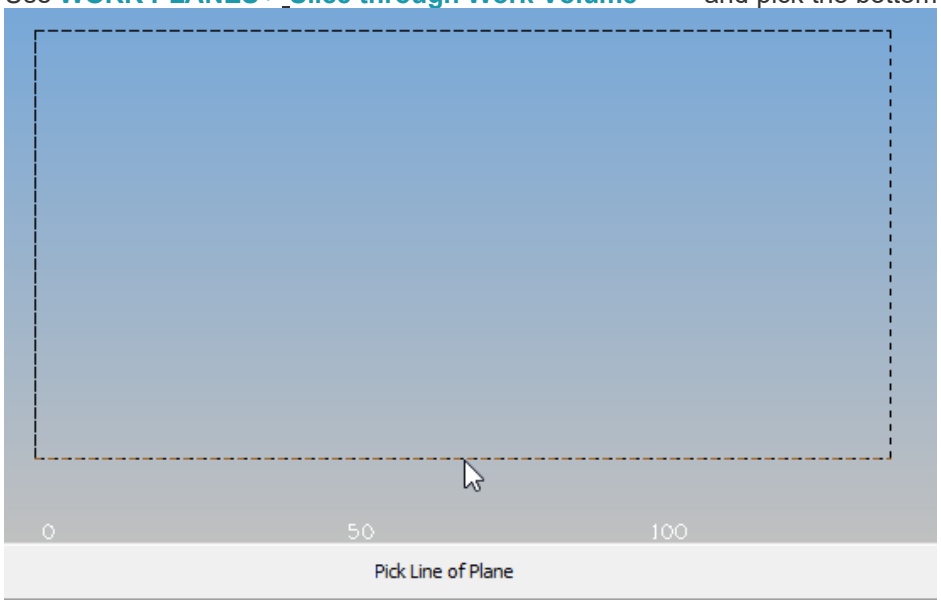


Figure 44 - Creating the bottom work plane

Select **Iso view**

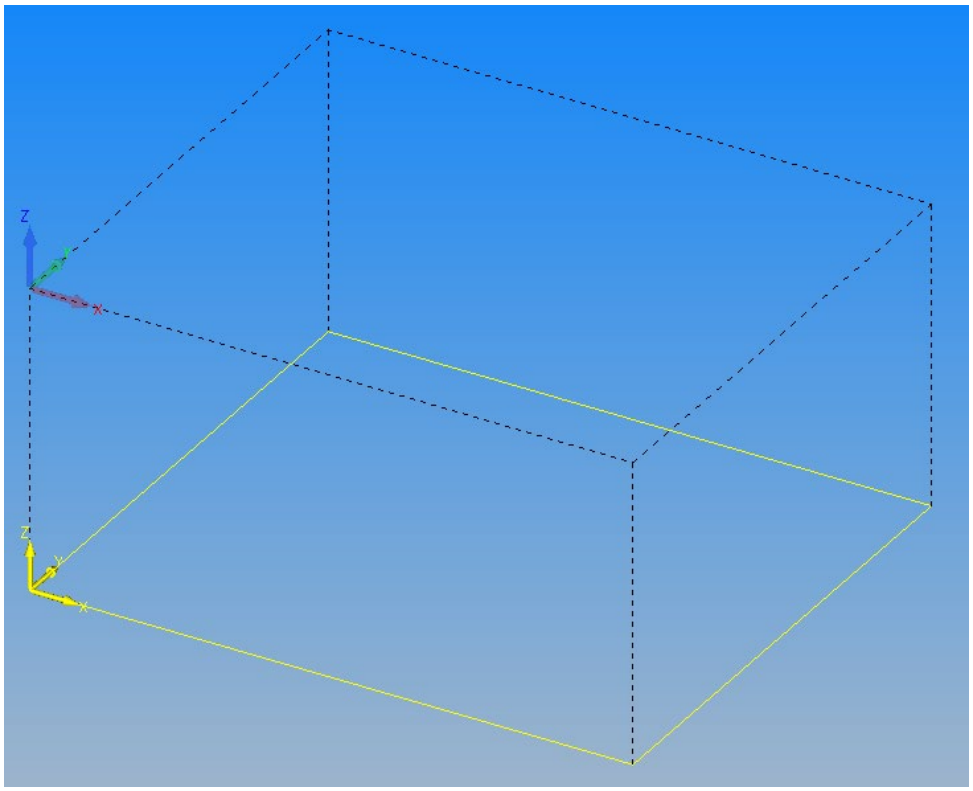


Figure 45 - Bottom work plane in ISO

Using **GEOMETRY > Rectangle**  draw a rectangle from **20,20** to **120,100.**

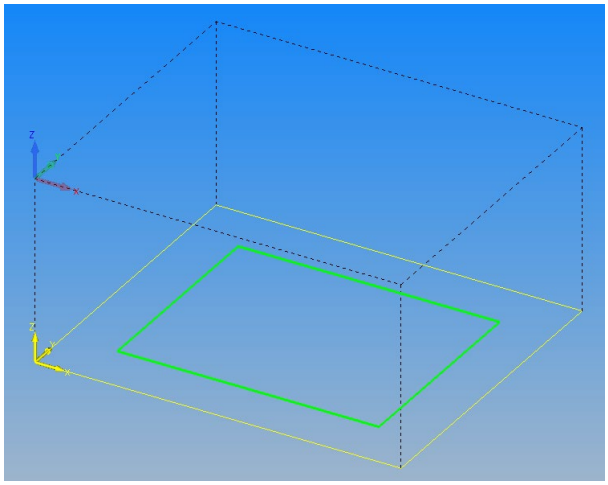


Figure 46 - Base rectangle

Use **EDIT > Fillet**  and apply the option to all corners of the rectangle.

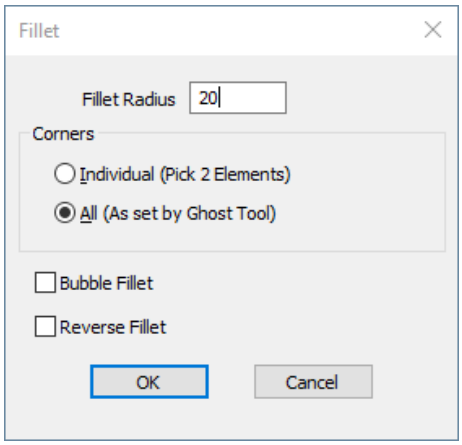


Figure 47 - Fillet settings

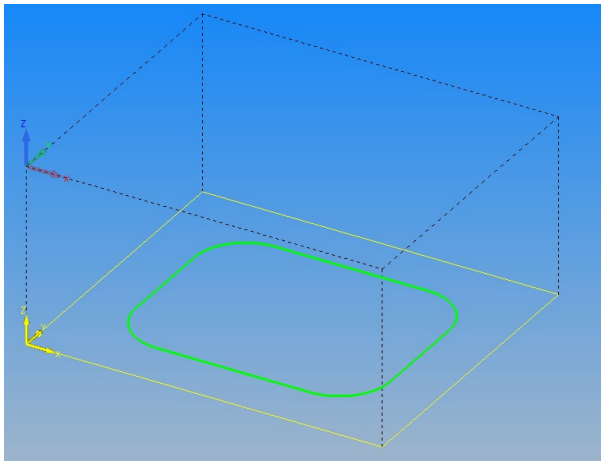


Figure 48 - Completed base rectangle

Using EDIT > Offset apply the following values to create an internal offset.

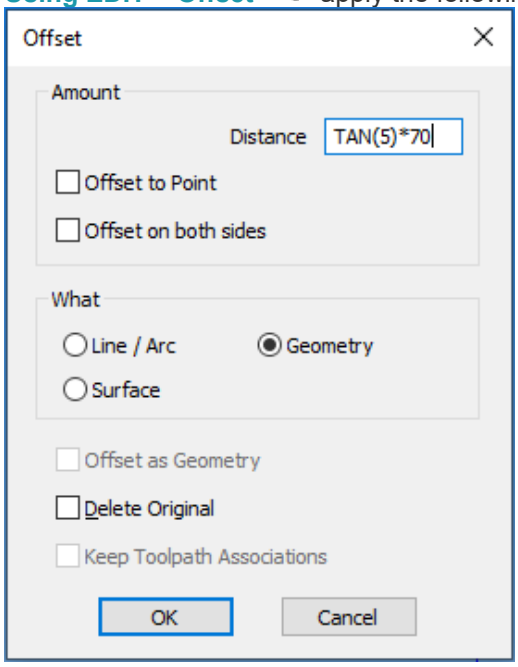


Figure 49 - Internal offset values

This will provide geometry, that when positioned on the top of the work volume, 70mm above the current rectangle location, will allow the creation of a ruled surface forming the 5° taper.
<LClick> [OK] to continue, <LClick> the rectangle to select it.

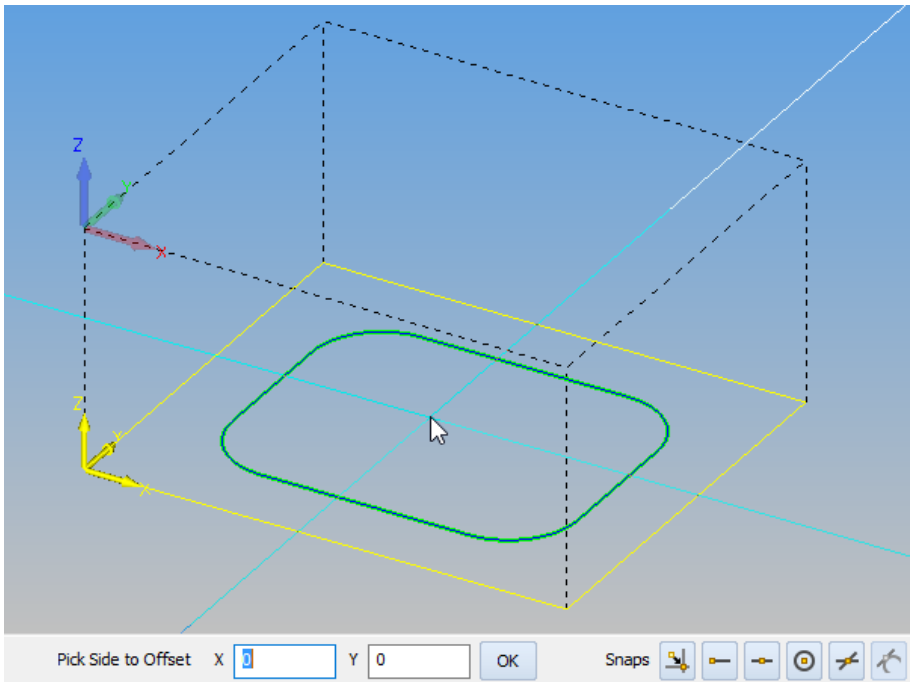


Figure 50 - Apply the Offset to the inside

<LClick> inside to place the new geometry.
<RClick> to finish.

Select **WORK PLANES > Cancel work plane** .

We now need to move the inner geometry from its current location to the correct position 70mm above where it is. To do this we must force a 3D move on the geometry which is currently only a 2D item.

Use **EDIT > Auto/2D/3D > 3D** to set the drawing mode to 3D.

Select **EDIT > Move** .

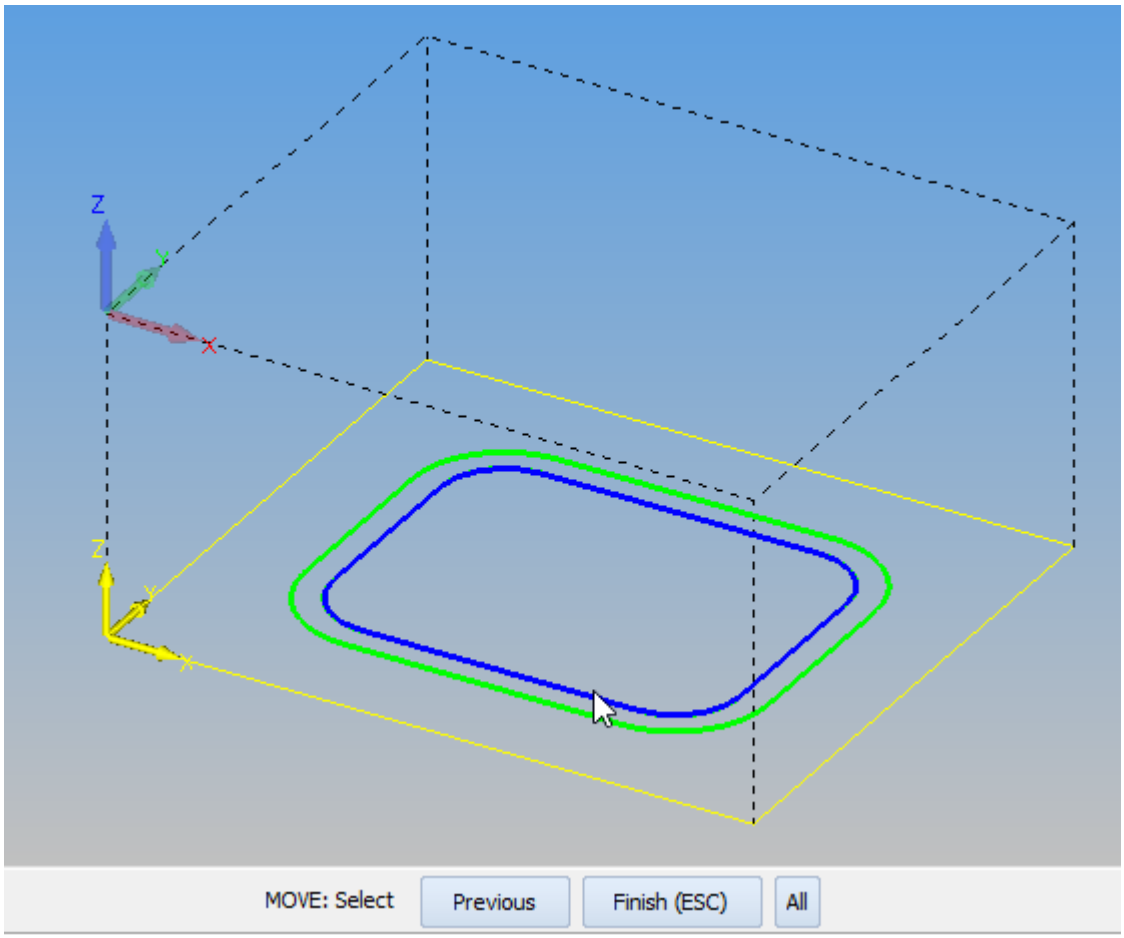


Figure 51 - Select the item to move

<LClick> the inner shape as the item to Move.

<RClick> or **[Finish (ESC)]** to continue.

For the base point use **X0, Y0, Z-70**,

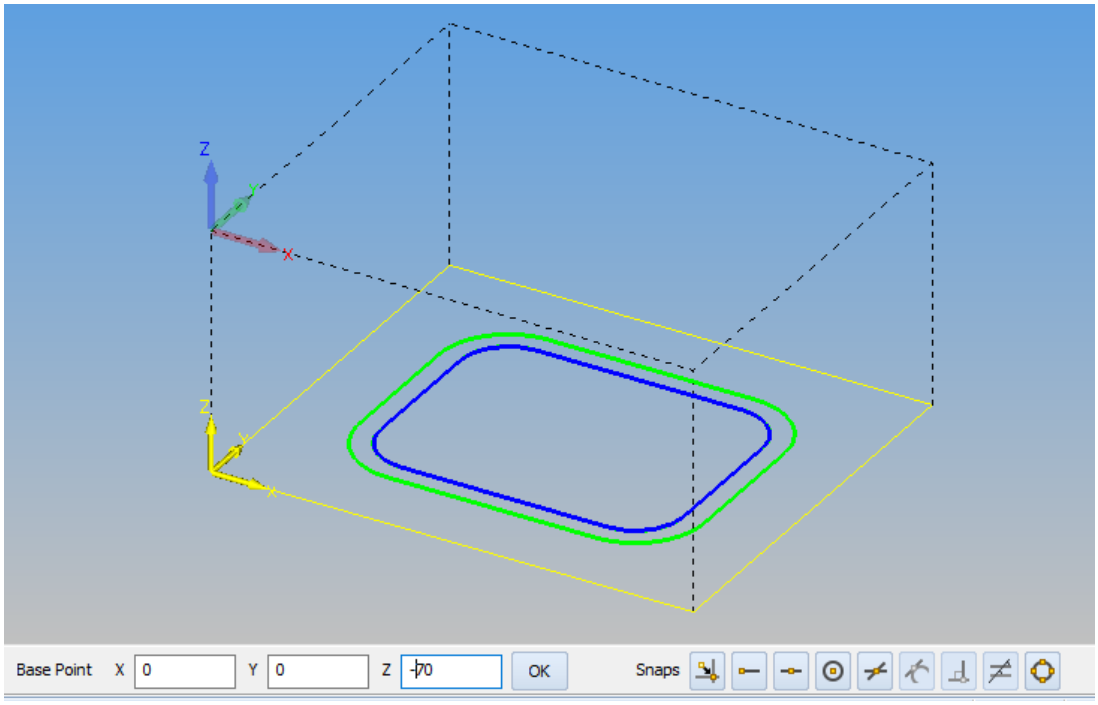


Figure 52 - Base Point selection

For the new location, set **X0, Y0, Z0**

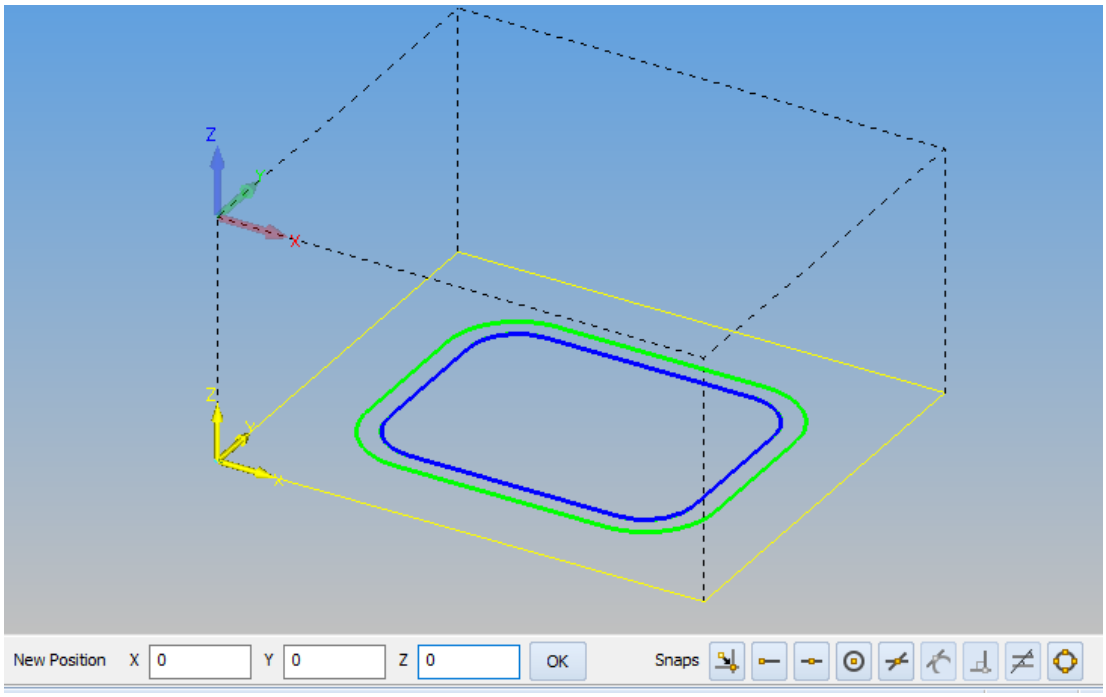


Figure 53 - New location point

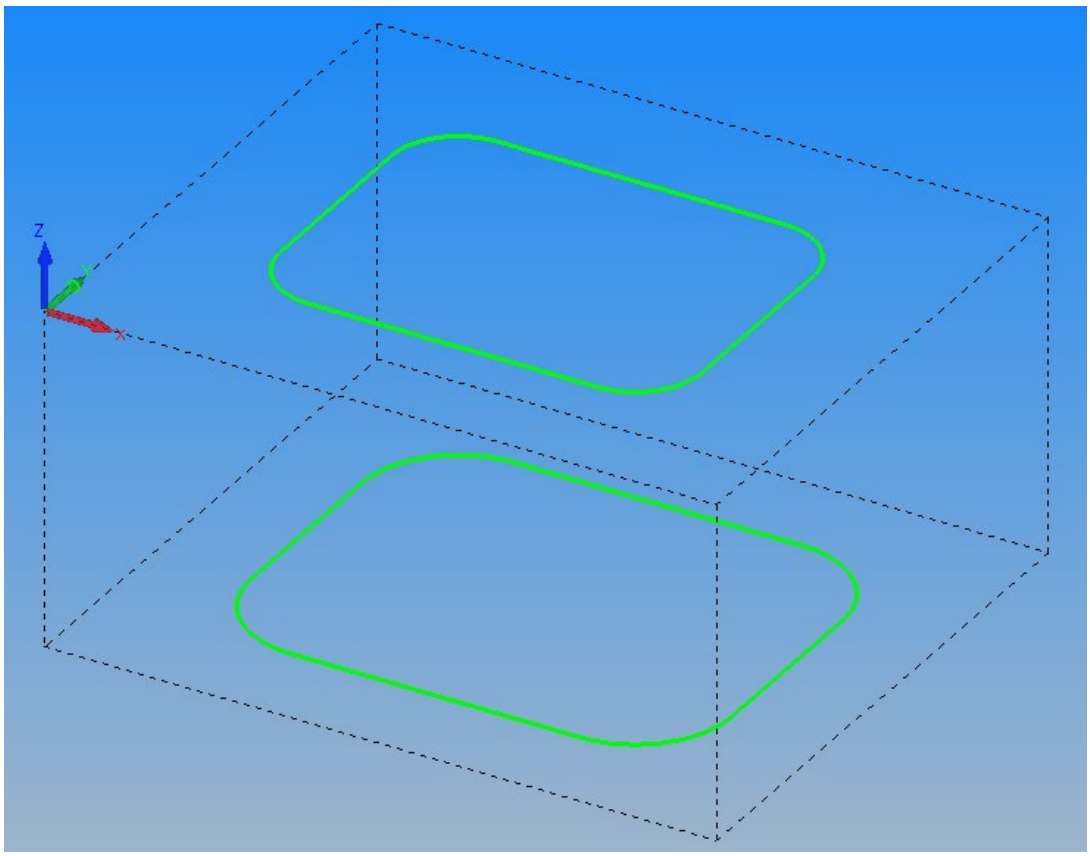


Figure 54 - Completed Move

Taper Surface Creation

Select **GEOMETRY > 3D Surfaces**

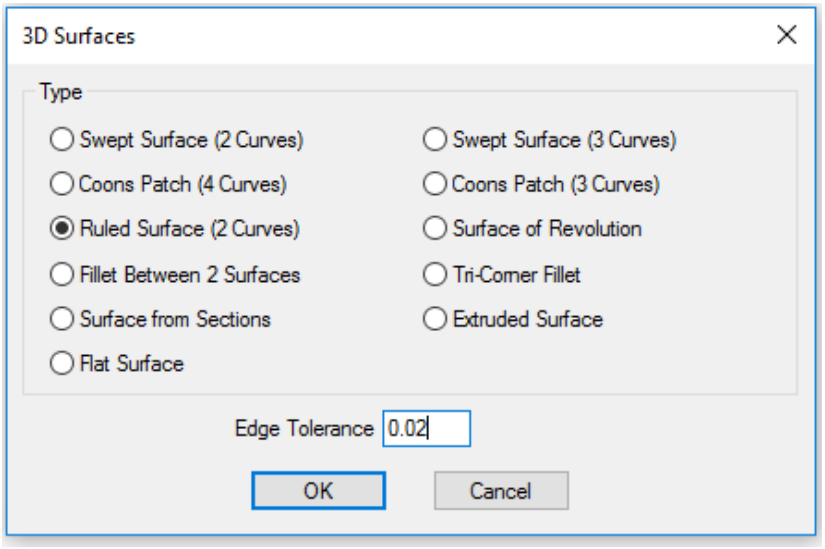


Figure 55 - Ruled Surface option

Set the option to ⊙ **Ruled Surface (2 curves**
<LClick> [OK] to continue.
<LClick> the bottom geometry as the Start Section.

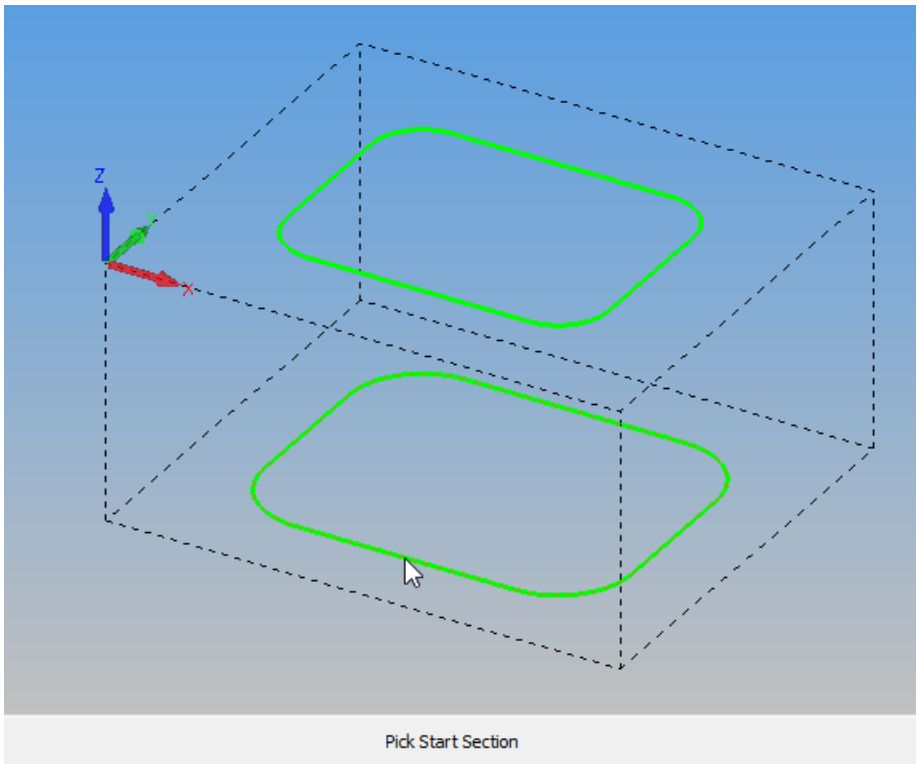


Figure 56 - Start Section Geometry

<LClick> the top geometry as the Finish Section.

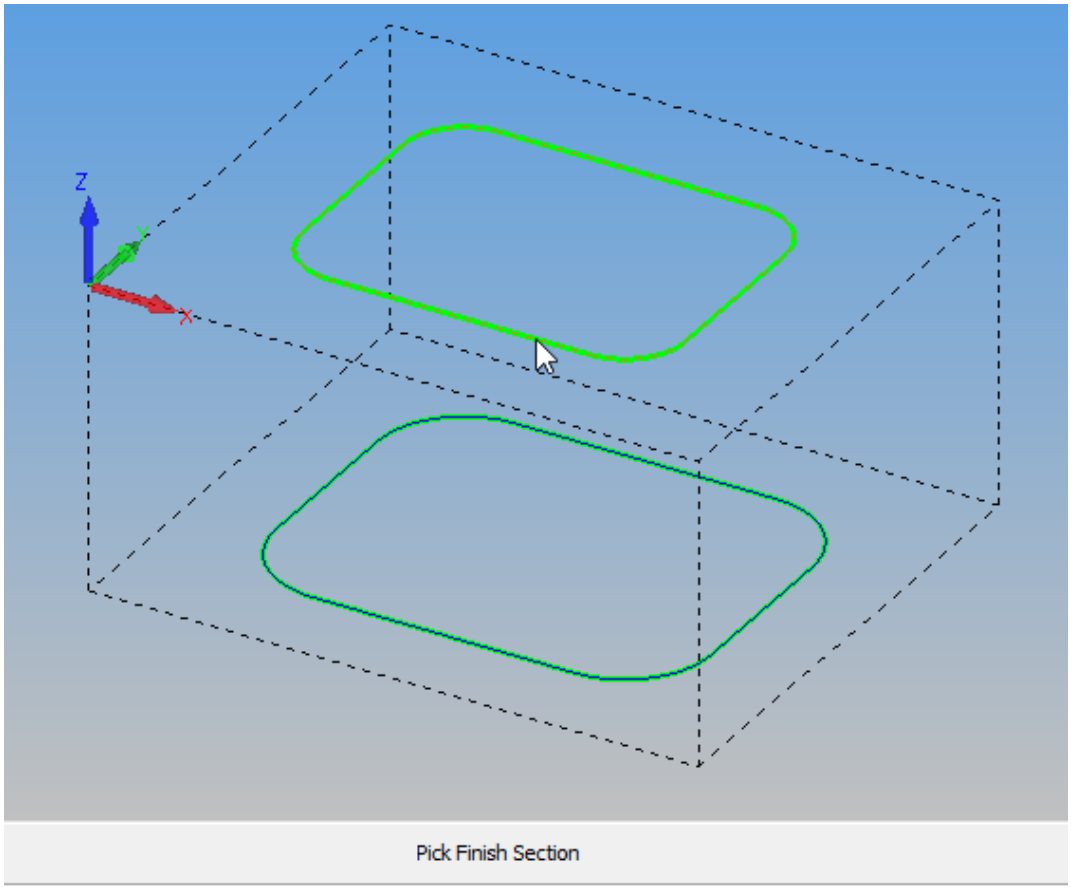


Figure 57 - Finish Section Geometry

You are then presented with a requirement for the **Number of Steps along First Edge**.

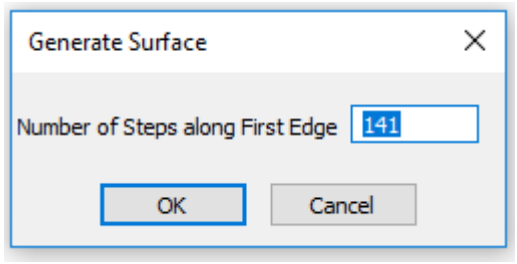


Figure 58 - Surface precision option

This can be thought of as how good the surface will be created.
The higher the value in the dialogue, the more accurate the surface will be created, this could have an impact of complex parts when they are machined and the final finish of the created part.

For this example, we will accept the default values.

<LClick>[OK] to continue

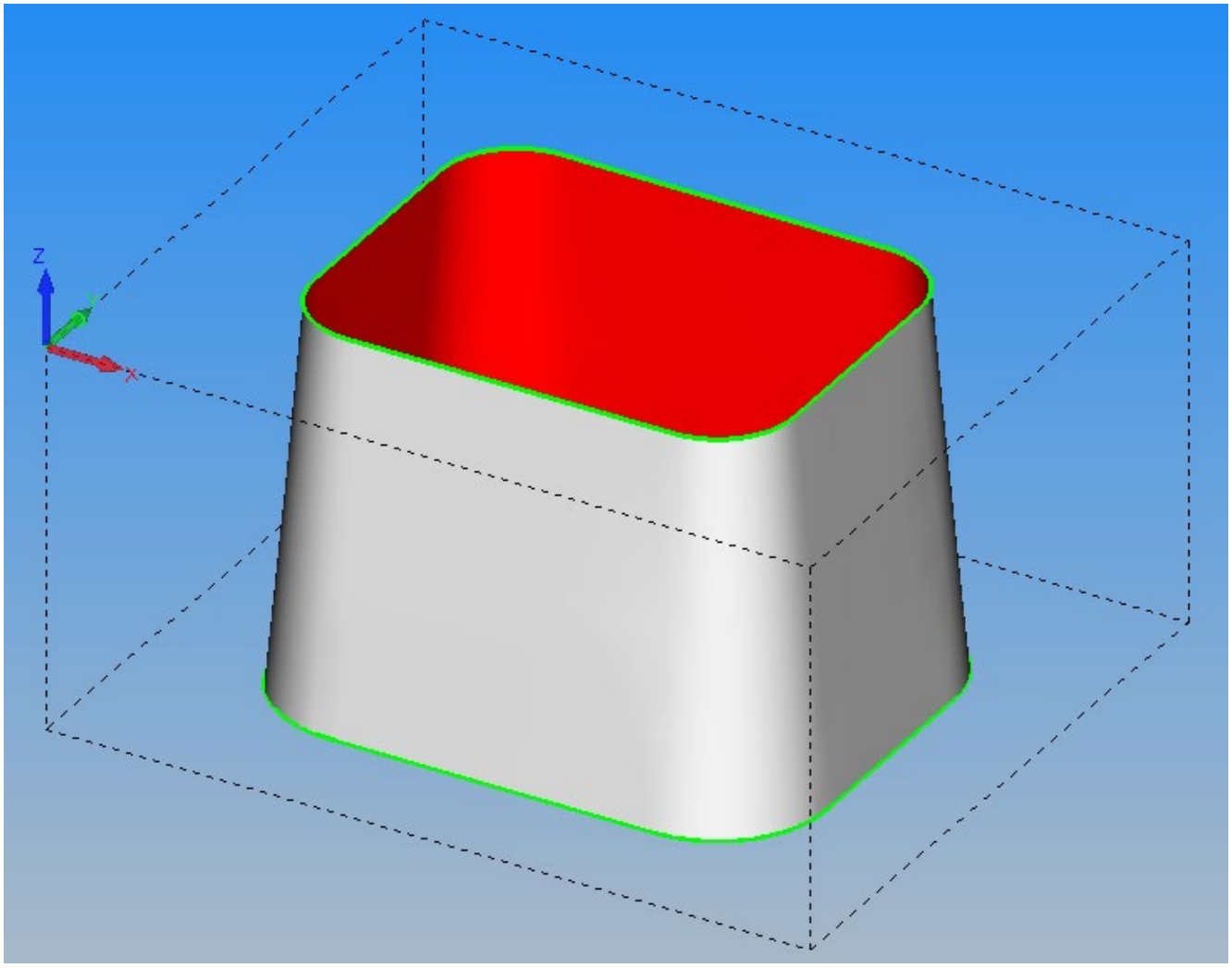


Figure 59 - Complete taper surface

The Flat Bottom

To create the bottom ruled surface, we need two more pieces of geometry adding to the current drawing.

Using **GEOMETRY > 3D Polyline**, draw two polylines
The first from X**0, Y0, Z-70** to X**0, Y120, Z-70** then **<RClick>** to finish.
The second from X**140, Y0, Z-70** to X**140, Y120, Z-70.**

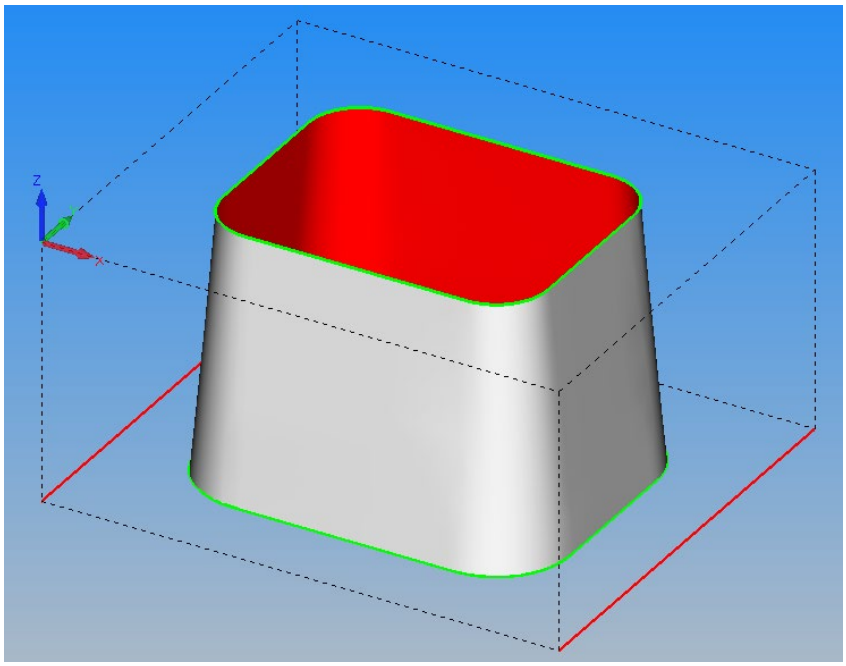


Figure 60 - Polylines added

Select **GEOMETRY > 3D Surfaces**, make the option to **⊙ Ruled Surface (2 Curves)**.

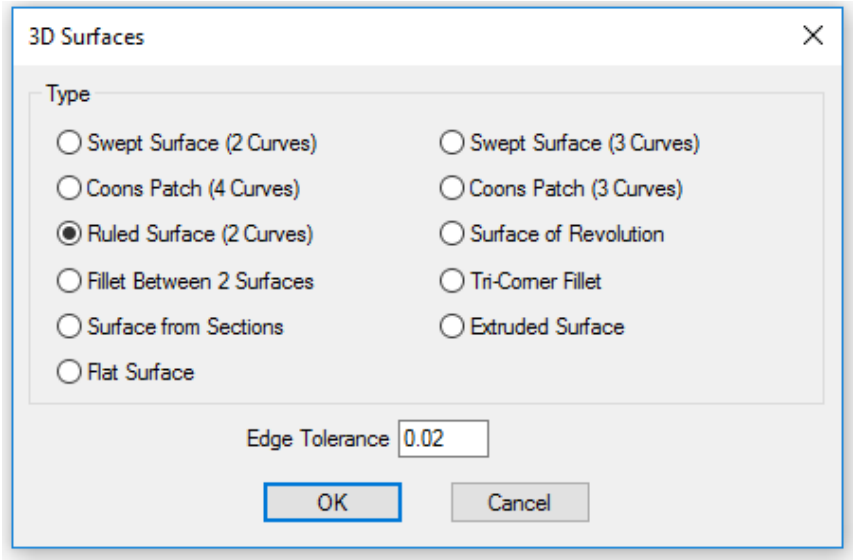


Figure 61 - Surface Creation dialogue

<LClick> [OK] to continue.

<LClick> both of the polylines as the Start and Finish requirements, then **<LClick> [OK]** for the precision for this example.

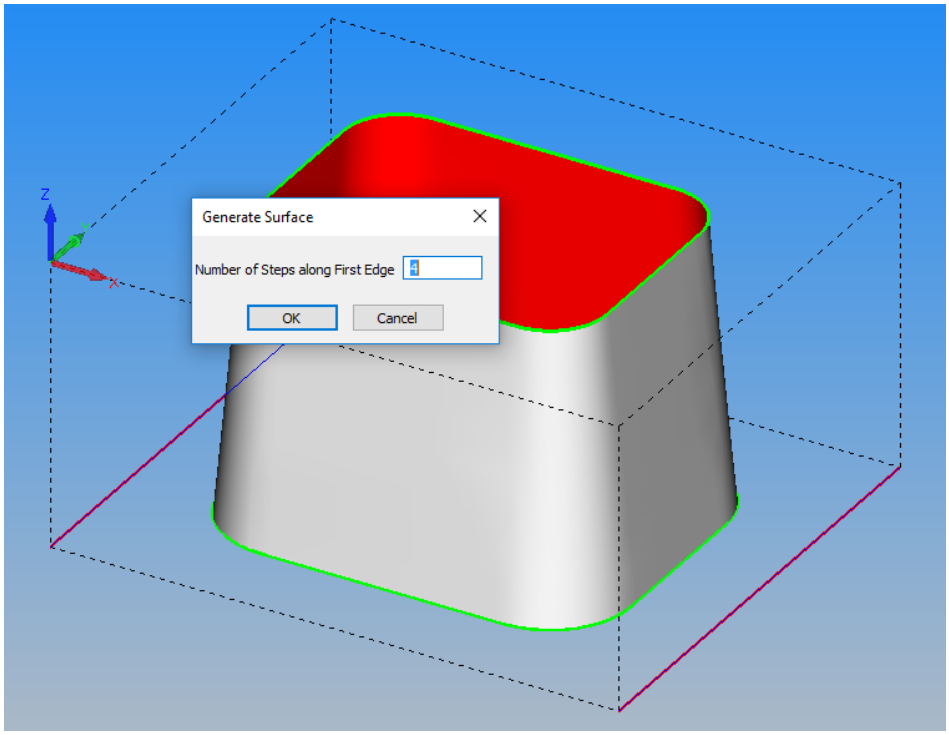


Figure 62 - Surface accuracy option

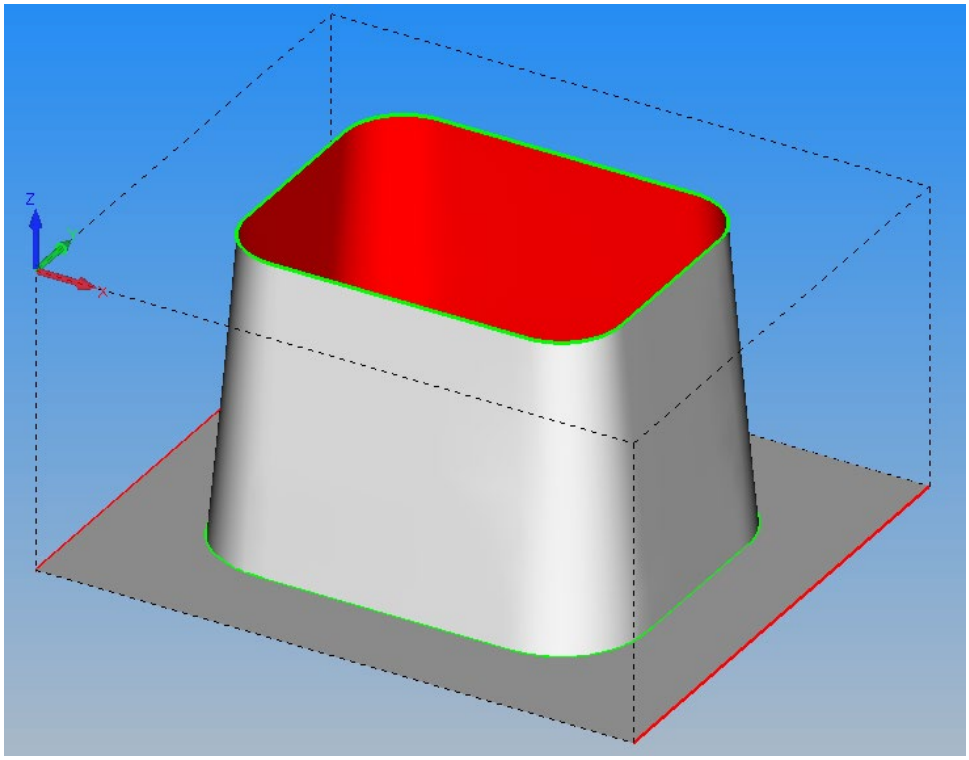


Figure 63 - Completed Ruled Surface

The Curved top

To create the curved top surface, we need to create a Work Plane on the front of the Work Volume to allow creation of the arc geometry used to generate the finished surface.

Use **VIEW > Iso**  if you are not in this view already.

Use **WORK PLANES >_Slice through Work Volume**  and **<LClick>** the Work Volume line running along the global X axis.

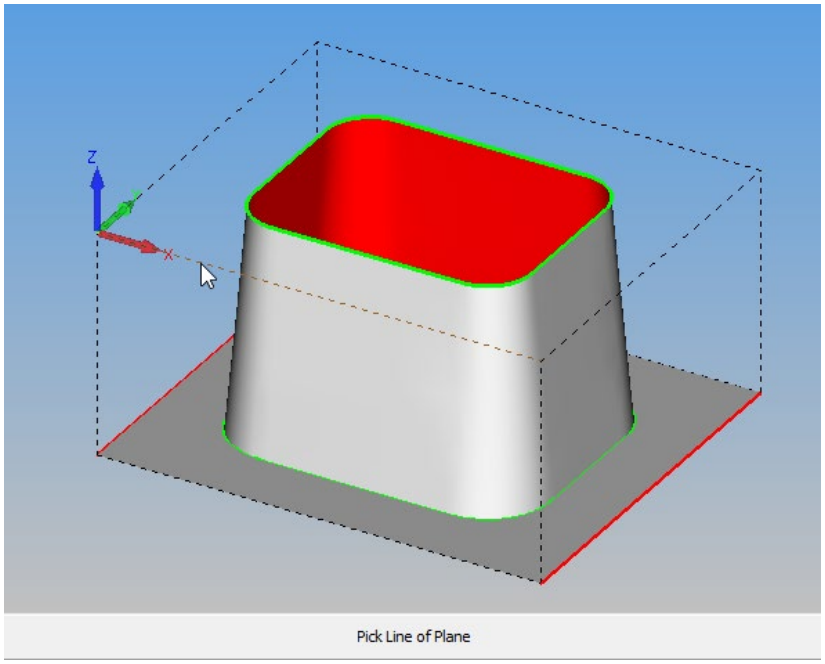


Figure 64 - Line for Work Plane slice

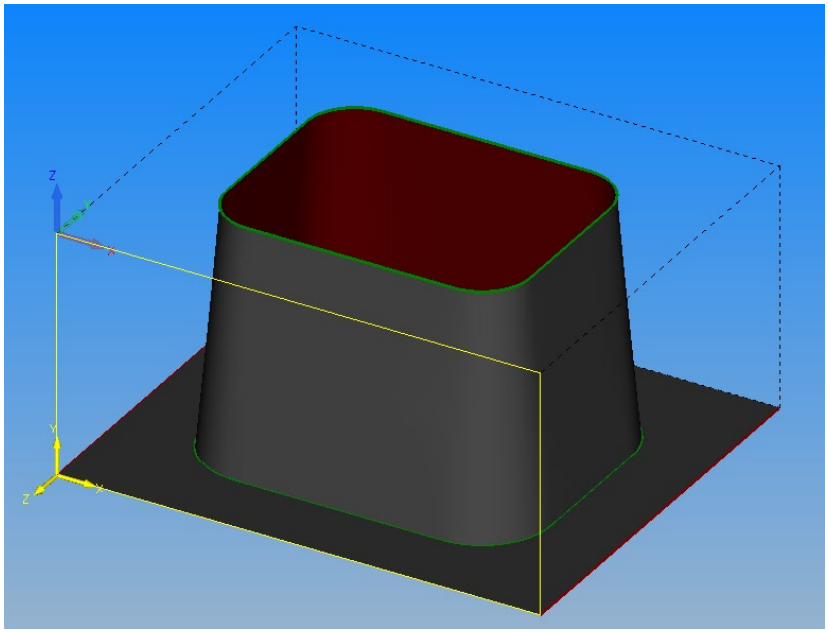


Figure 65 - Generated Front Work Plane

Creating the reference points for the curve

Select **VIEW > Work Plane**  to visualise only the section being drawn.

Using **GEOMETRY > Line**  , create a line from **X20, Y0, <LClick> [OK].**

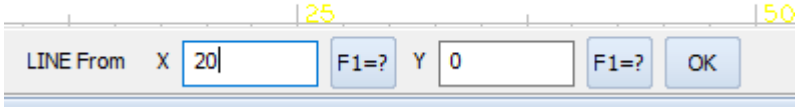


Figure 66 - Line Start Point

to **X [F1 =?], Y50, <LClick> [OK].**

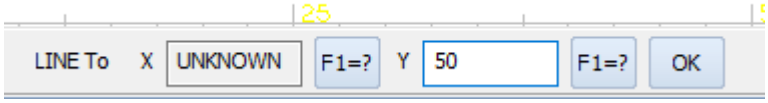


Figure 67 - Line Unknown End Point

When prompted enter an angle value of **85, <LClick> [OK].**

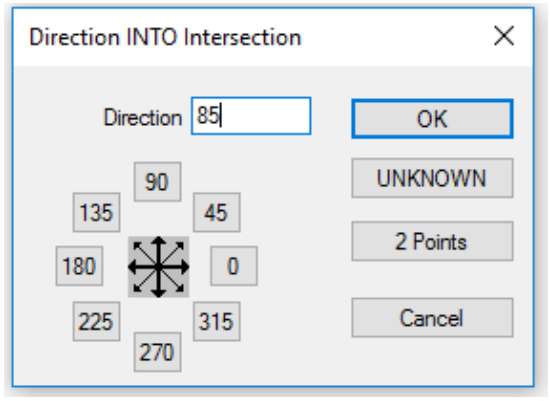


Figure 68 - Direction options

Figure 69 - Created Line

To create the second line, we need to mirror the first one, but we have no reference points to use in the current view.

Use **VIEW > Iso**  to make the mirroring easier.

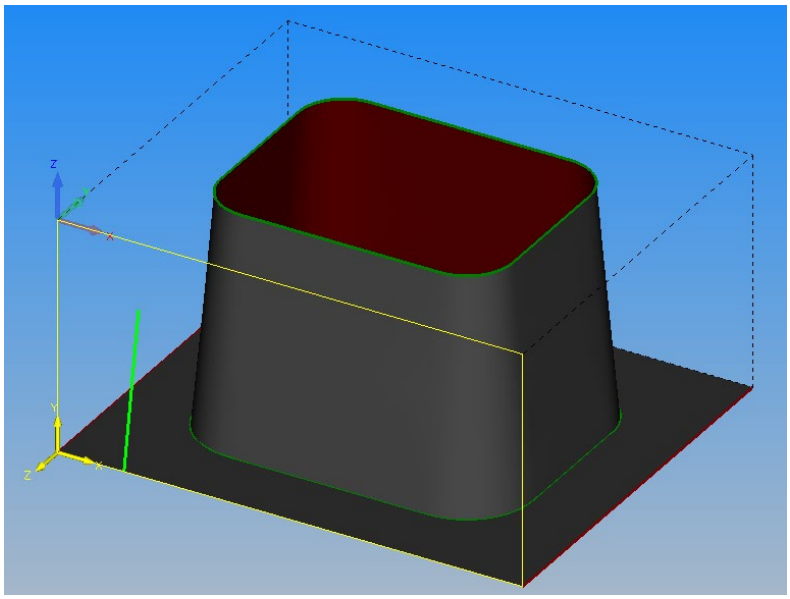


Figure 70 - ISO view of first side line

Using **EDIT> Mirror** 
<LClick> the previously drawn line as the item to mirror.

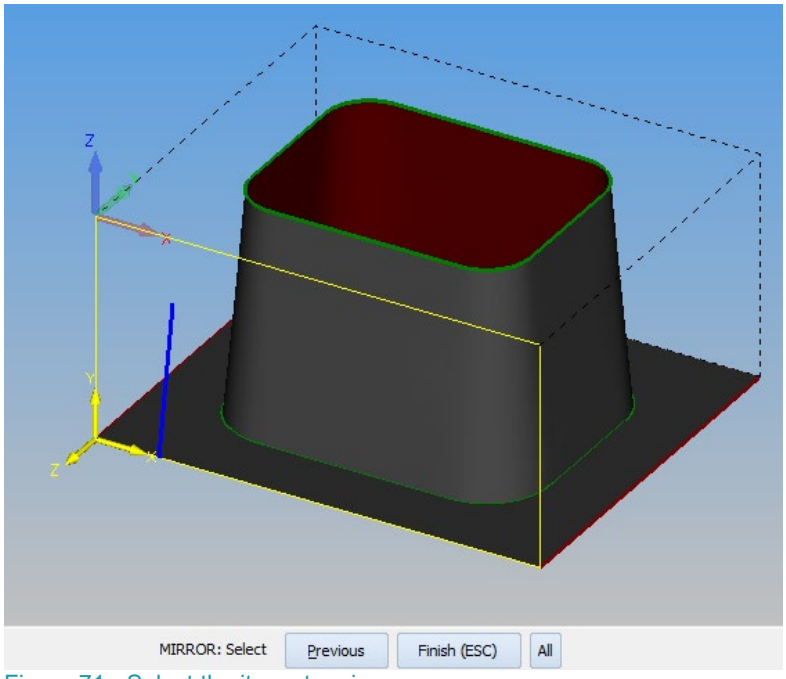


Figure 71 - Select the items to mirror

<RClick> or [Finish (ESC)] to continue.

For the First Point of Mirror, use **MID-Point of <F7>** and **<LClick>** the lower line shown below.

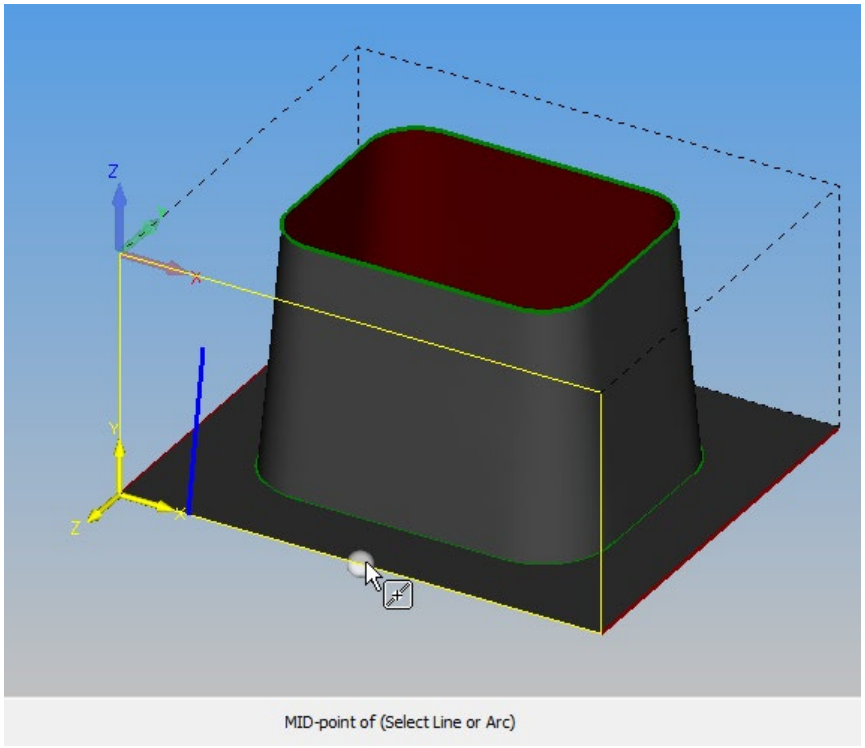


Figure 72 - First point of Mirror

For the Second Point of Mirror, use **MID-Point of <F7>** and **<LClick>** the upper line shown below.

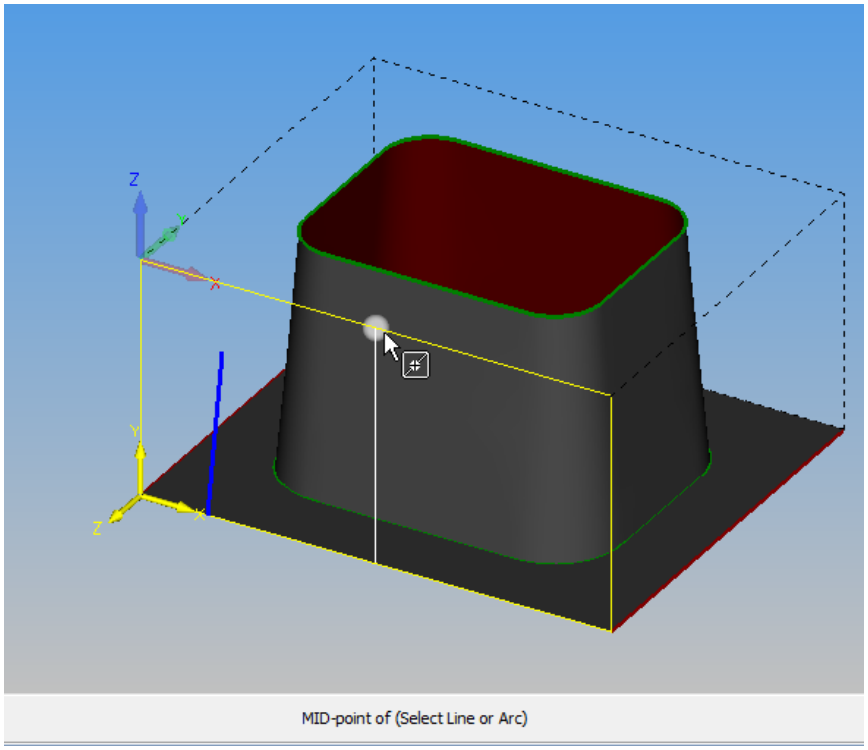


Figure 73 - Second point of Mirror

When prompted, answer **[Yes]** to keep the original.
Then **<RClick>** to finish.

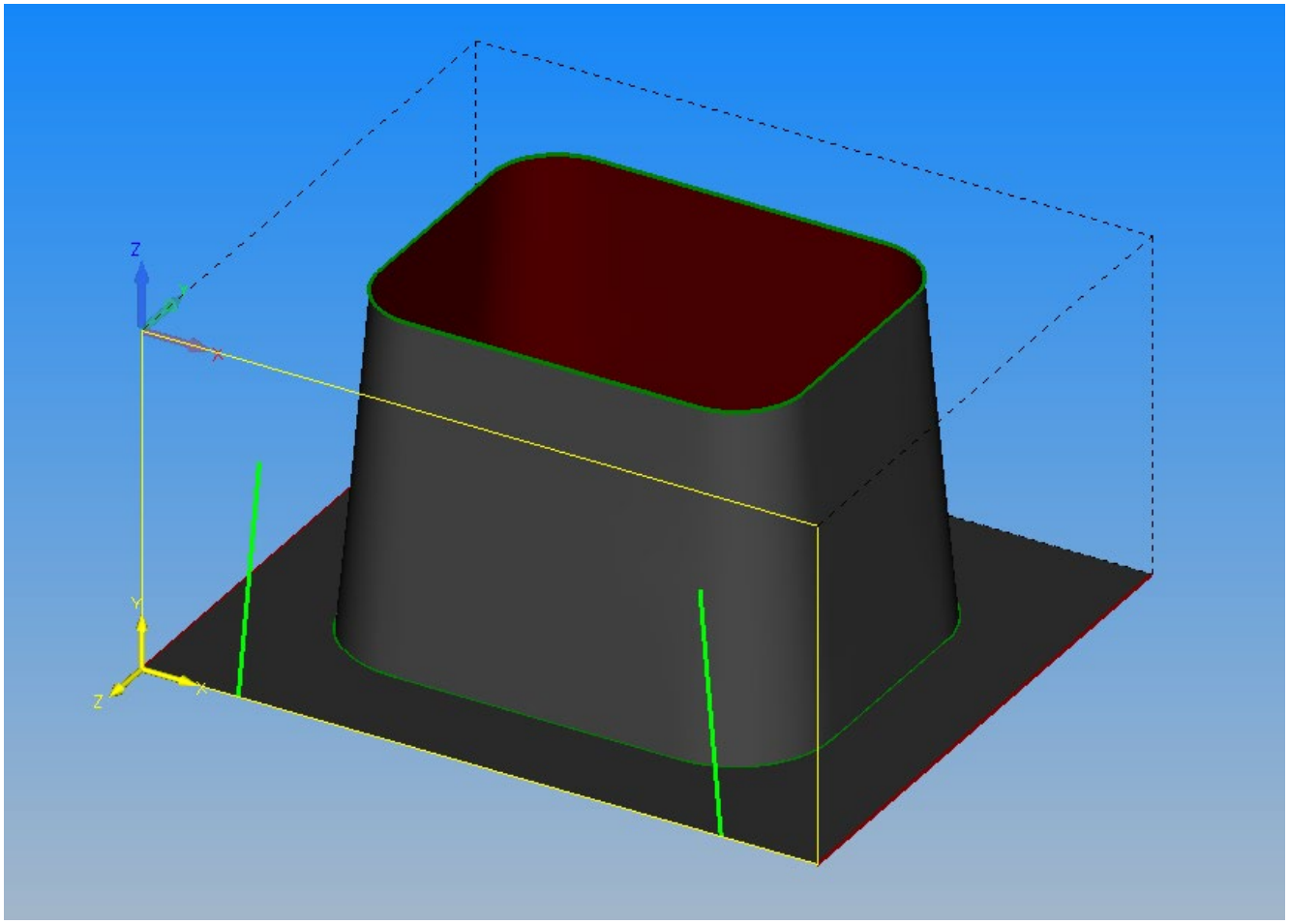


Figure 74 - Completed Mirror

Create the Curve

Using **GEOMETRY > Arc > 2 Points and Radius** 
For the First point, use **END-Point of <F6>** and **<LClick>** the top of the right hand line.

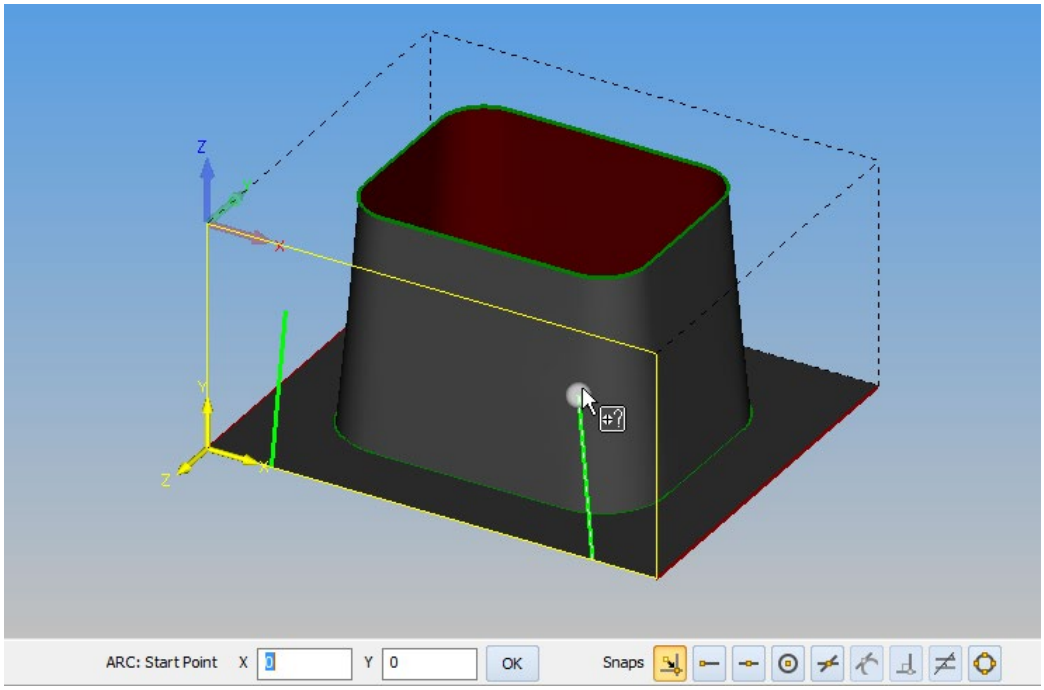


Figure 75 - First point of the arc

For the Second point, use **END-Point of <F6>** and **<LClick>** the top of the left hand line.

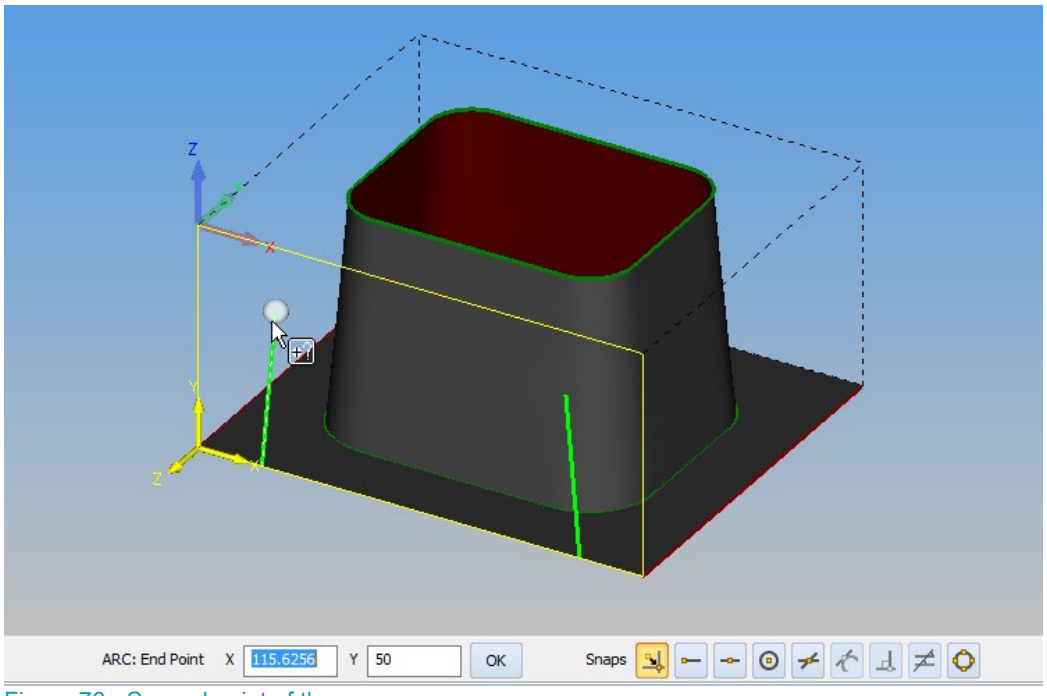


Figure 76 - Second point of the arc

For the **Arc Radius**, enter the value **103**.

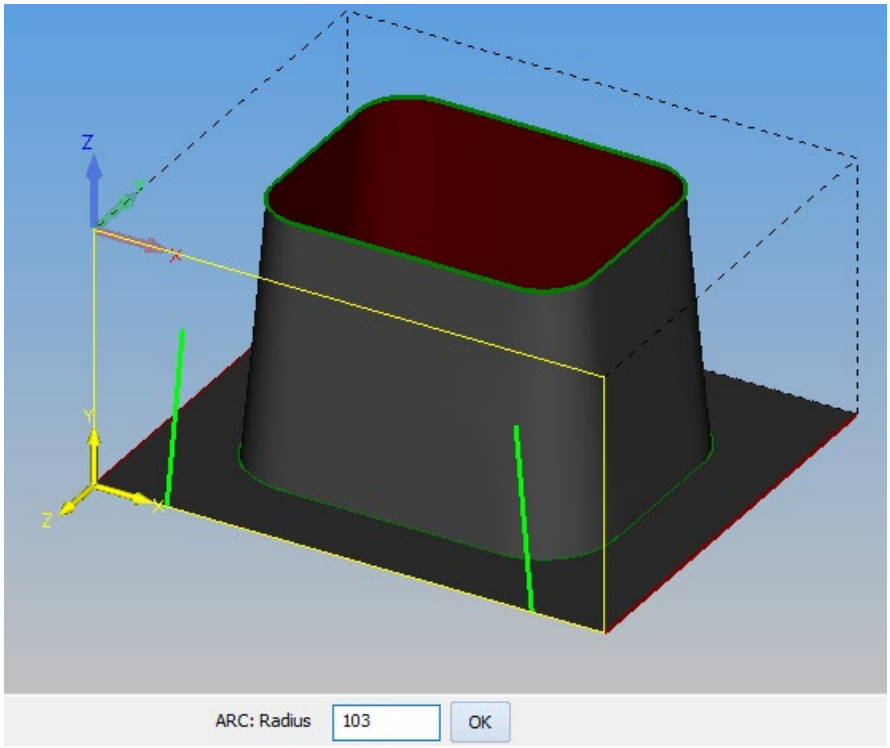


Figure 77 - Arc radius value

<LClick> [OK] to complete, **<RClick>** to finish the command.

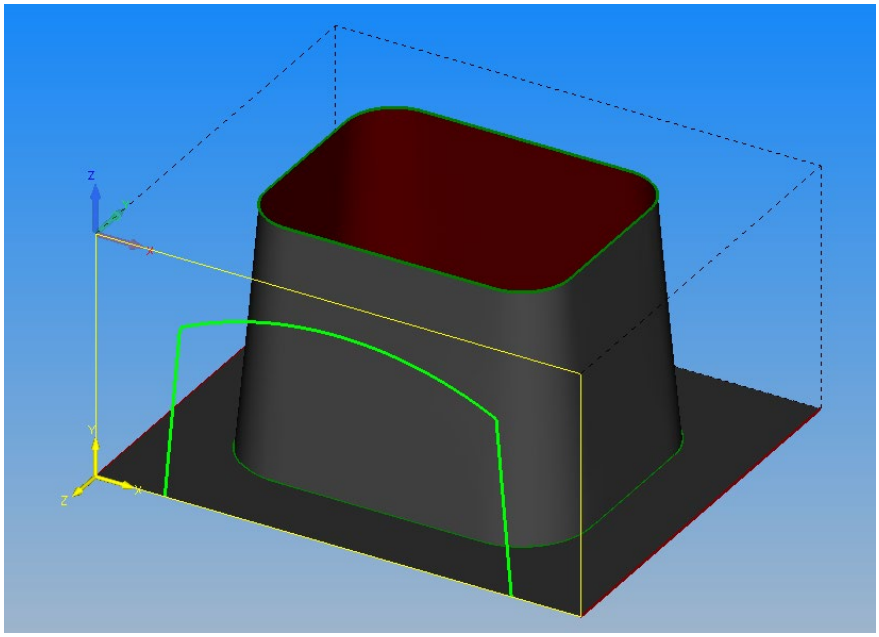


Figure 78 - Finished arc

Arcs in Alphacam are always drawn Counter Clockwise.

To allow the surface creation process to work correctly, the arc needs to be longer than we have drawn.

Using the **EDIT > Extend by Distance** command, extend the arc each side by a value of 10.

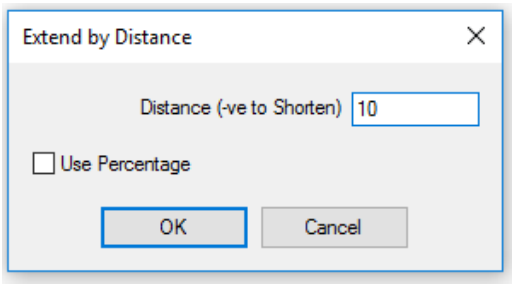


Figure 79 - Extend by Distance dialogue

<LClick> [OK] to continue, then when prompted **<LClick>** each end of the arc to extend it.

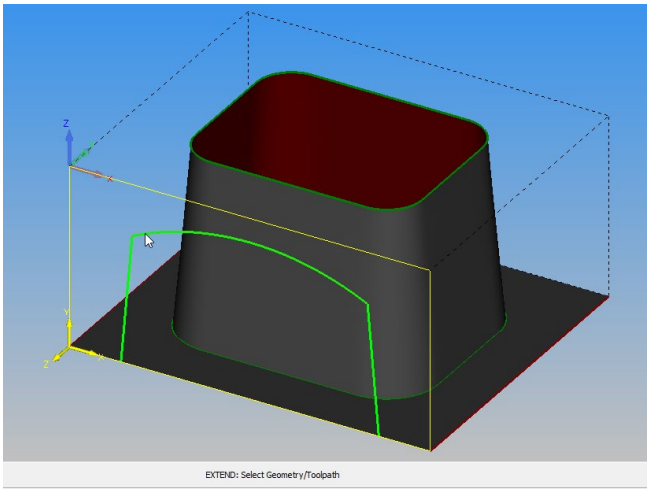


Figure 80 - Extend Prompt

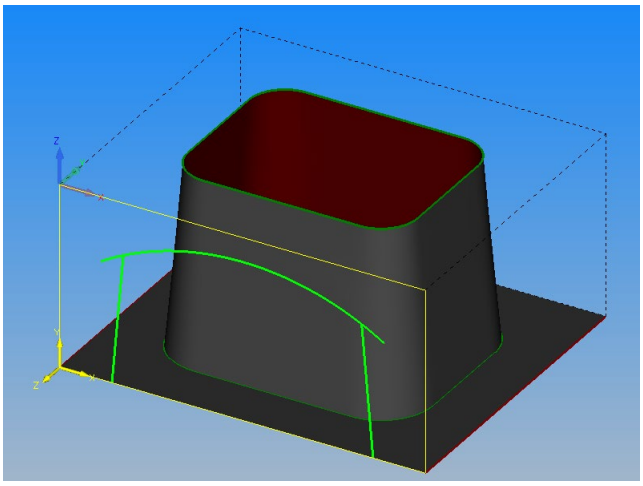


Figure 81 - Completed Extend by Distance

Select **WORK PLANE > Cancel Work Plane**

Create the Far side reference geometry

Use **EDIT > Auto/2D/3D > 3D**  to set the drawing mode to 3D.

Use **EDIT > Copy** 
For the geometry to be copied, **<LClick>** the Arc to select it.

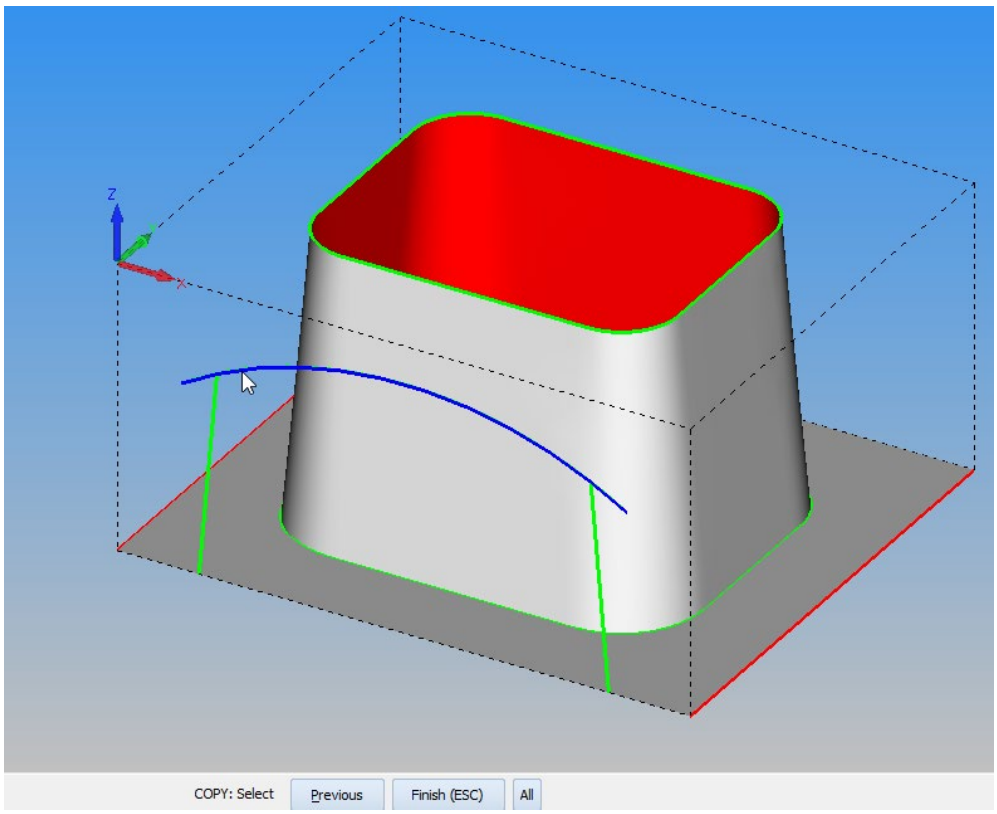


Figure 82 - <LClick> the arc to select it

<RClick> or **[Finish (ESC)]** to continue.
For the base point use X**0, Y0, Z0. <LClick> [OK]** to continue.

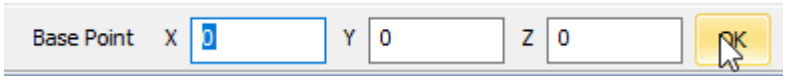


Figure 83 - Copy Base Point

For the New Position, use **X0, Y120, Z0 <LClick> [OK]** to continue.

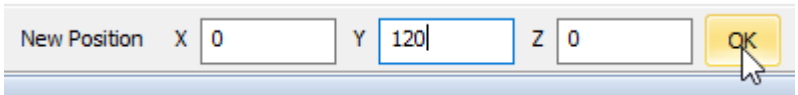


Figure 84 - Copy New Point

<RClick> or **[Finish (ESC)]** to apply the copy.

<RClick> to finish the command completely.

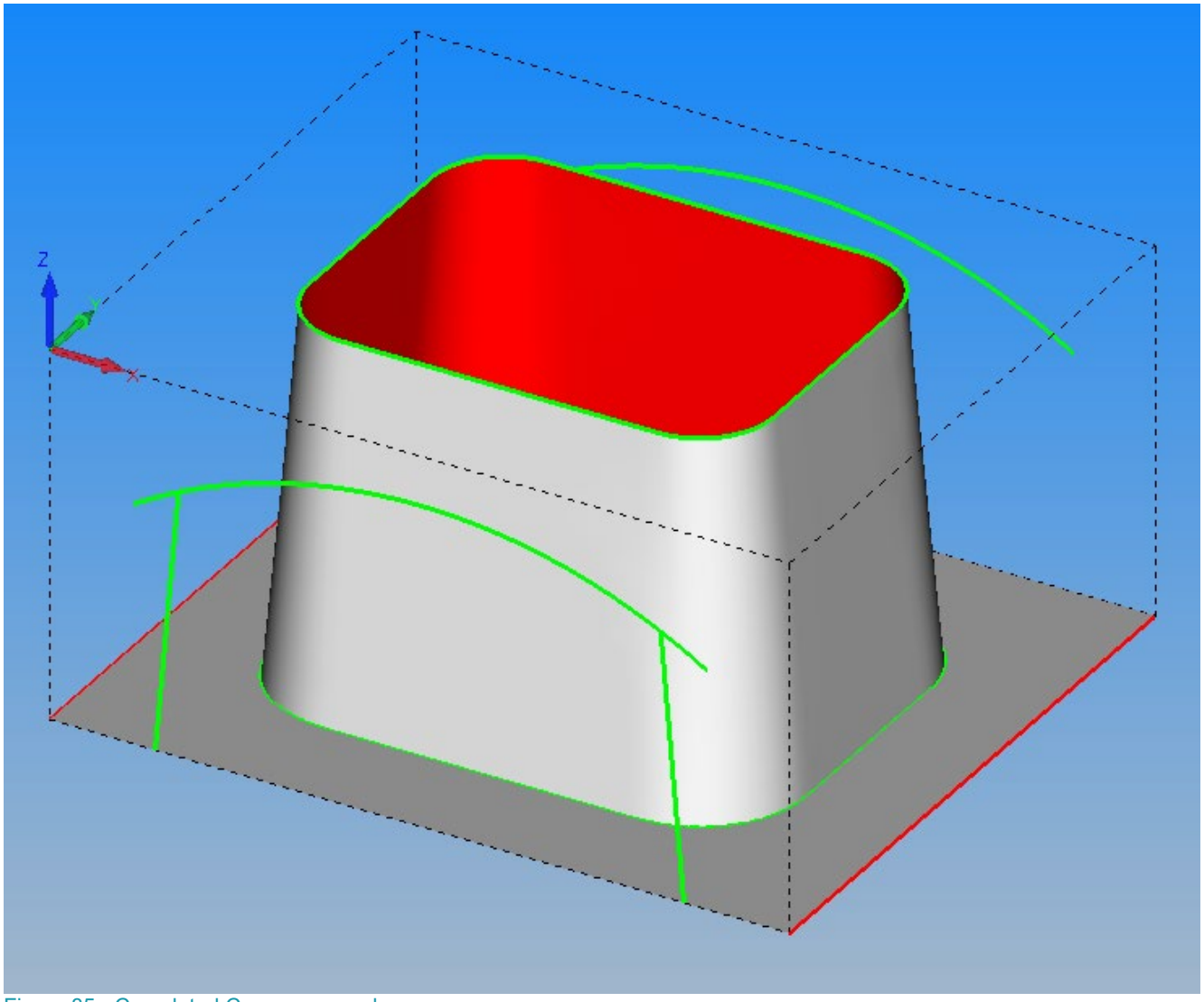


Figure 85 - Completed Copy command

Arc surface creation

Select **GEOMETRY > 3D Surfaces** , make the option to **⊙ Ruled Surface (2 Curves)**.

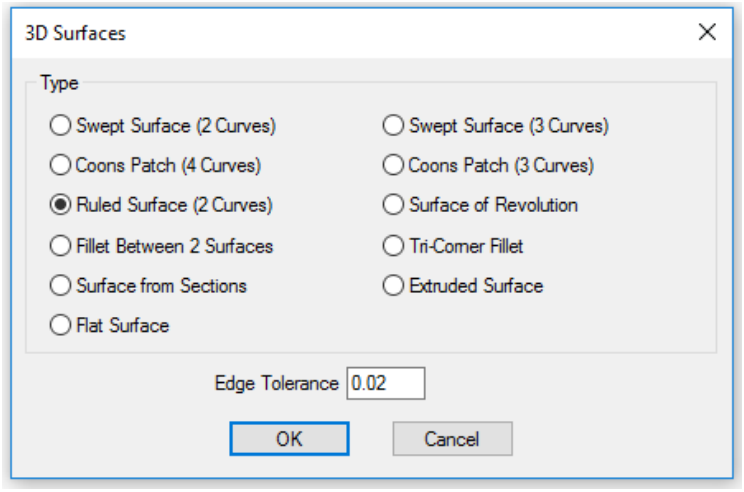


Figure 86 - Surface Creation dialogue

<LClick> [OK] to continue.
For the Start Section, **<LClick>** the front arc

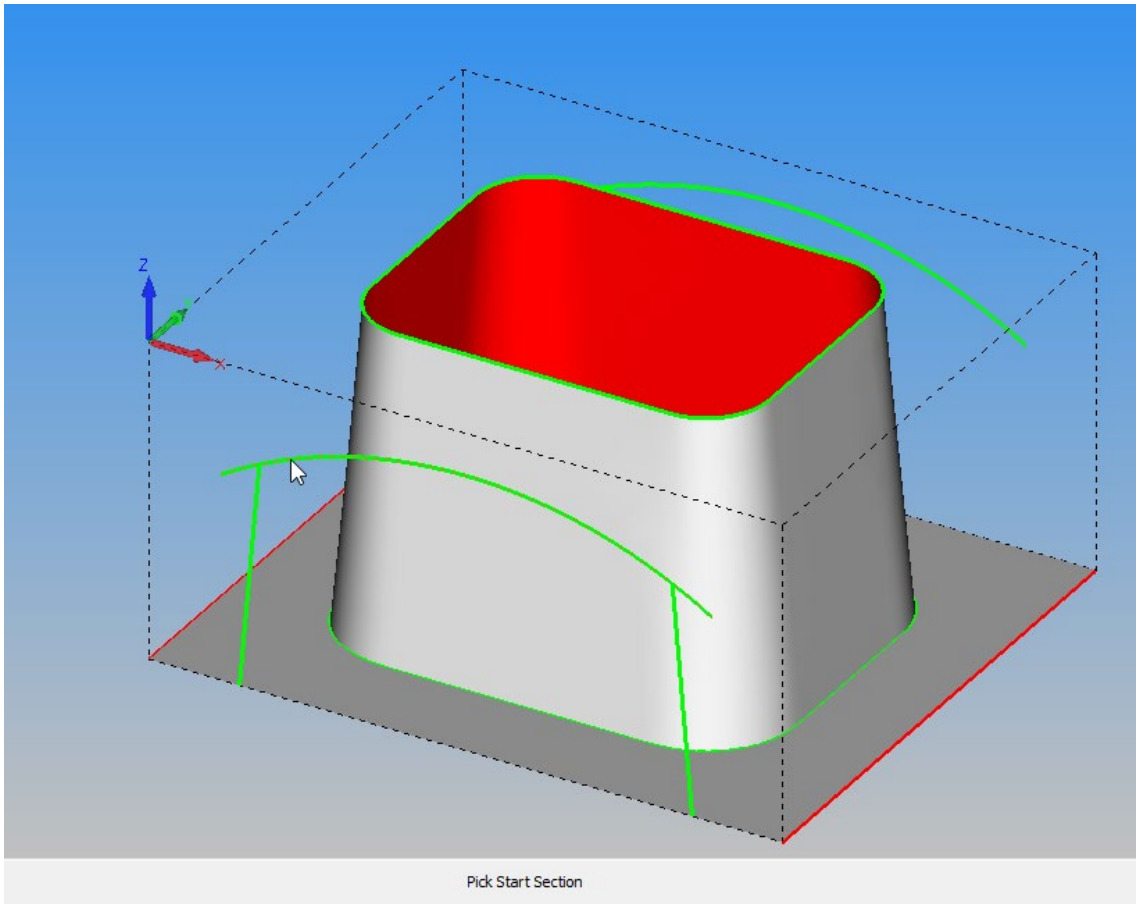


Figure 87 - Start Section Selection

For the Finish Section, **<LClick>** the rear arc.

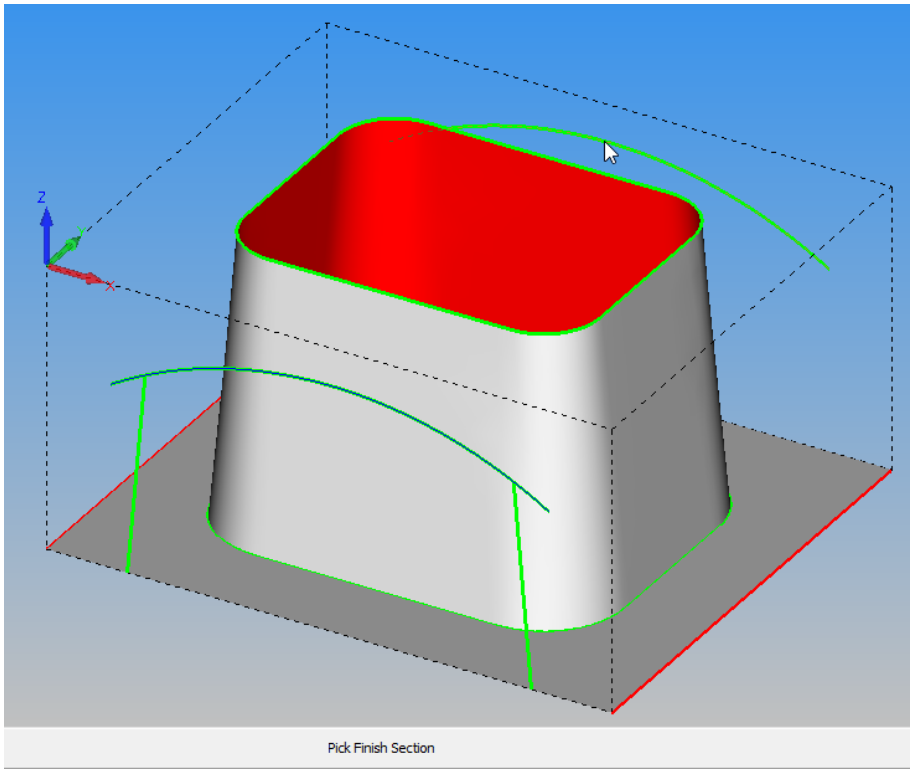


Figure 88 - Finish Section selection

<LClick> [OK] for the default precision options when prompted to complete the surface.

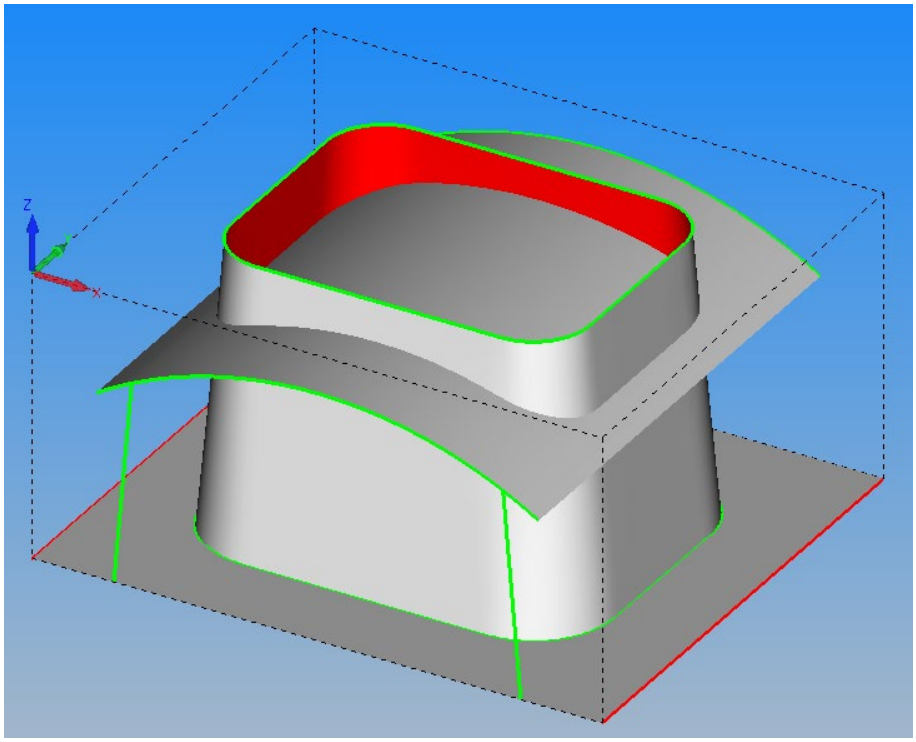


Figure 89 - Completed curved surface

Adding the top surface Fillet

One important consideration prior to the next section of surface creation is the way in which the **Fillet Between 2 Surfaces** command works.
You can only apply a surface fillet to a silver face of a surface.
On our current example, if we applied a fillet to the tapered surface and the arc surface, the finished part would be incorrect, so we must manipulate both of these two surfaces prior to the filleting option.

Use **3D > Reverse Tool Side**  , then **<LClick>** both the tapered and arc surfaces when prompted.

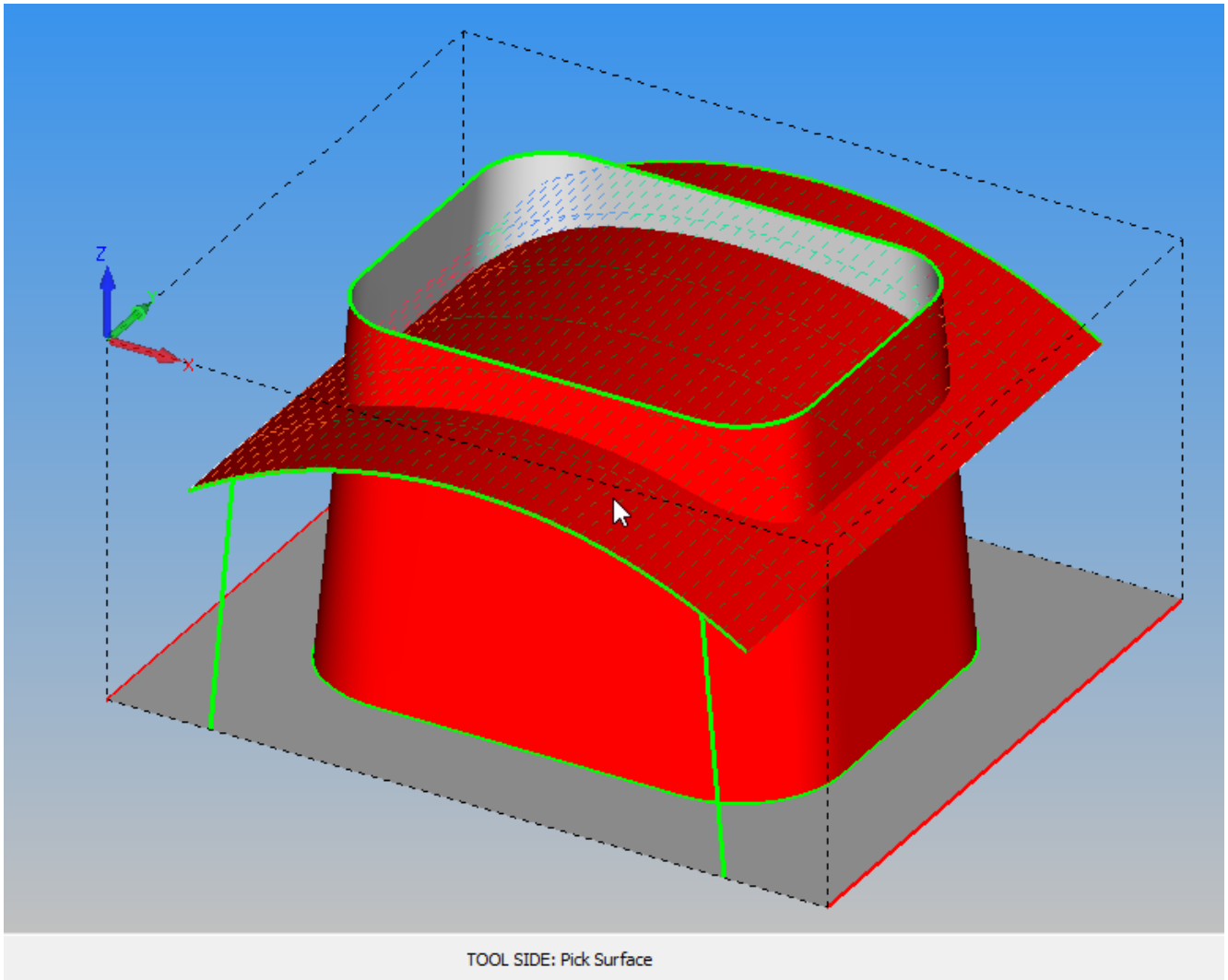


Figure 90 - Reversed surfaces

<RClick> to finish.

Select **GEOMETRY > 3D Surfaces** , make the option to **⊙ Fillet Between 2 Surfaces**.

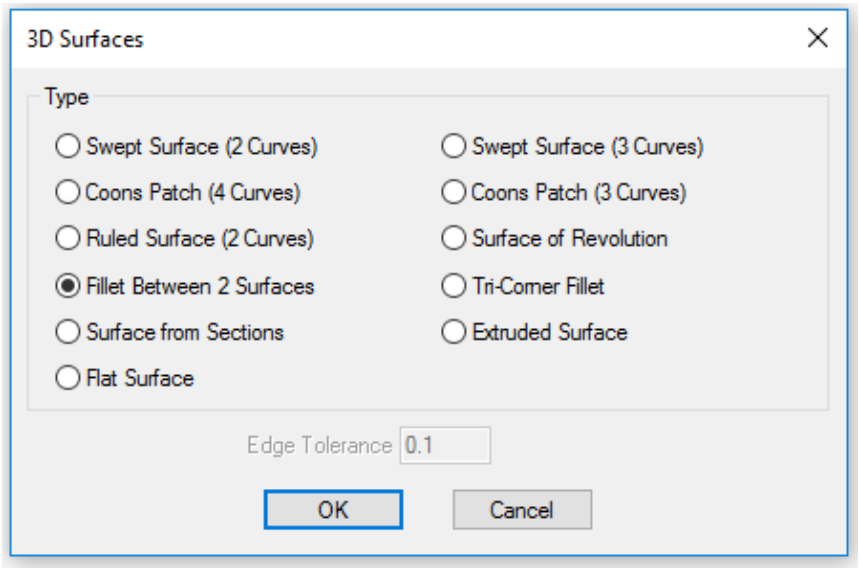


Figure 91 - Surface Creation dialogue

<LClick> [OK] to continue.
You are prompted to select a location on the first surface where the fillet will come from.
<LClick> on the arc surface as shown below.

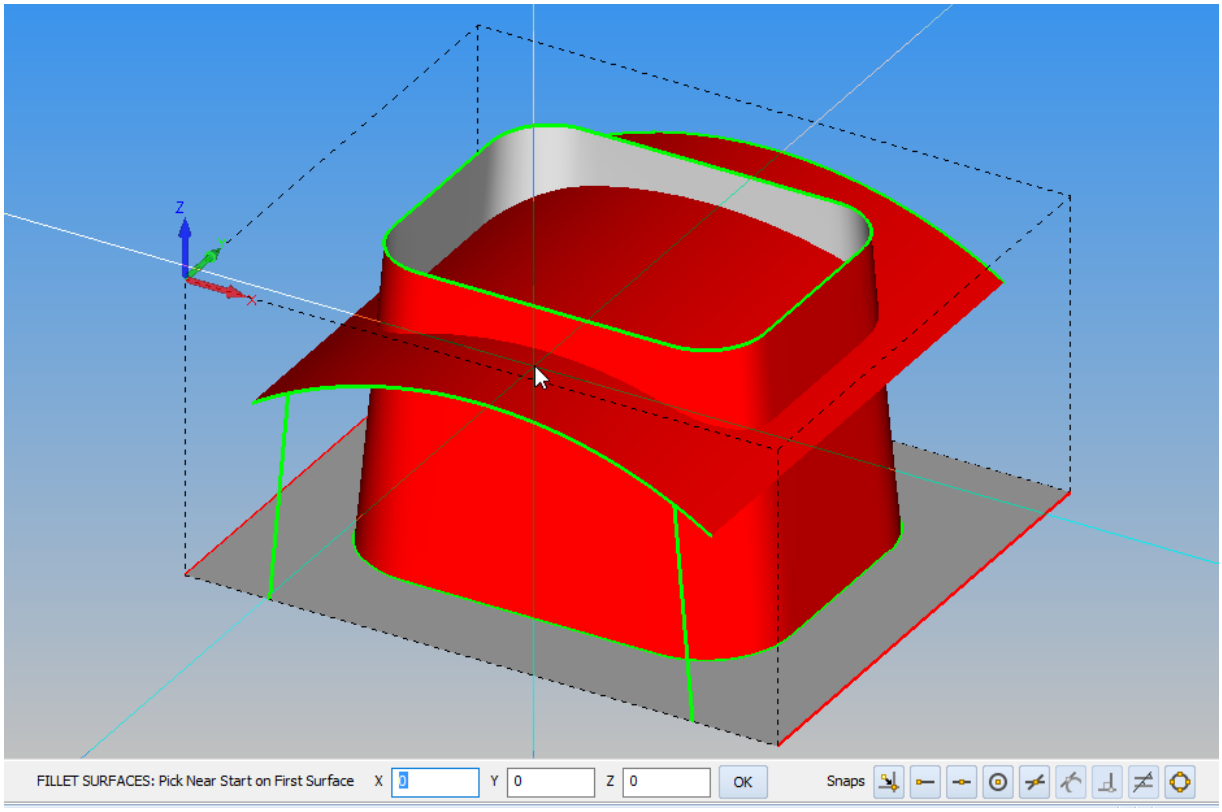


Figure 92 - Selecting the start of the fillet surface

You will then be asked for a second surface to create the fillet to, **<LClick>** on the tapered surface as indicated below.

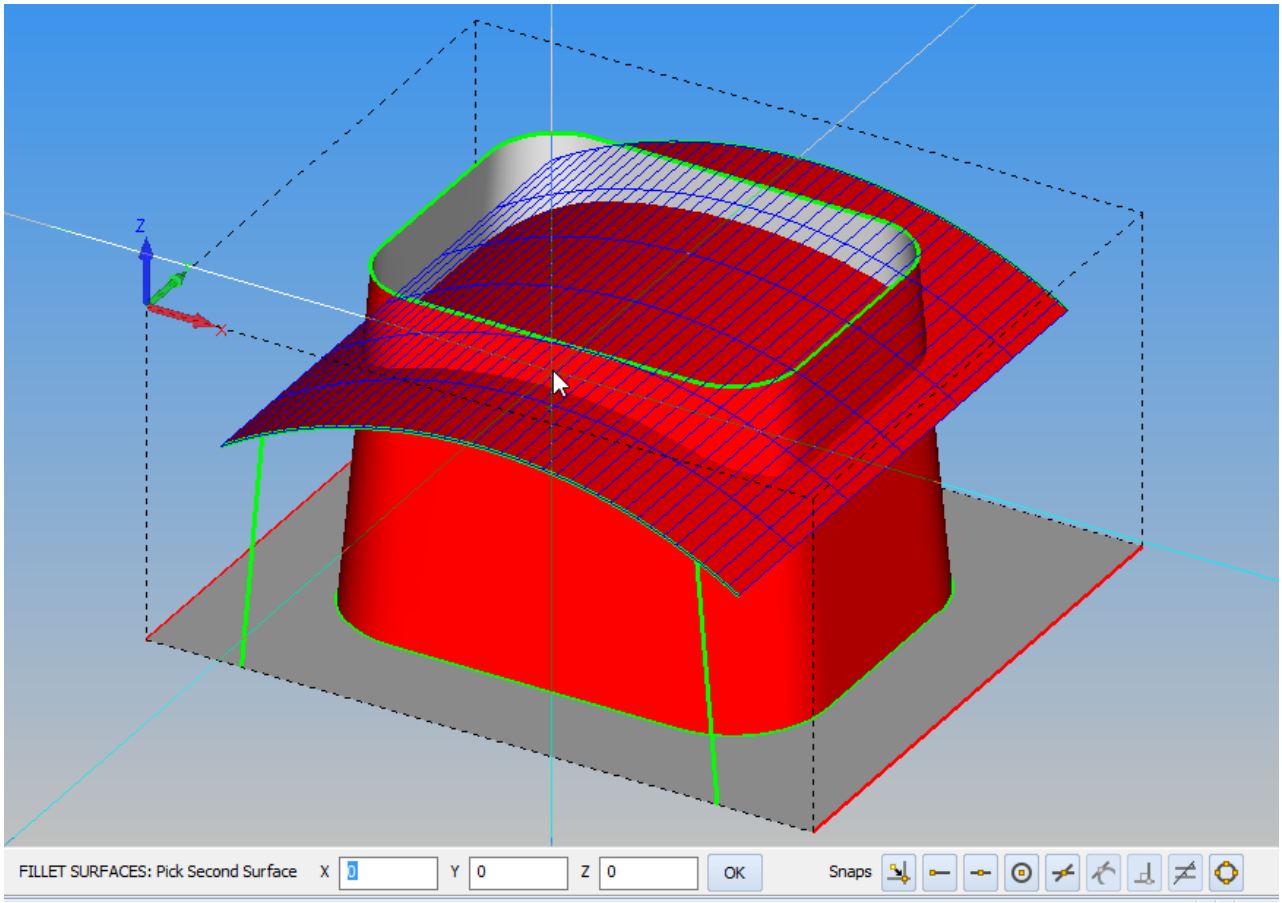


Figure 93 - Selecting the second surface.

Once you have selected the second surface, the **Fillet** size option dialogue is displayed.
Both the Start and End radius values are 8, this is a complete fillet around a full surface, if you were working on a single edge it is possible to apply a tapering fillet by entering different values for the Start and End radius.

The **Step Length** allows you to change the precision at which the fillet is created, for this example we will accept the default options.

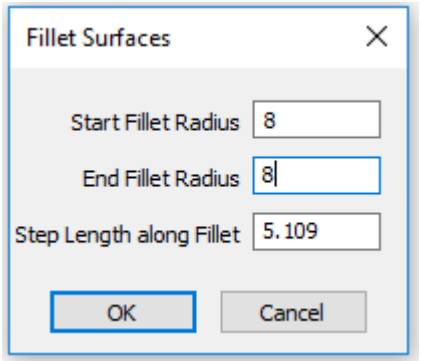


Figure 94 - Surface Fillet dialogue options

<LClick> [OK] to apply the fillet.

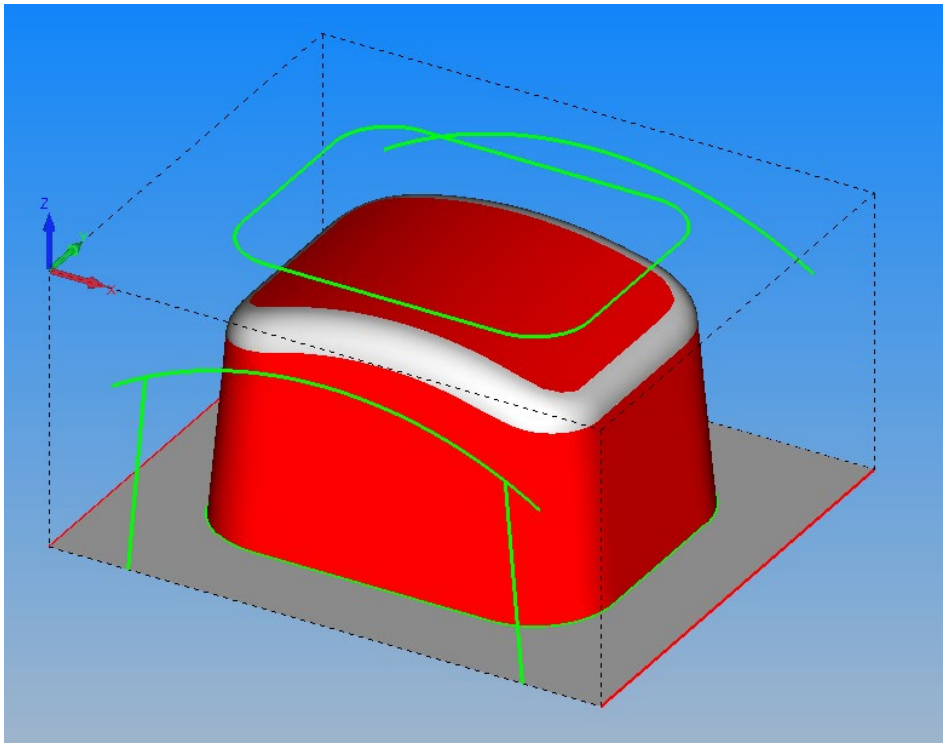


Figure 95 - Complete applied fillet surface

Use **3D > Reverse Tool Side**  , then **<LClick>** both the tapered and arc top surfaces when prompted. **<RClick>** to finish.

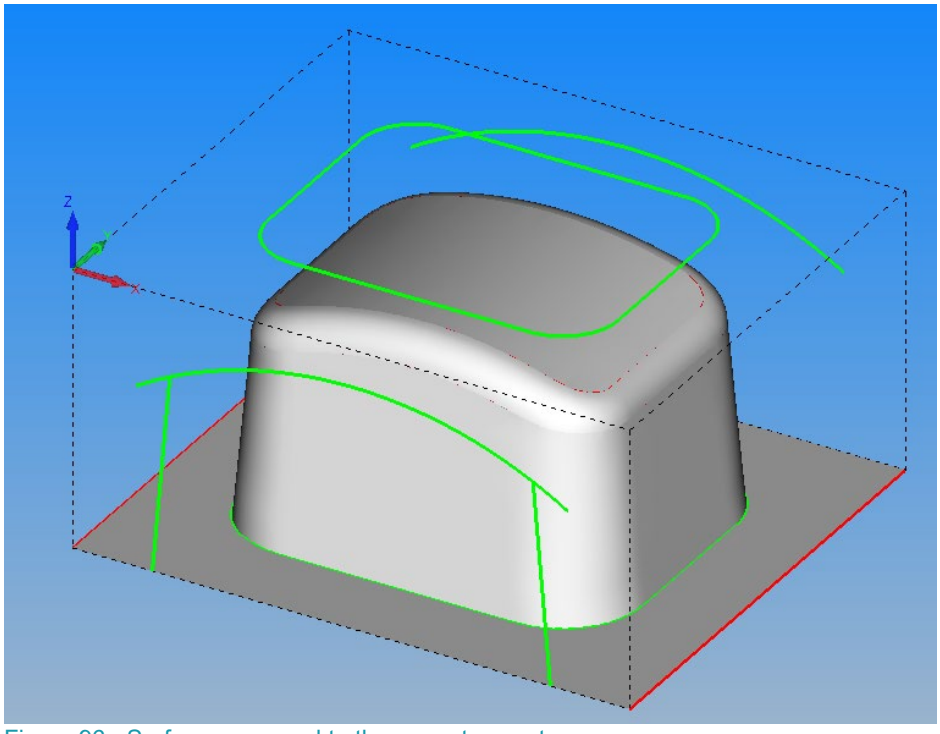


Figure 96 - Surfaces reversed to the correct aspect

The Bottom Fillet

Select **GEOMETRY > 3D Surfaces** , make the option to ⊙ **Fillet Between 2 Surfaces**.

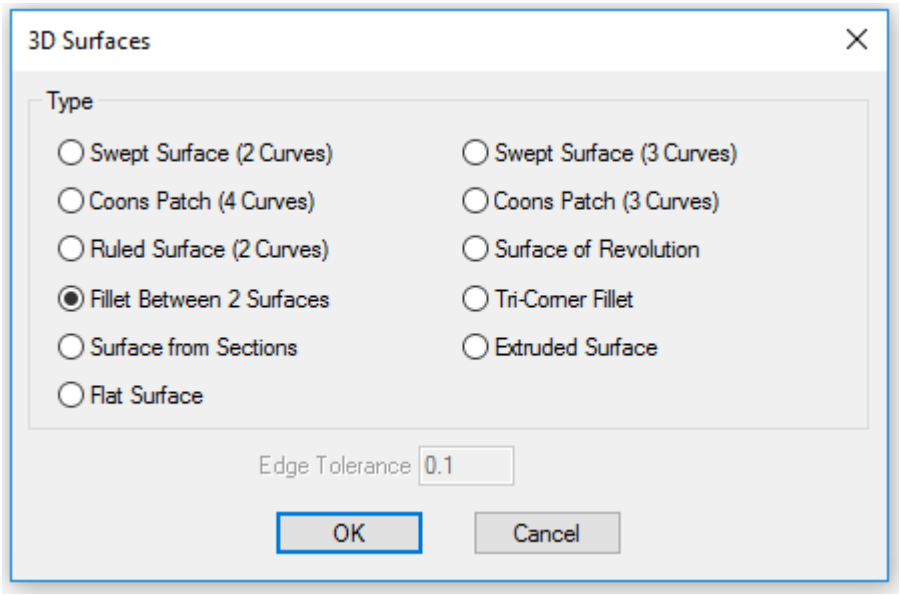


Figure 97 - Surface Creation dialogue

<LClick> [OK] to continue.

You are prompted to select a location on the first surface where the fillet will come from.
<LClick> on the flat surface as shown below.

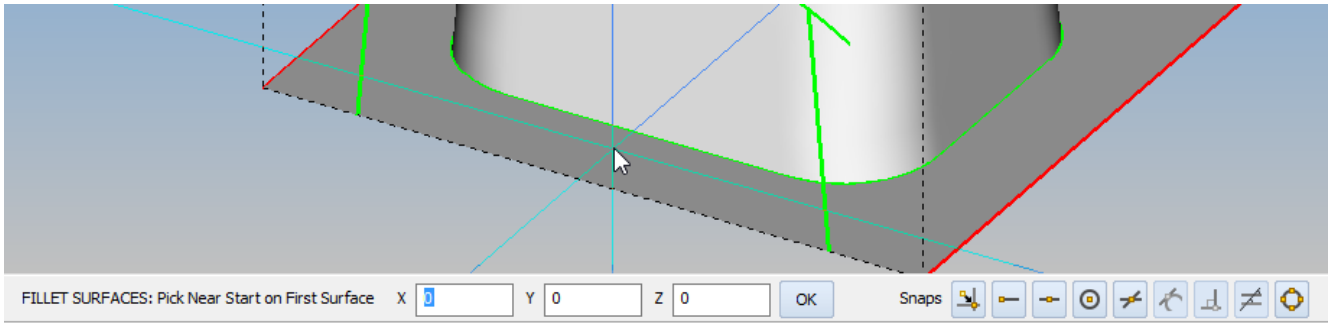


Figure 98 - Selecting the start of the fillet surface

You will then be asked for a second surface to create the fillet to, **<LClick>** on the tapered surface as indicated below.

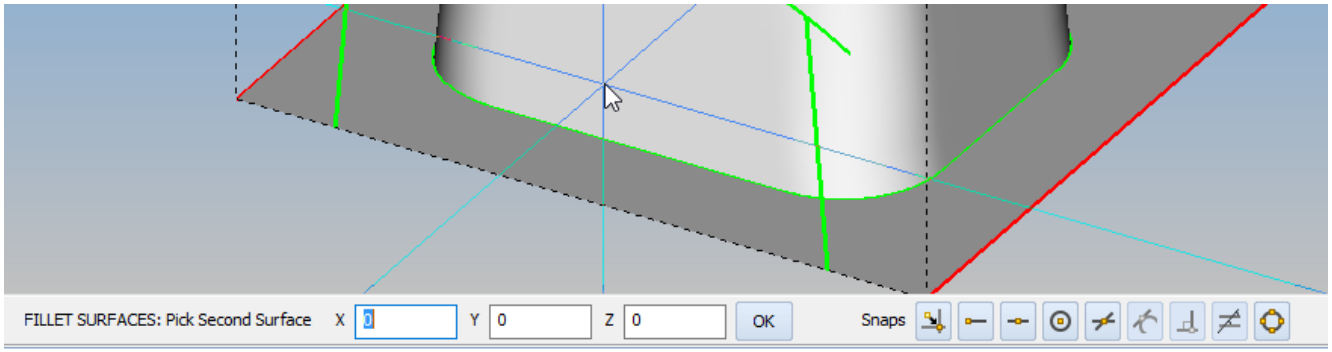


Figure 99 - Selecting the second surface.

Once you have selected the second surface, the Fillet size option dialogue is displayed.
Both the Start and End radius values are 8.

The Step Length allows you to change the precision at which the fillet is created, for this example we will accept the default options.

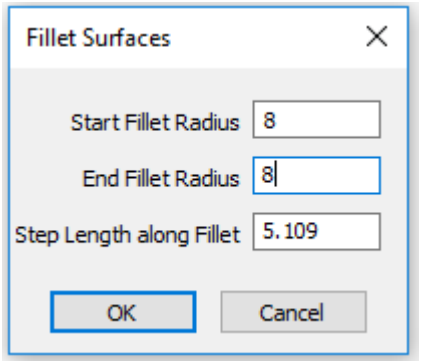


Figure 100 - Surface Fillet dialogue options

<LClick> [OK] to apply the fillet.

Select **GEOMETRY > 3D Surfaces** and pick the **Fillet between 2 Surfaces** option.
For the surfaces select a parameter line in the bottom surface and a parameter line in the tapered surface Set the fillet size to **8** for BOTH starting and ending radius.

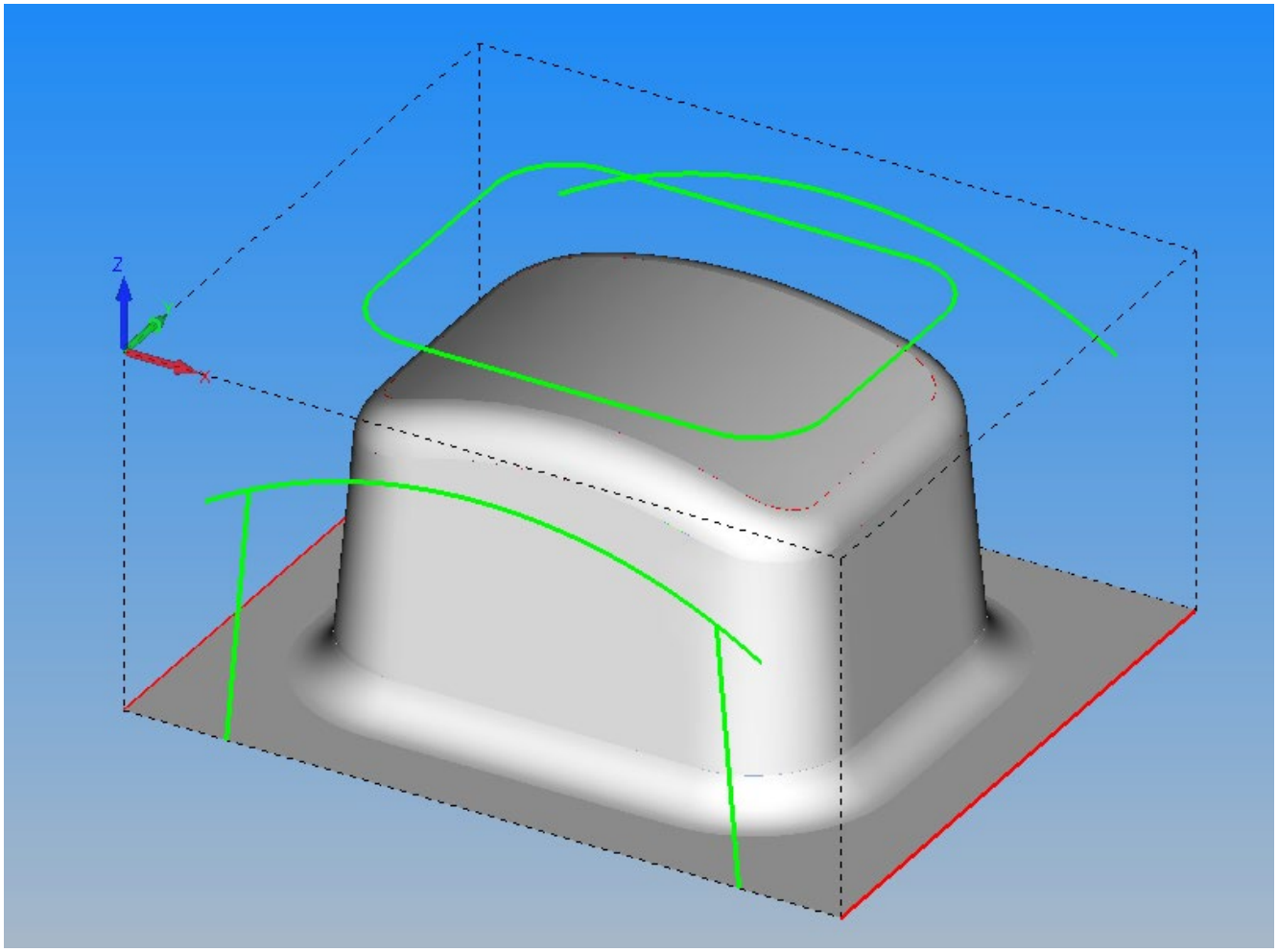


Figure 101 - Completed surface model Example 1 with applied fillets

Save your work with a suitable name.

Example 2 Tutorial

Geometry creation

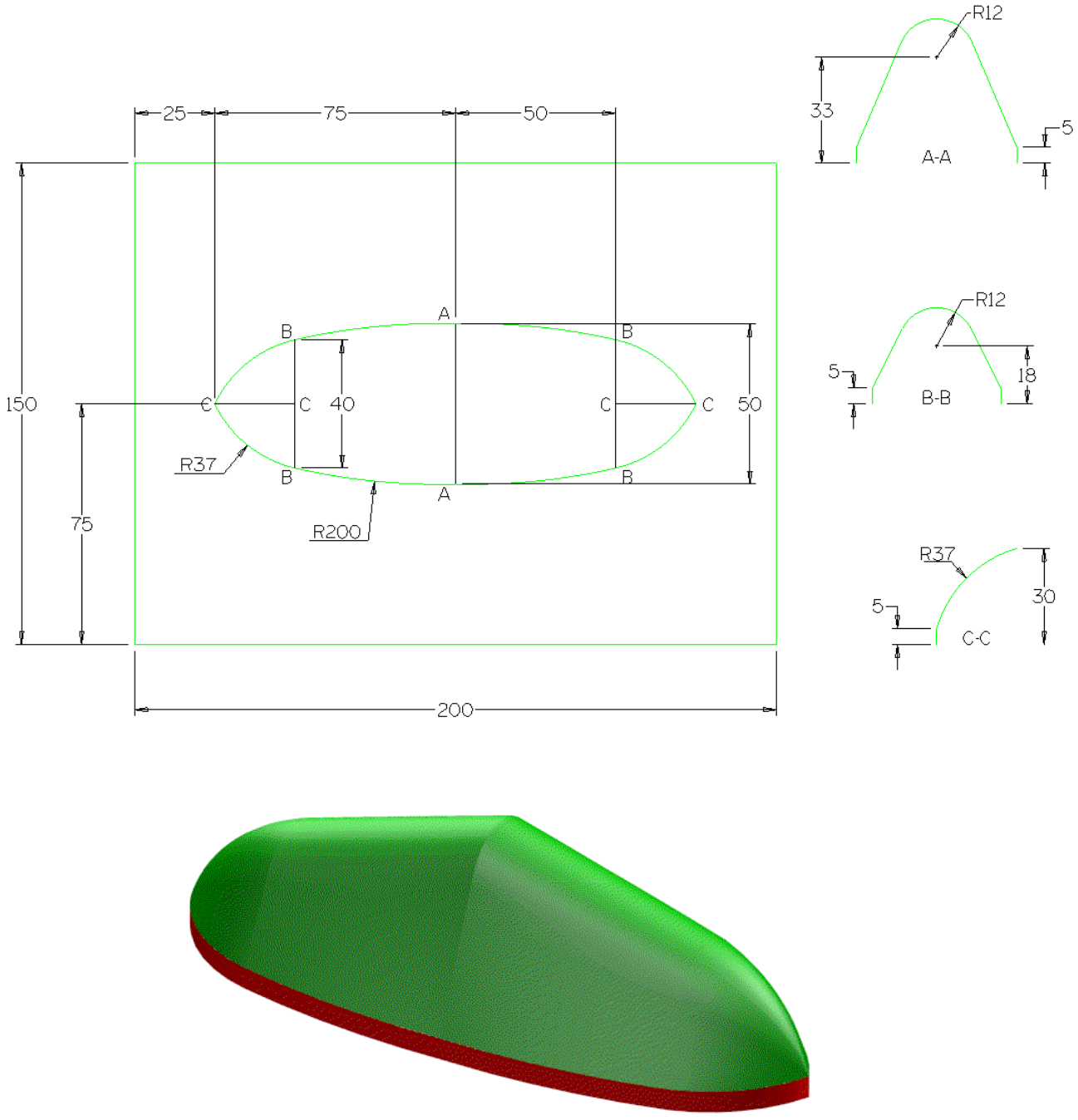


Figure 102 - Worked example reference drawing and completed example

Draw the 2D geometry as shown below.
Only one side with the cross sections needs to be created as we will use the Mirror function to replicate the created surfaces for the opposite end.
The plan section lines AA BB CC are to be drawn on the construction layer.
The sectional views are created WITHOUT the 5mm flat, as it is NOT possible to create a surface containing a sharp edge.

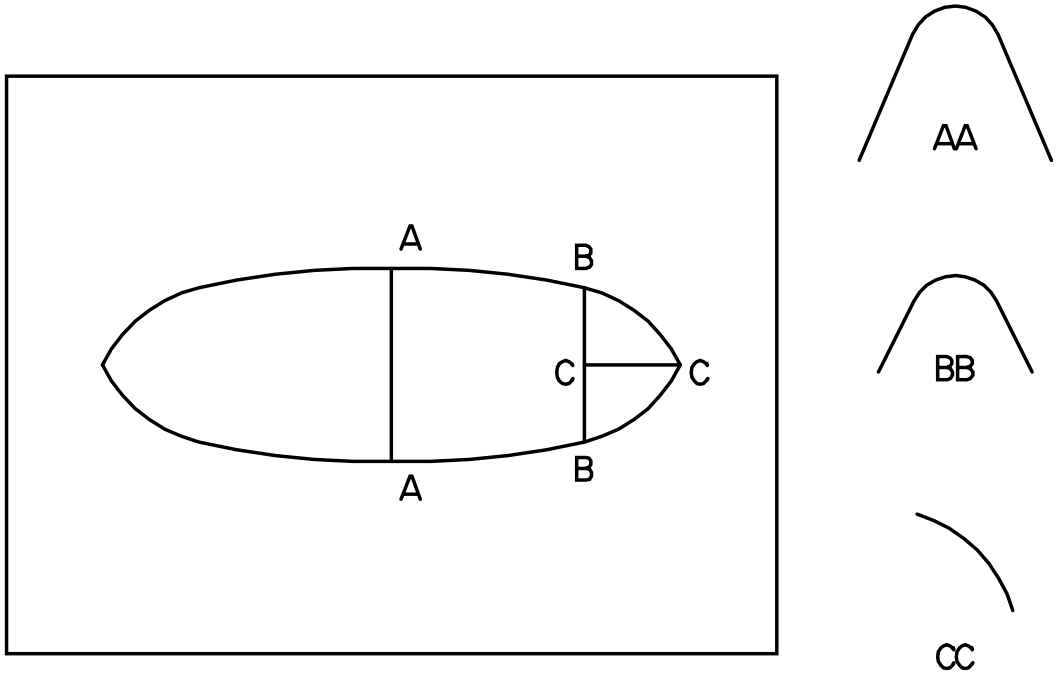


Figure 103 - Surface creation example 2 cross section information

Create a Work Volume for visualisation only

Using **GEOMETRY > Rectangle**  and create a rectangle from **X0, Y0** to **X200, Y150**

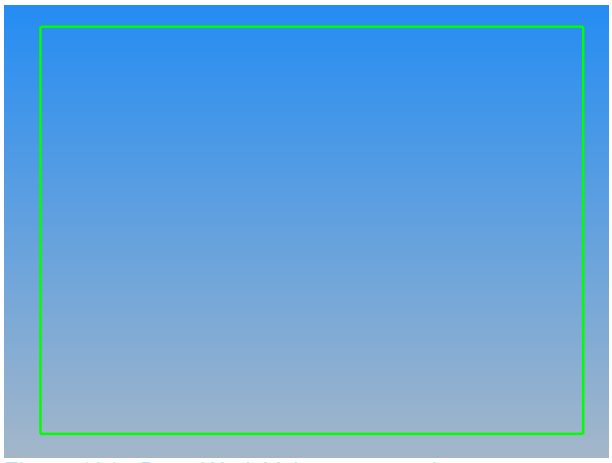


Figure 104 - Base Work Volume rectangle

Using **3D > Set Work Volume**  **<LClick>** the rectangle and set the work volume top to **50** and bottom to **0.**

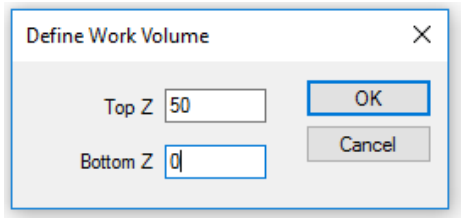


Figure 105 - Work Volume dialogue options

Select **VIEW > Views > Iso**  .

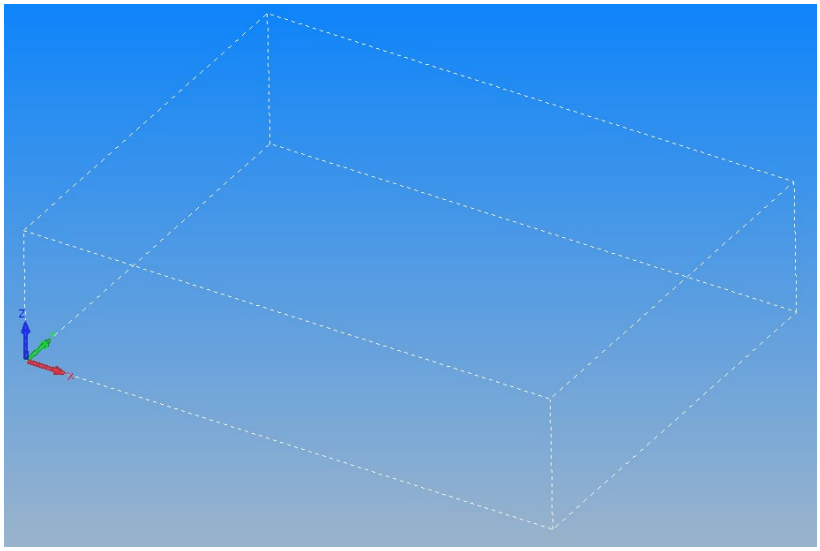


Figure 106 - Iso view of the work volume

Create the base geometry shapes

Construction Guide lines

To make creation of the required arcs for the base geometry it is easier to create a number of Construction guide lines to assist in location choices for the arcs start and end points and the generation of the required Work Planes.

Use **GEOMETRY > Construction**  to turn on this drawing mode.

Use **GEOMETRY > Line**  and create two central guides across the bottom of the Work Volume, having AutoSnap turned on, **<F2>** will make the process easier.

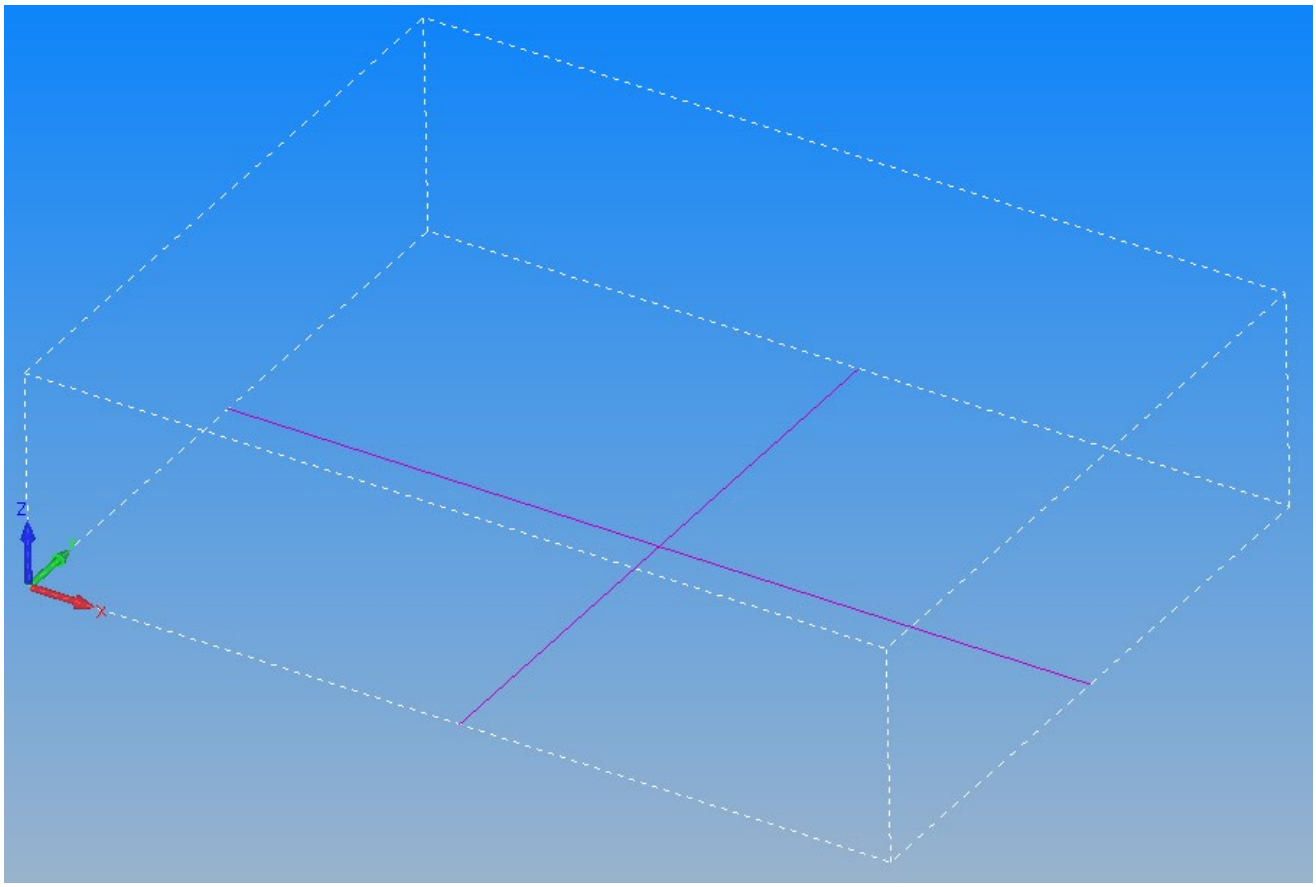


Figure 107 - Mid point guide lines

Using EDIT > Offset apply the following values to create an offset guide for the left side arc end.

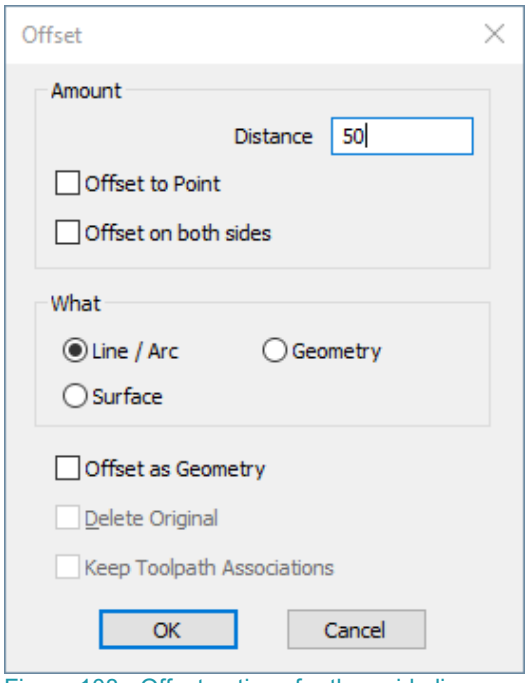


Figure 108 - Offset options for the guide line

<LClick> [OK] the **<LClick>** the middle line going across the **Y** direction of the part, **<LClick>** to the left side to place the new geometry.

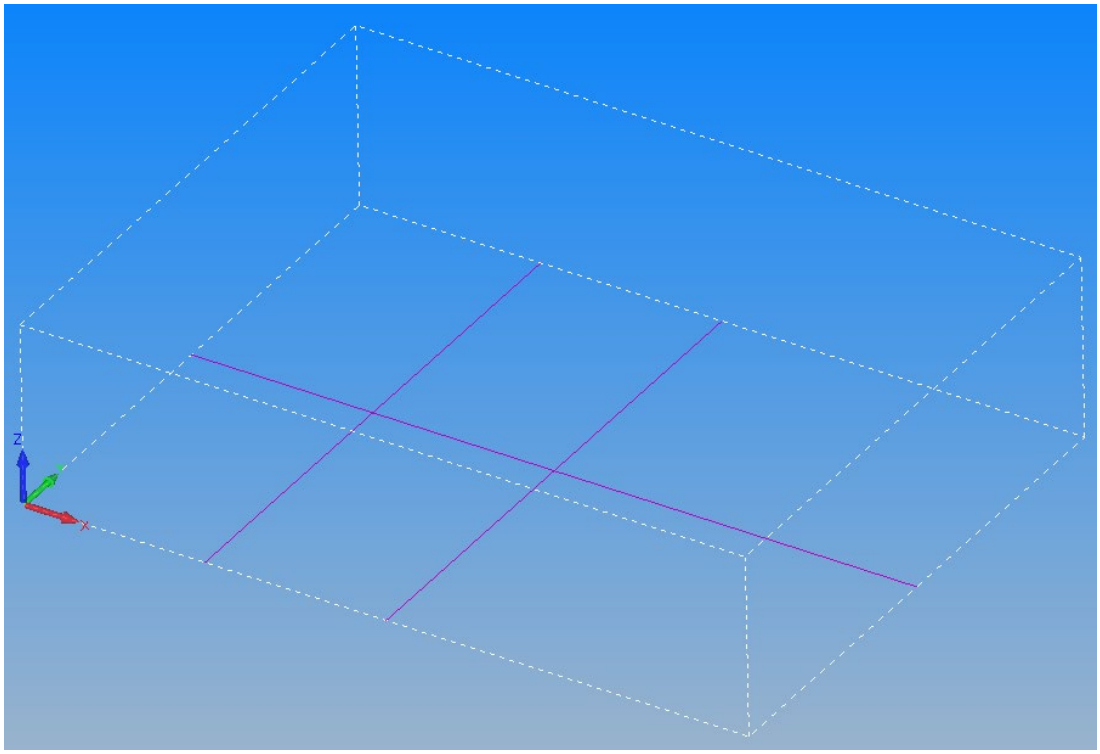


Figure 109 -First arc guide lines

Using **EDIT > Offset** again, once with an offset value of 25 selecting the construction line running along the part in the X direction and placing the new geometry above the line, then use the Offset command once more with a value of 20. Select the same line and place the new geometry also above the original central line. See below for the finished placements.

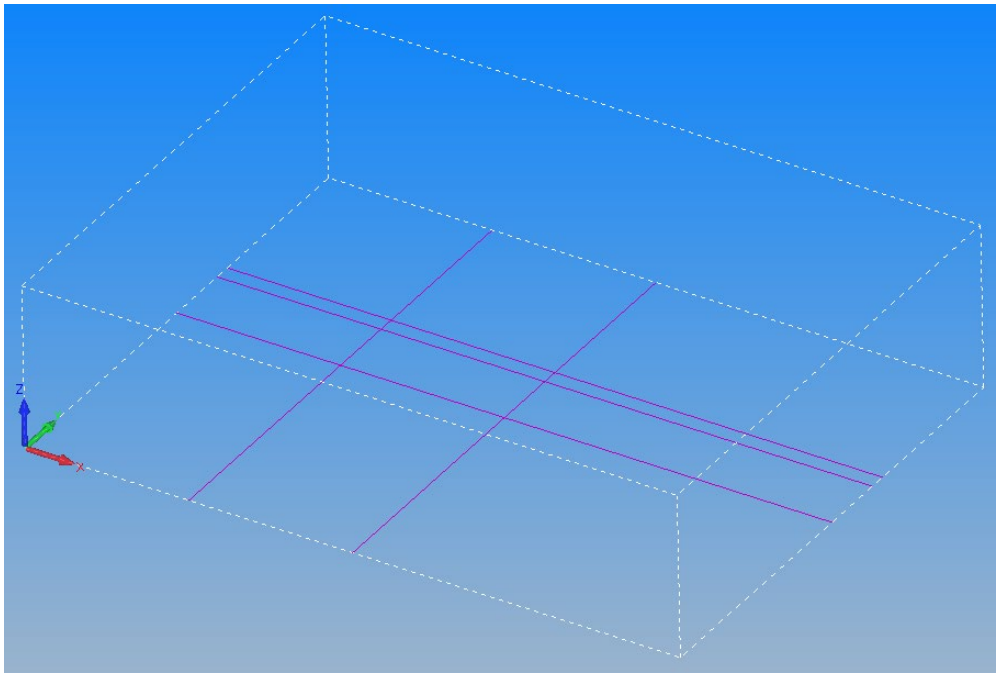


Figure 110 - Above centre guide lines

Using **EDIT > Offset** one final time with an offset value of 75 selecting the construction line running across the part in the Y direction and placing the new geometry to the left of the line as a guide for the left end of the part.

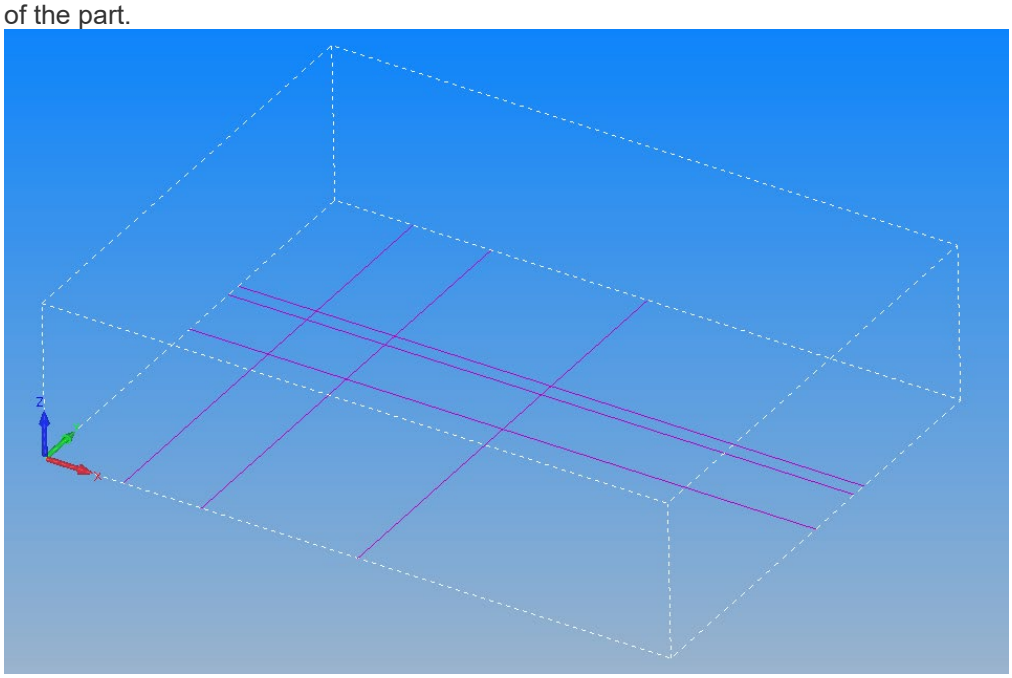


Figure 111 - Final guide in place

Arc Creation

Select **VIEW > Views > XY**  to make the next process easier to see.
All arcs are created in a Counter Clockwise direction which is why we have drawn the guides to the left side of centre to make the next section easier.

Select **GEOMETRY > Arc > 2 Points and Radius** 

For the Start Point, use **Intersection of (F9)**  then select the vertical centre line and the furthest vertical offset line to identify this point.

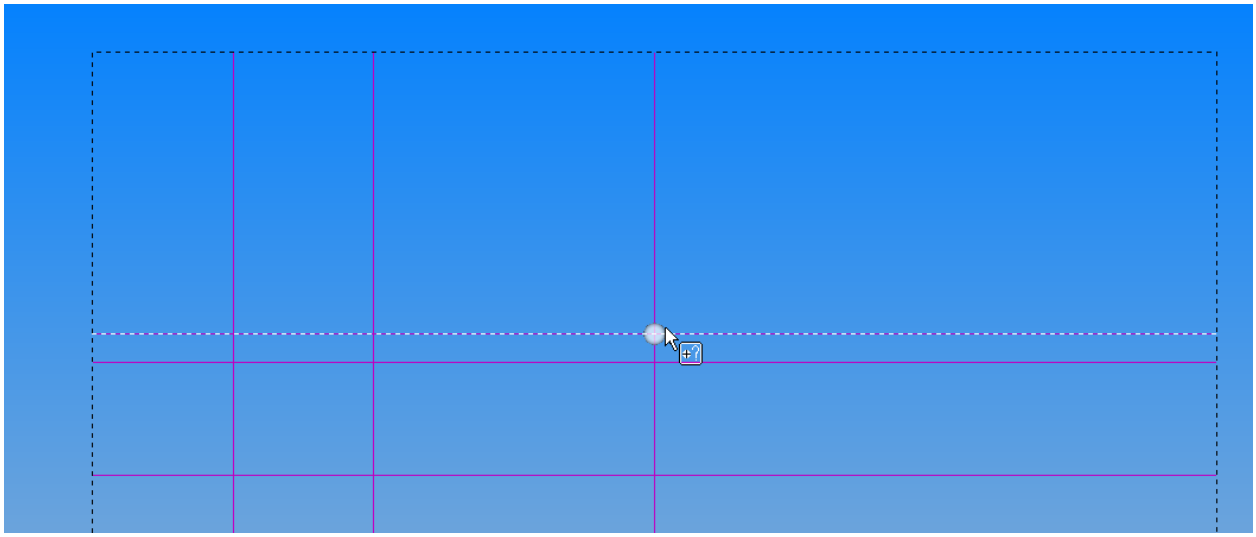


Figure 112 - Arc Start location

For the Finish Point, use **Intersection of <F9>**  then select the first left vertical offset line and the lower vertical offset line to identify this point.

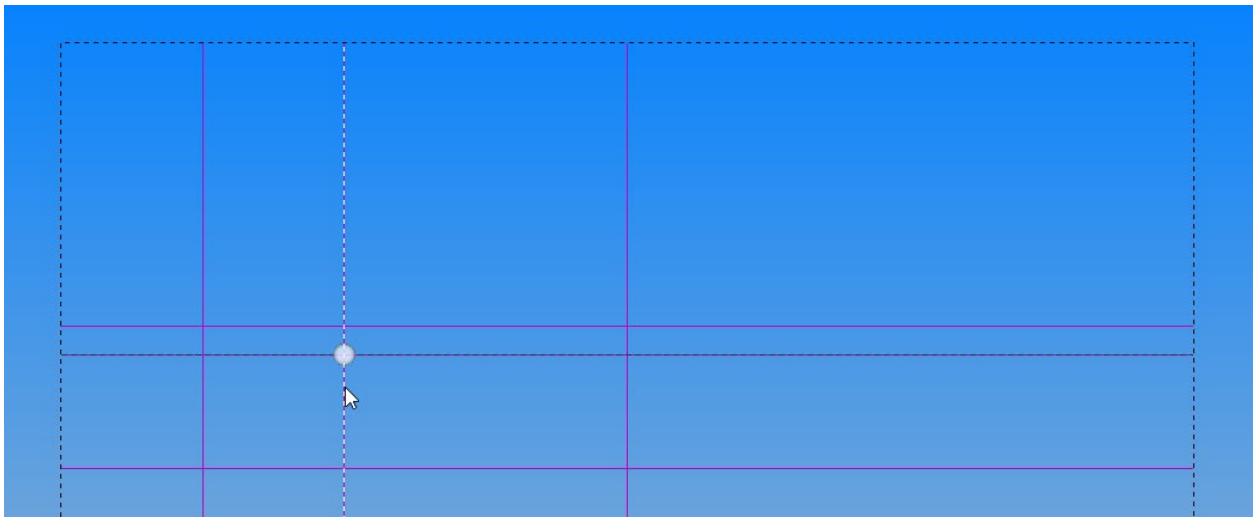


Figure 113 - Arc End location

When prompted, the **Arc Radius** is **200**

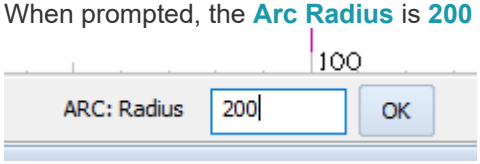


Figure 114 - Arc Radius value

<LClick> [OK] to apply.

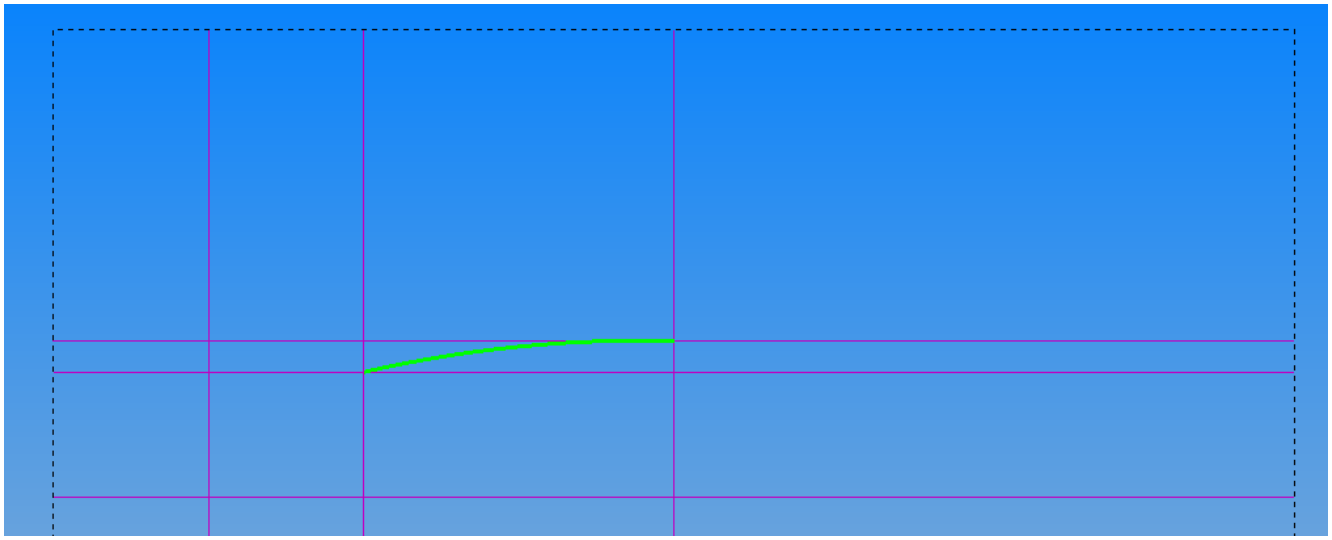


Figure 115 - First Arc section

As the Arc command is still active, use **END-point of <F6>** 
and select the left end of the previously created arc as the Start location.

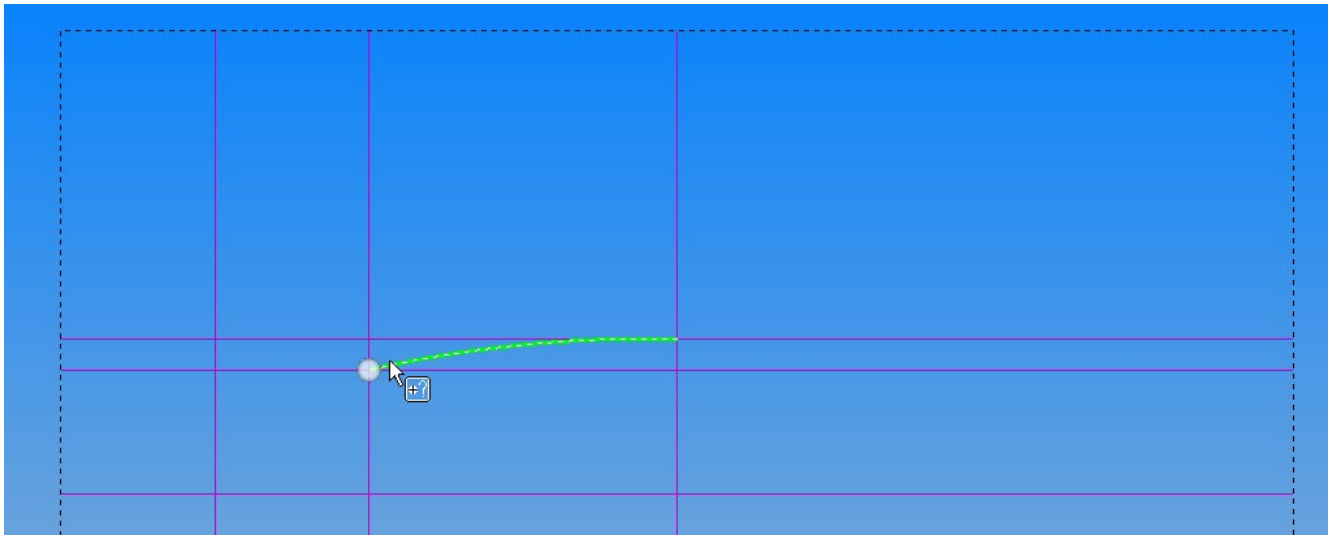


Figure 116 - Second Arc Start location

For the Finish Point, use **Intersection of <F9>**  then select the far left vertical offset line and the horizontal centre line to identify this point.

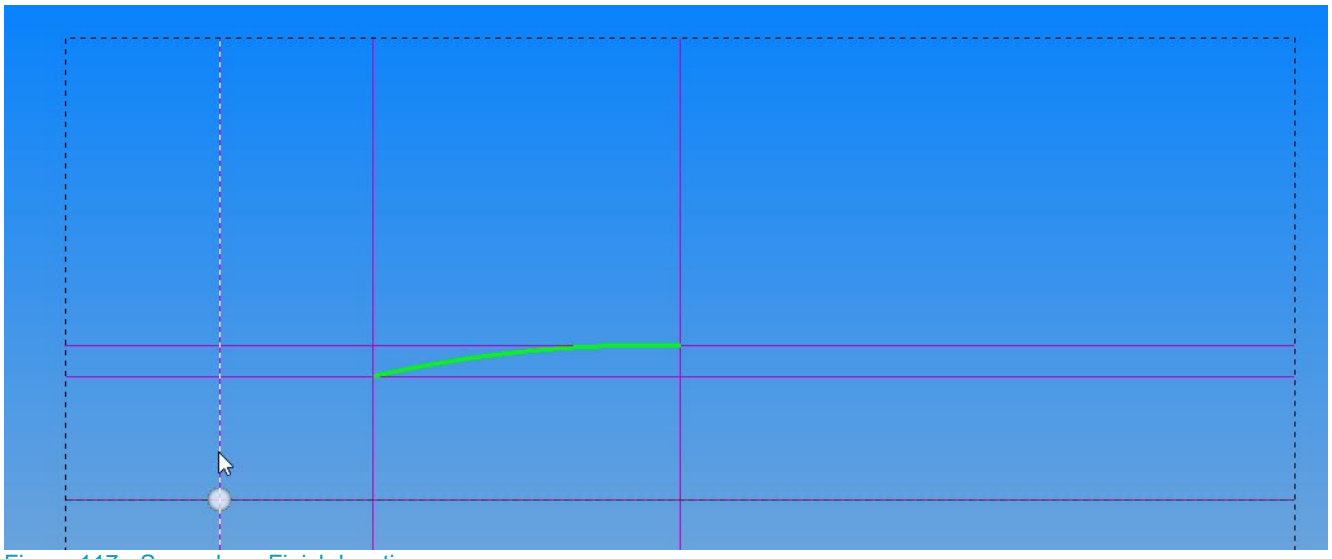


Figure 117 - Second arc Finish location

When prompted, the **Arc Radius** is **37**

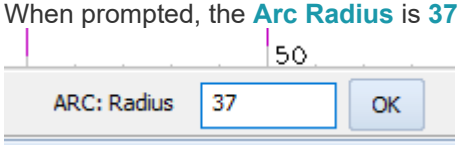


Figure 118 – Second Arc Radius value

<LClick> [OK] to apply.

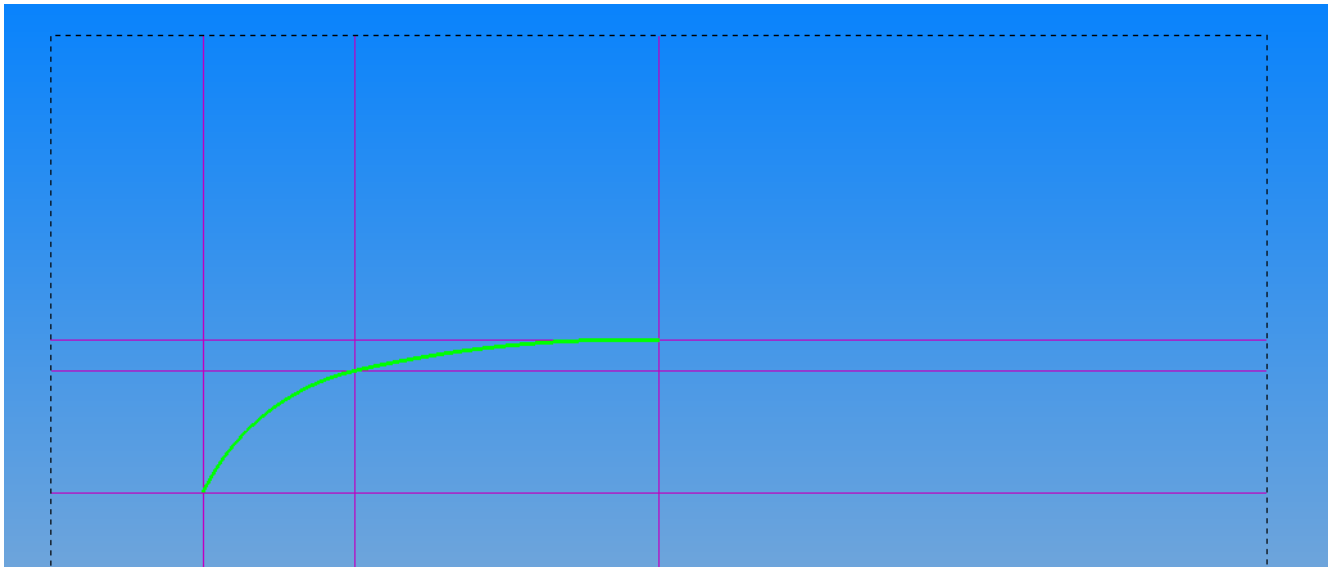


Figure 119 - Created second arc

<RClick> to finish the command.

Using **EDIT > Mirror**  **<LClick>** the two arcs as the items to mirror, then for the first and second points for mirror **<LClick>** each end of the central vertical line.
<LClick> [Yes] to keep the originals.

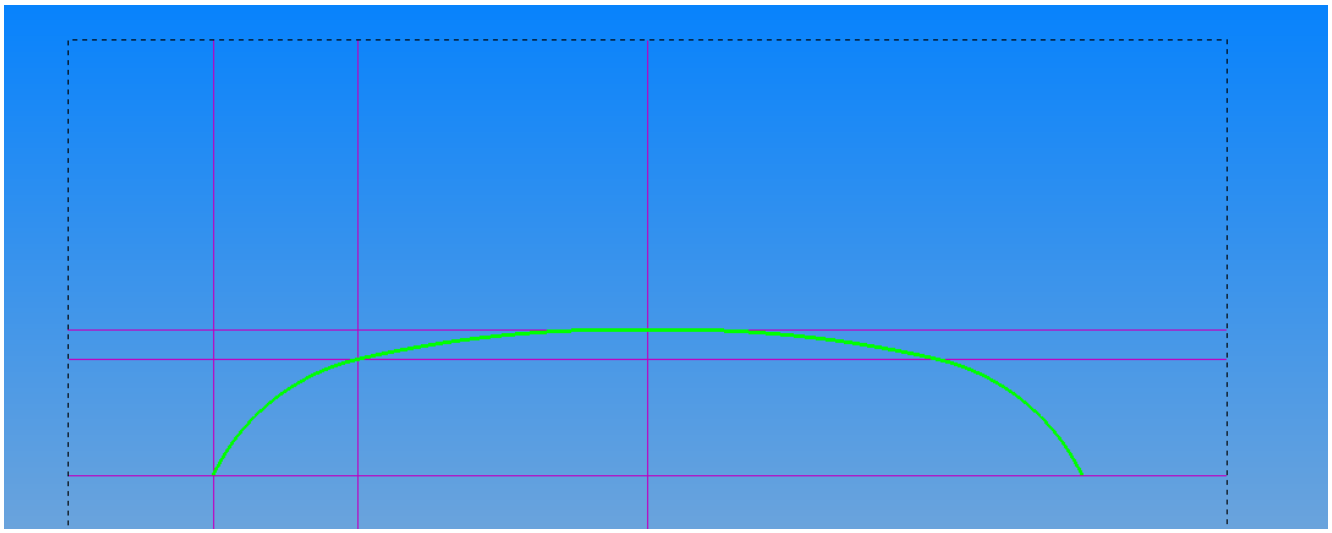


Figure 120 - First Mirrored set.

With the Mirror command still active, **<LClick>** all four arcs and use the ends of the horizontal centre line as the two points for mirror.

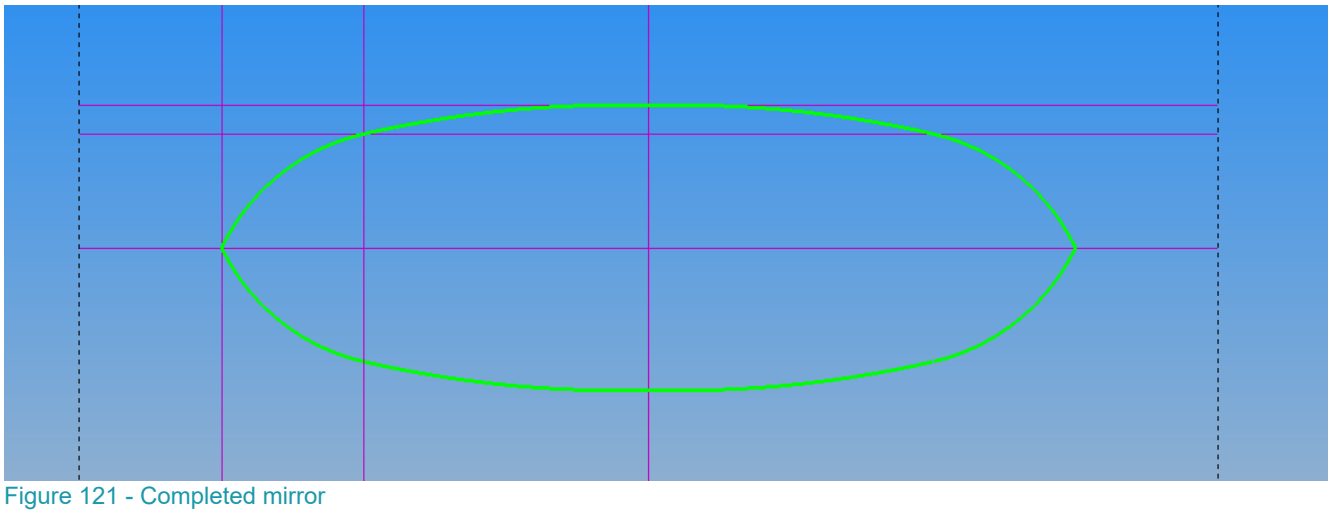


Figure 121 - Completed mirror

<RClick> to complete the command.

Cross Sections and Work Planes

Although the geometry may appear to be on the bottom of the work volume it is only 2D geometry and as such has no concept of Z values. To create the surfaces, it will be necessary to manipulate the geometries into position.
We will also need to create three work planes for the cross sections and their associated geometries.

Manipulation of the base geometry to its required location

As we will be forming a vertical side set of surfaces to create the lower section, we will need to have upper and lower geometry to create the surfaces from.
We have already created the lower geometry set, now we will need to copy this geometry up by 5mm to create the upper geometry set.

Select **VIEW > Views > Iso** .

Select **EDIT > Auto 2D 3D > 3D** this causes any manipulation to be a 3D manipulation.

Select **EDIT > Copy** then select the four arcs that we have previously drawn.

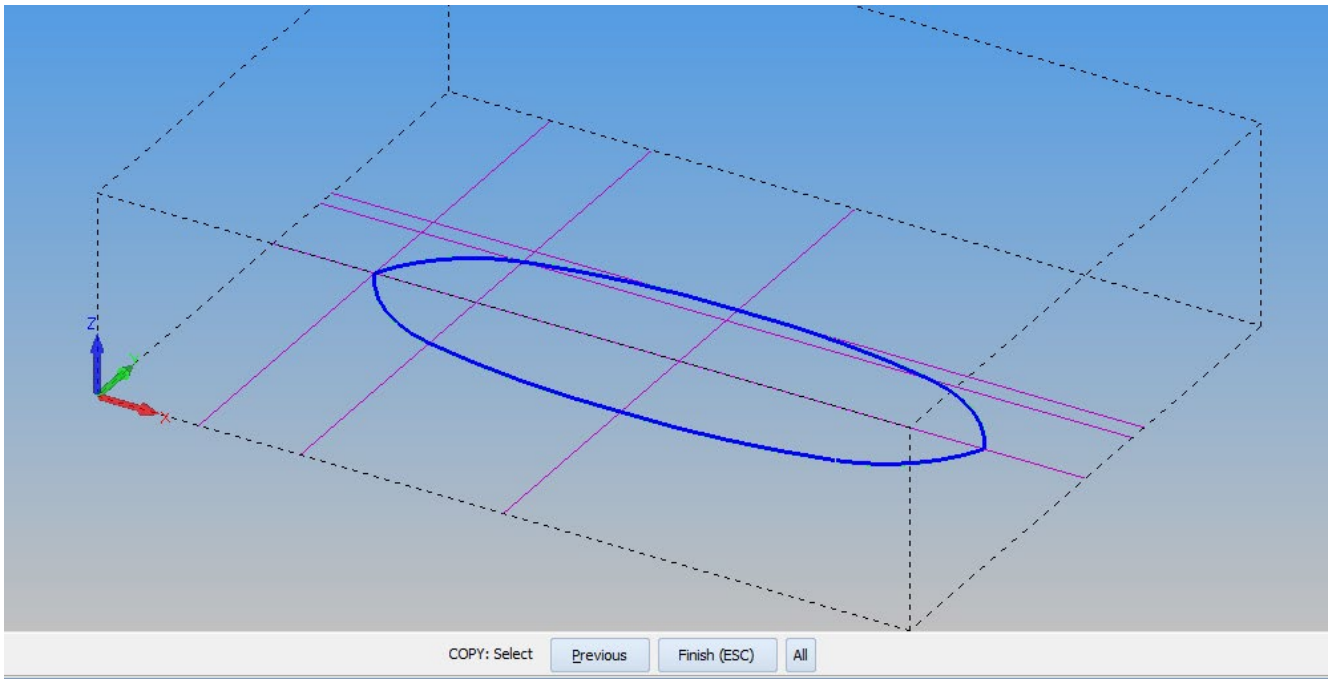
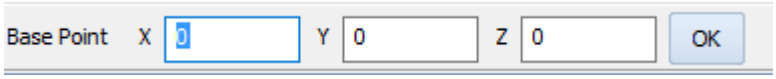


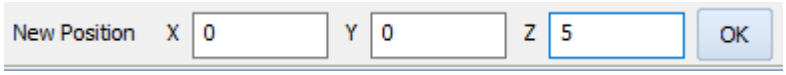
Figure 122 - Items for the 3D Copy

<RClick> or **[Finish (ESC)]** to continue.

For the Base Point use X**0, Y0, Z0**, **<LClick> [OK]**



then for the new position use X**0, Y0, Z5**. **<LClick>[OK]**



<RClick> twice to complete the process.

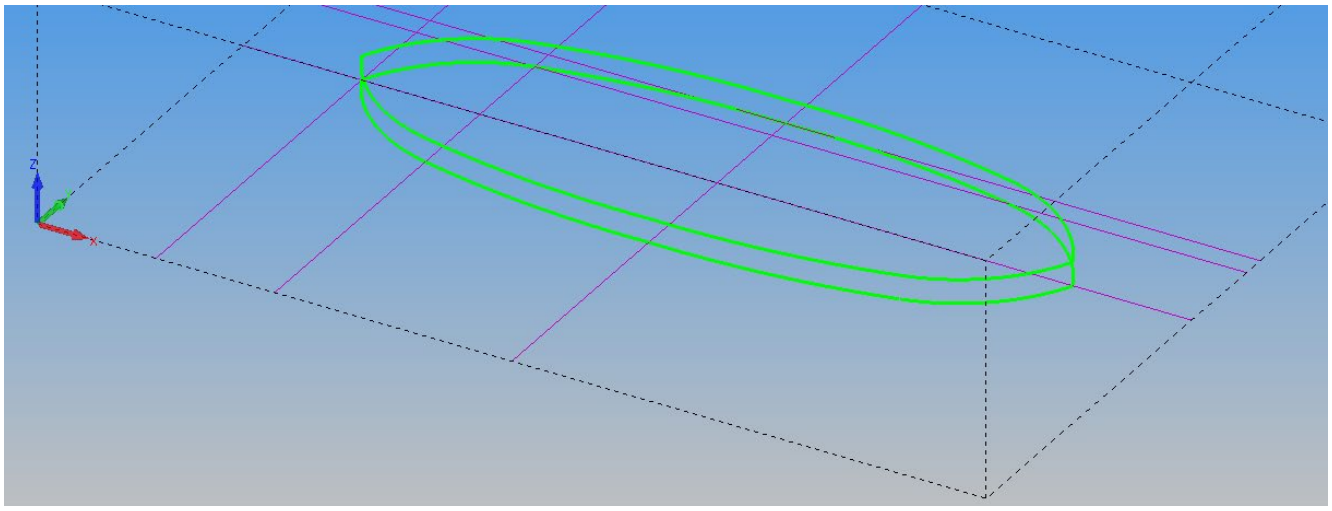


Figure 123 - 3D Copy complete

Select **EDIT > Auto 2D 3D > Auto**  to set the manipulation method back to default.

Work Plane cross section A-A

Using **WORK PLANES > Slice through work volume**  **<LClick>** on the vertical centre line to create a new Work Plane for the central cross section A-A.

Use **WORK PLANES > Set Work Plane Origin**  to move the original left hand side origin position to the required central location as shown below. You will need to use **MID-point of <F7>** to pick the required location on the construction line, as it is on a different work plane Auto Snap will not work.

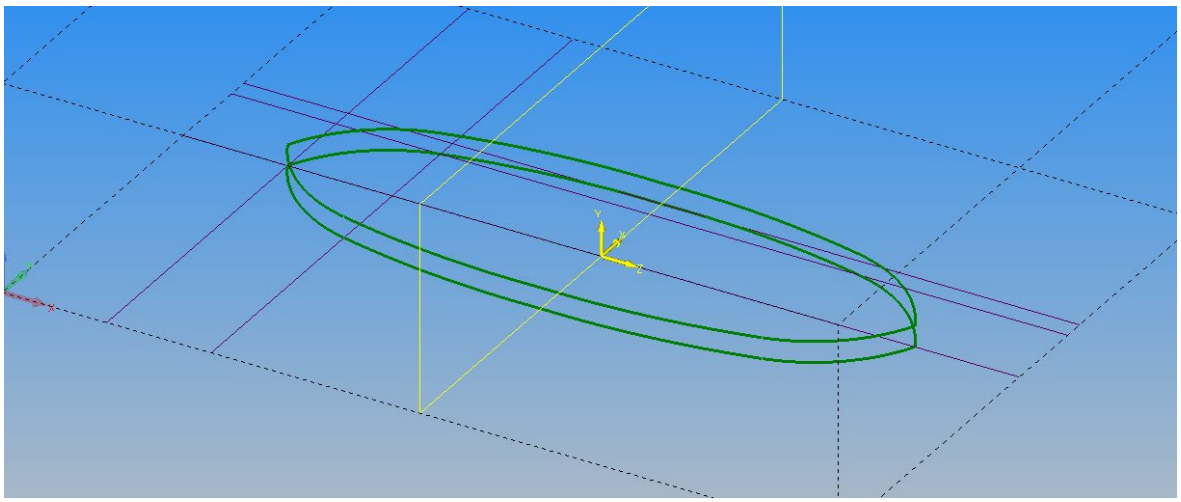


Figure 124 - Work Plane with correct origin location

On the **Work Planes** tab of the Project Manager, **<RClick>** on the new work plane and select Properties.

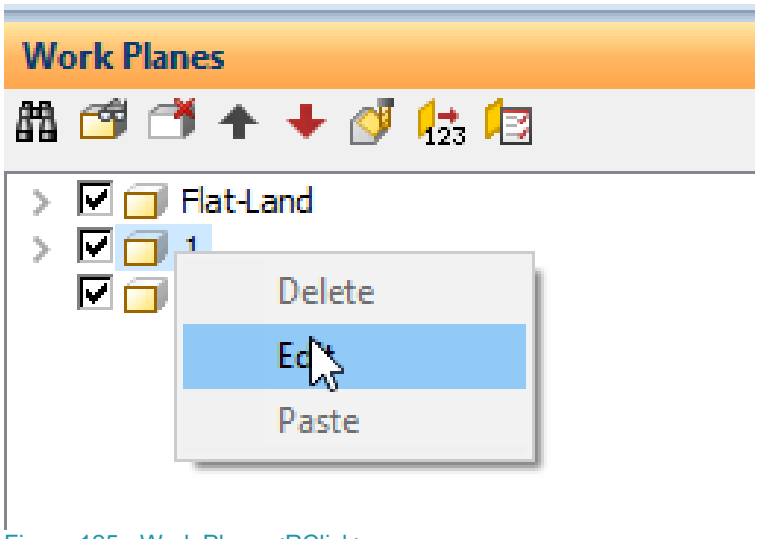


Figure 125 - Work Plane <RClick> menu

Rename the work plane to **A-A**.

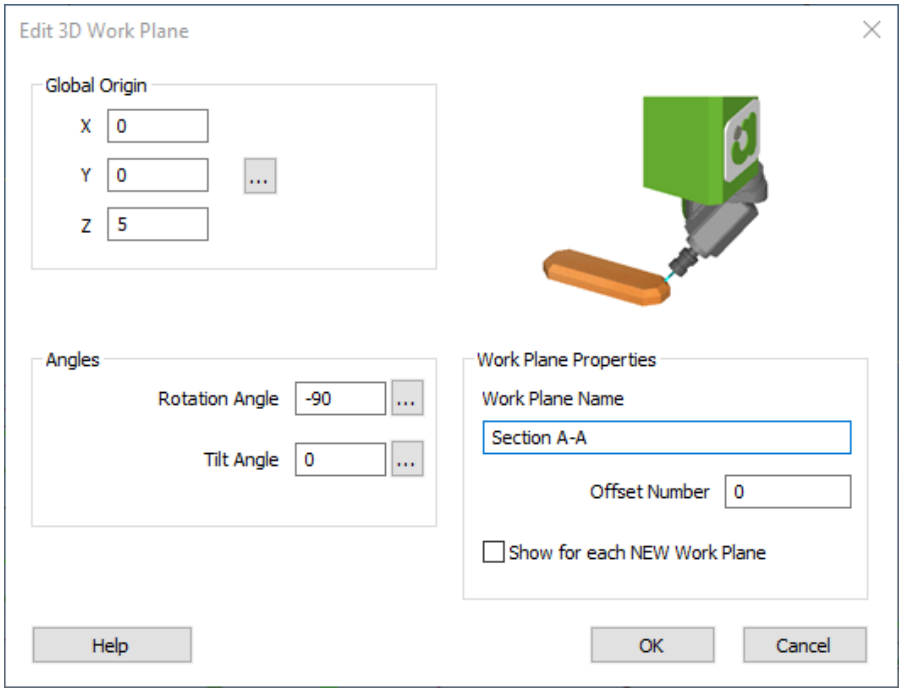


Figure 126 - Work Plane Edit dialogue

A second method to rename is to use the Properties options at the top of the Work Planes Project Manager page.

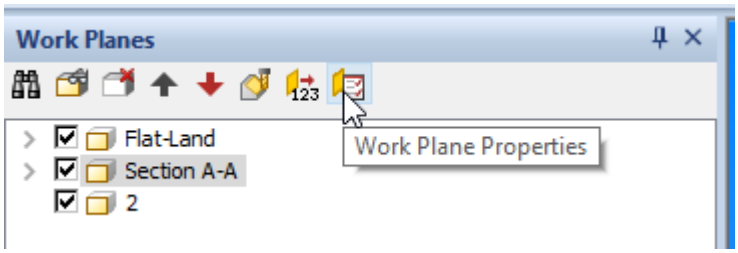


Figure 127 - Work Planes Properties on Project Manager

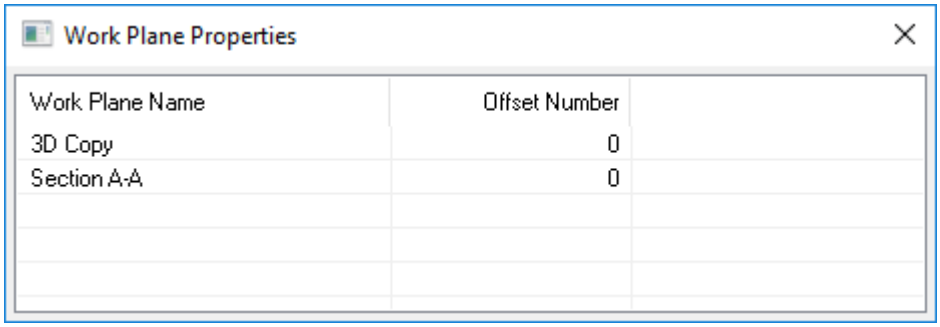


Figure 128 - Work Plane Names in Properties

Using the Properties option allows for renaming many work planes at once.

Renaming Work Planes allows for ease of identification on complex parts.

Use **VIEW > Work Plane**  to look directly at the active work plane.
Then using any suitable methods create the following geometry set.

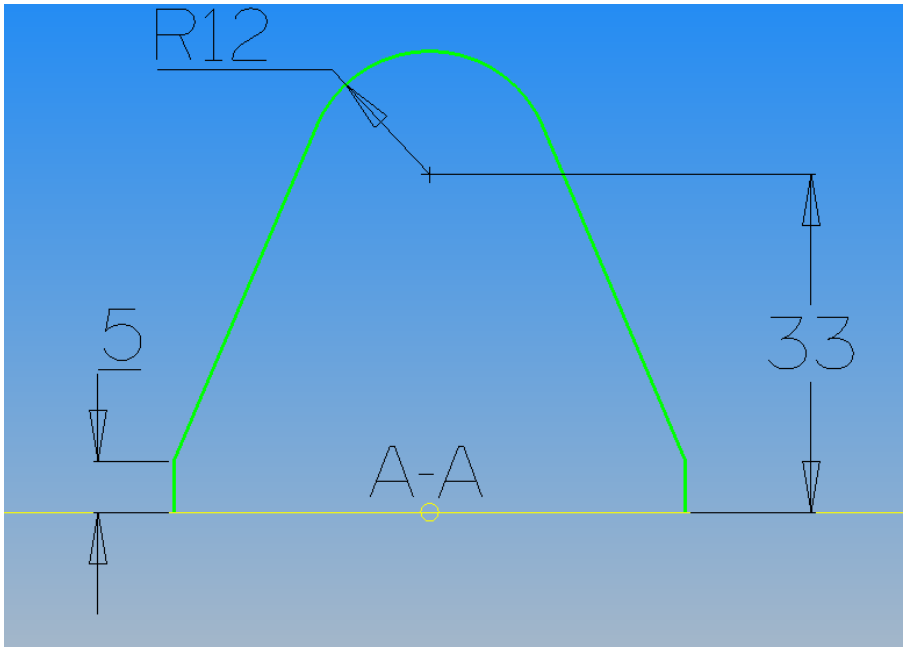


Figure 129 - Geometry set for cross section A-A

Select **VIEW > Views > Iso**  .

Select **WORK PLANES > Cancel Work Plane** 

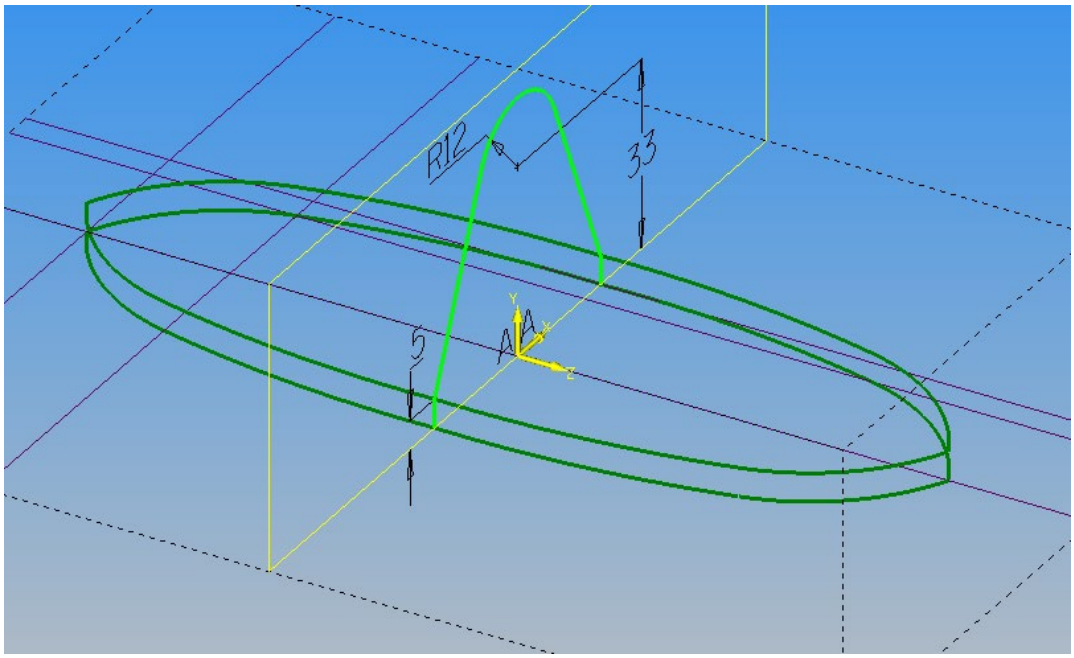


Figure 130 - Created cross section A-A

Work Plane cross section B-B

Using **WORK PLANES > Slice through work volume**  **<LClick>** on the first left hand offset from the centre line to create a new Work Plane for the central cross section **B-B**.

Use **WORK PLANES > Set Work Plane Origin**  to move the original left hand side origin position to the required central location as shown below. You will need to use **MID-point of <F7>** to pick the required location on the construction line, as it is on a different work plane Auto Snap will not work.

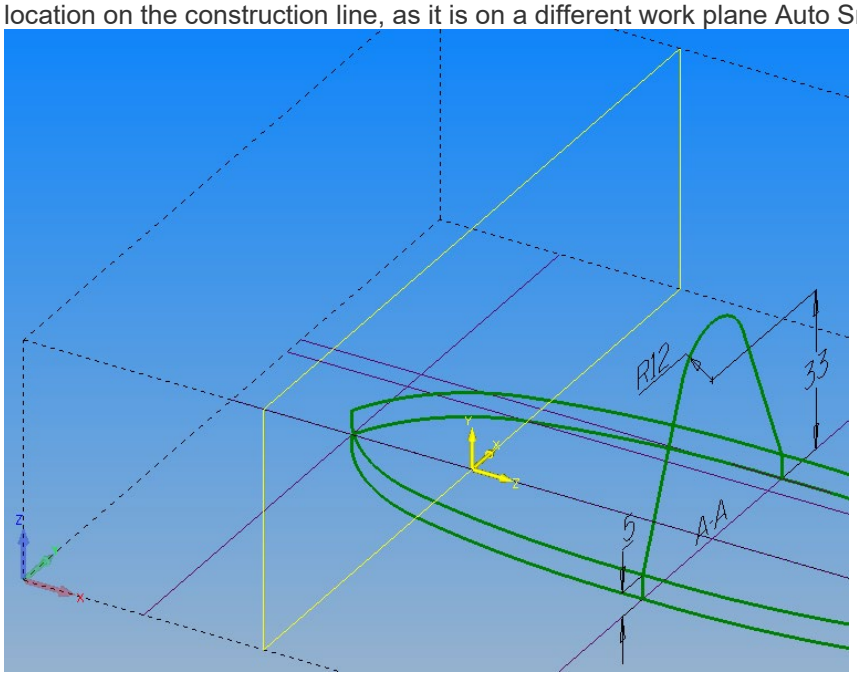


Figure 131 - Work Plane with correct origin location

On the **Work Planes** tab of the Project Manager use the Properties command to rename the work plane.

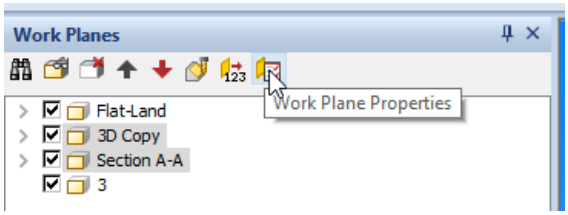


Figure 132 - Work Plane Properties command

Rename the work plane to B-B.

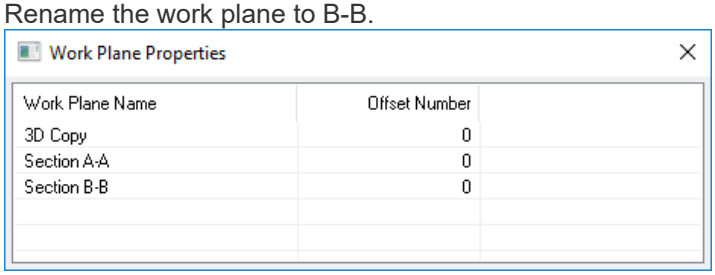


Figure 133 - Work Plane Properties dialogue

Renaming Work Planes allows for ease of identification on complex parts.

Use **VIEW > Work Plane**  to look directly at the active work plane.
Then using any suitable methods create the following geometry set.

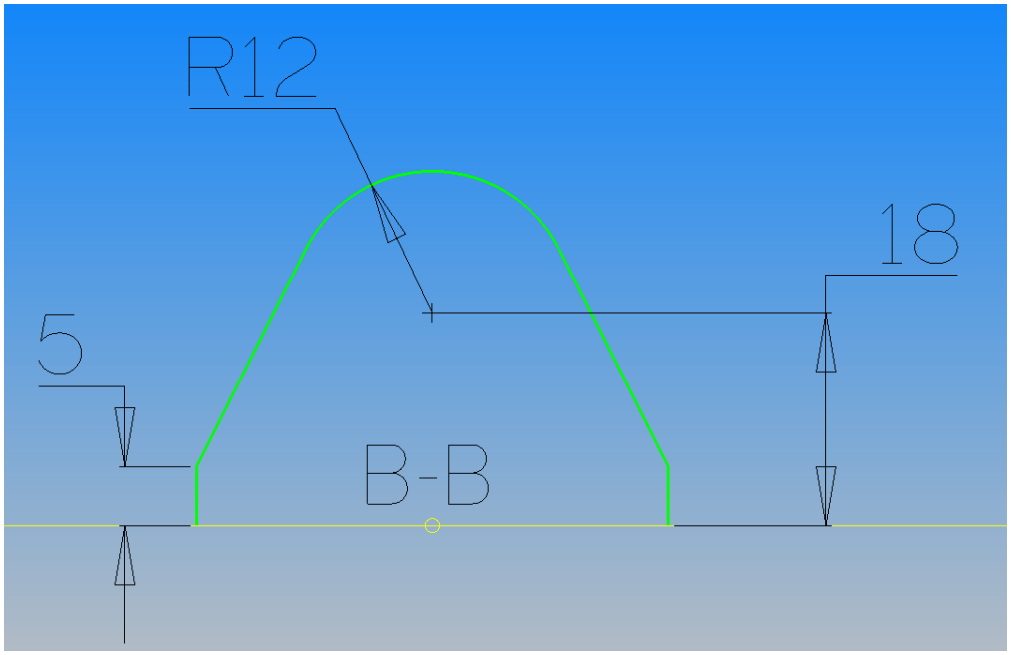


Figure 134 - Geometry set for cross section B-B

Select **VIEW > Views > Iso**  .

Select **WORK PLANES > Cancel Work Plane** 

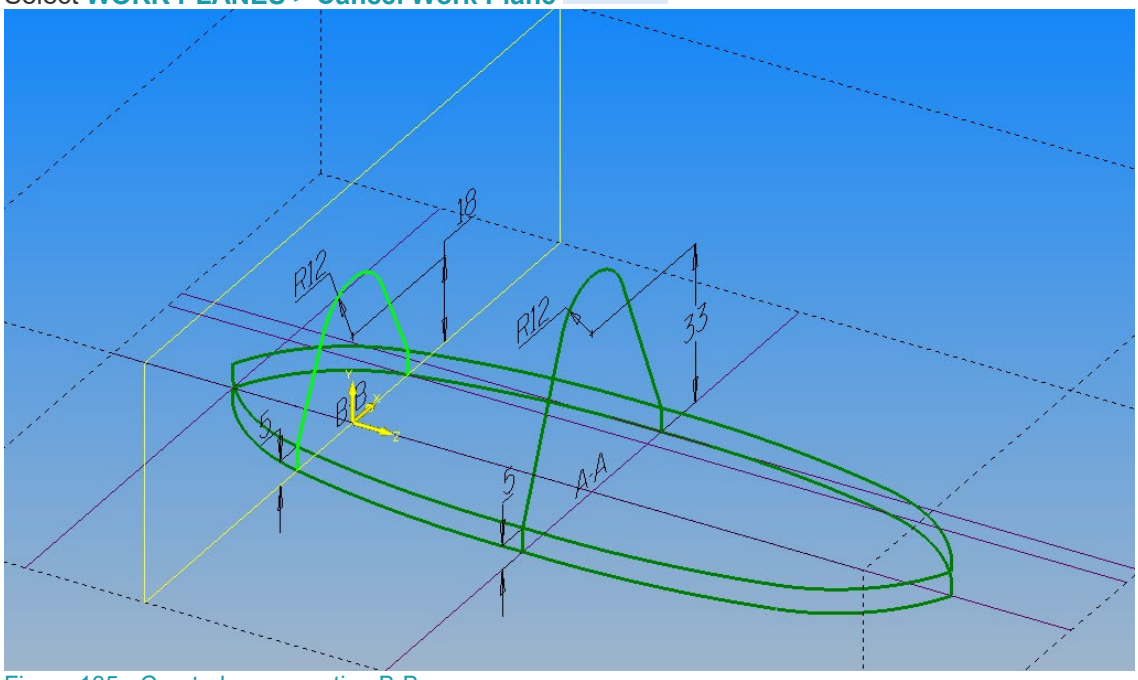


Figure 135 - Created cross section B-B

Work Plane cross section C-C

Using **WORK PLANES > Slice through work volume**  **<LClick>** on the horizontal centre line to create a new Work Plane for the central cross section **C-C**.

Use **WORK PLANES > Set Work Plane Origin**  to move the original left hand side origin position to the required central location as shown below. You will need to use **END-point of <F6>** to pick the required location on the arc, as it is on a different work plane Auto Snap will not work.

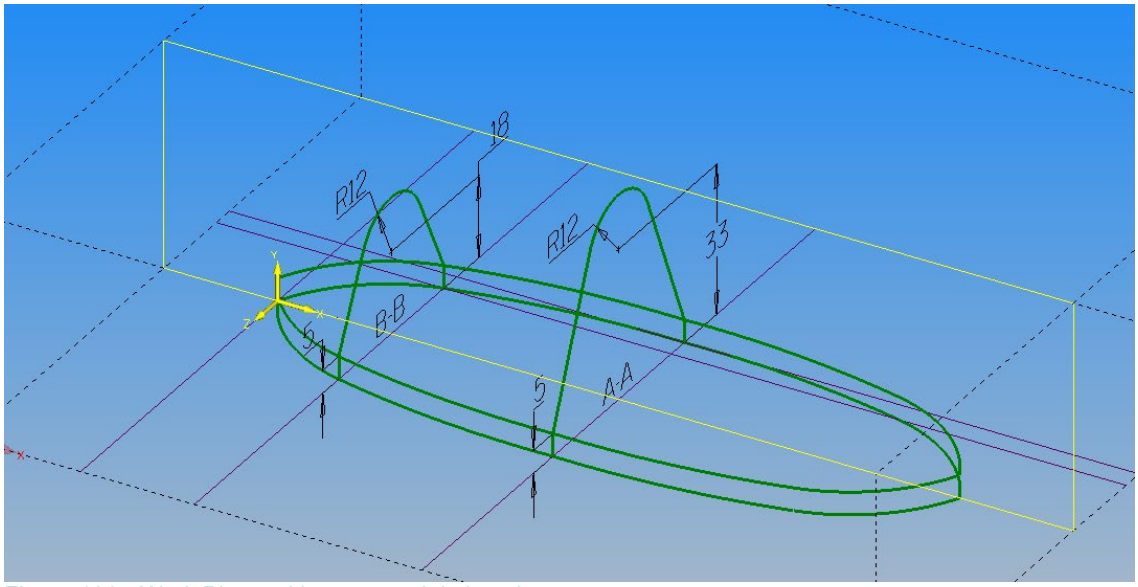


Figure 136 - Work Plane with correct origin location

On the **Work Planes** tab of the Project Manager use the Properties command to rename the work plane.

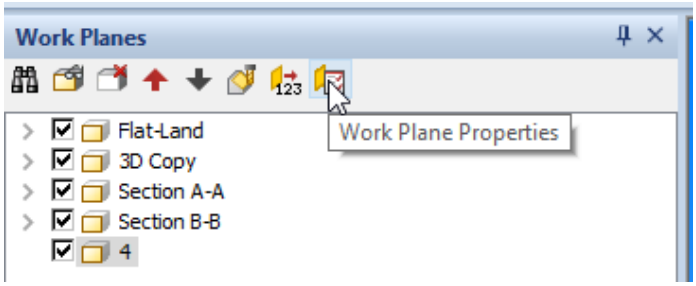


Figure 137 - Work Plane Properties command

Rename the work plane to C-C.

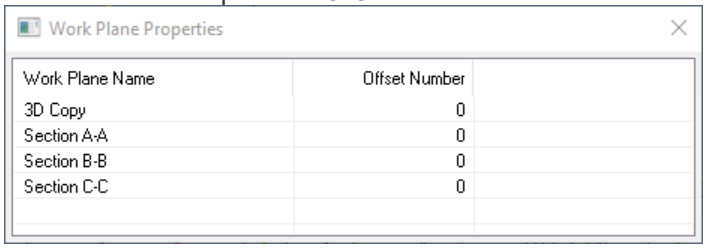


Figure 138 - Work Plane Properties dialogue

Renaming Work Planes allows for ease of identification on complex parts.

Use **VIEW > Work Plane**  to look directly at the active work plane.
Then using any suitable methods create the following geometry set.

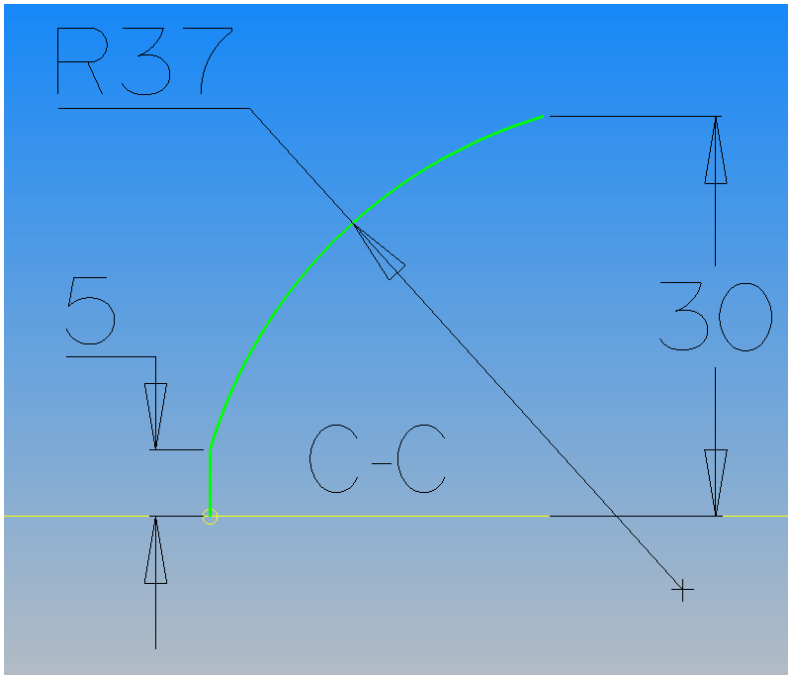
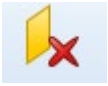


Figure 139 - Geometry set for cross section C-C

Select **VIEW > Views > Iso**  .

Select **WORK PLANES > Cancel Work Plane** 

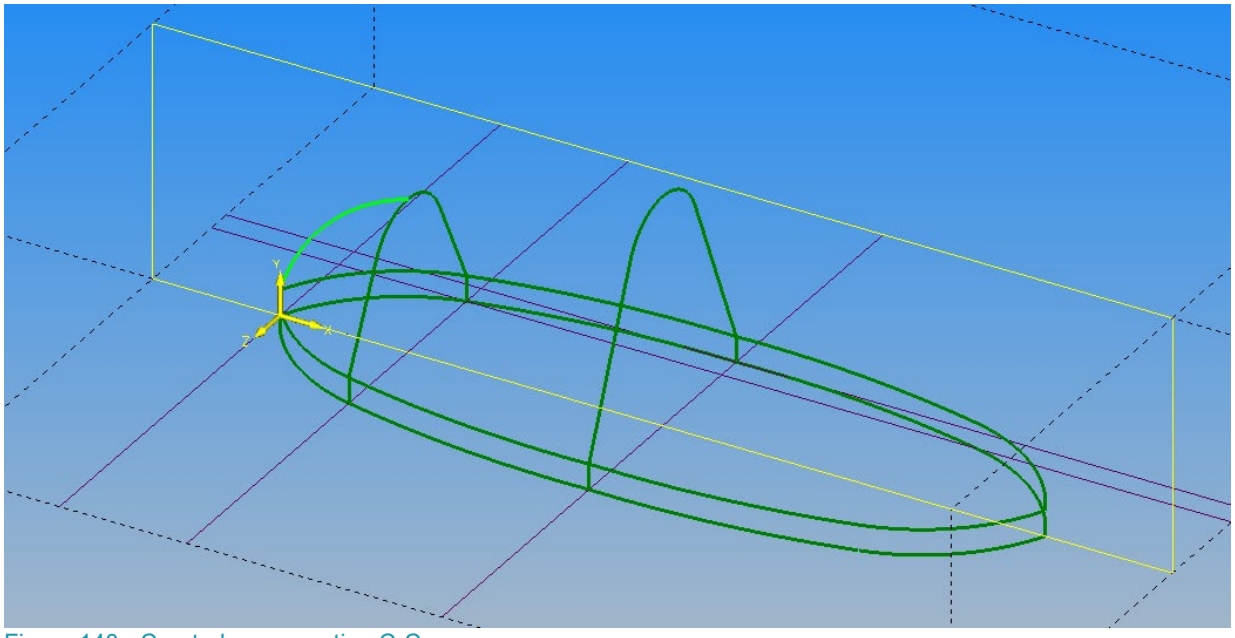


Figure 140 - Created cross section C-C

Preparing the profiles for Surface creation

The types of surfaces we will need to create to from the finished multi-curved form are called **Coons Patches**.
These surfaces require specific geometry set ups to allow the surface to be created.
All geometries which will form the boundaries must have their end points coincident with each other or the surface cannot be formed.

All the following images have had the unrequired geometries hidden using **CAD > Hide Parts**
Or the complete work plane has been unticked in the case of the Flat-Land work plane.

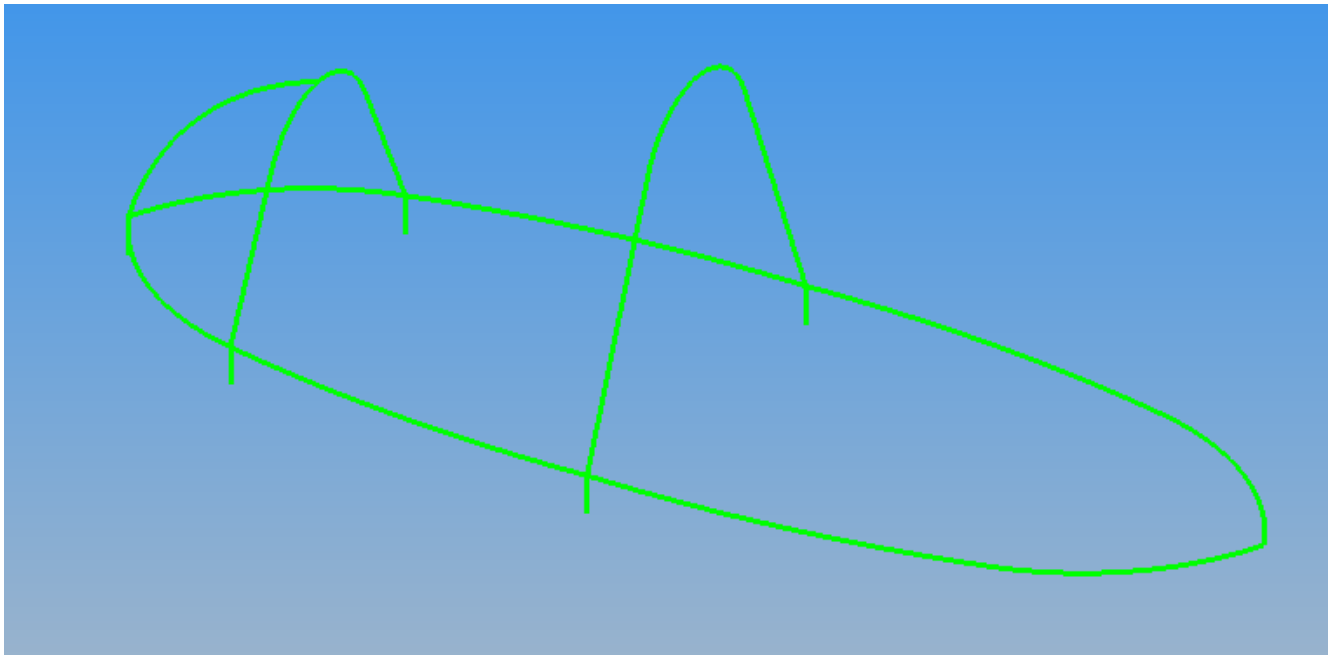


Figure 141 - Geometries for surface creation

Just to clarify, the small 5mm vertical lines are only shown to indicate which main profile is in use, it is not the one on **Flat Land.**

Making complete A-A and B-B profiles

To create this first surface, depending on how you created the geometries, it may be necessary to join together the top arc sections and the tapered lines on each work plane to form a complete single geometry for each cross section.
The main lower 200mm arc will also need to be a single piece geometry and not joined to the smaller end 37mm arc.

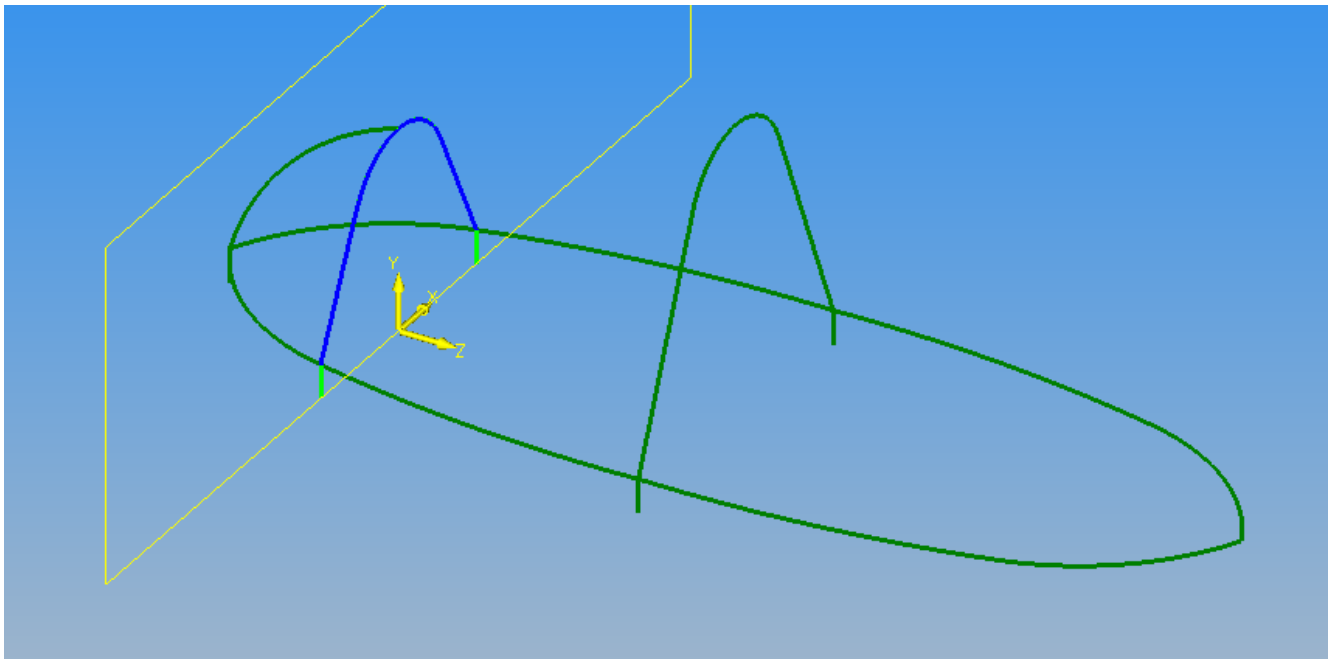


Figure 142 - Join together geometries as required

Using **EDIT > Join**  make complete entities on both **A-A** and **B-B** work planes.

> Note that you do not need to activate the required work plane to join the geometries, the action of **<LClick>** on a geometry during this process will automatically activate the plane on which the geometries are drawn.

To make things clearer, use **WORK PLANES > Cancel Work Plane** 

Surface between section A-A and B-B

Select **GEOMETRY > 3D Surfaces** and the surface creation dialogue is displayed.

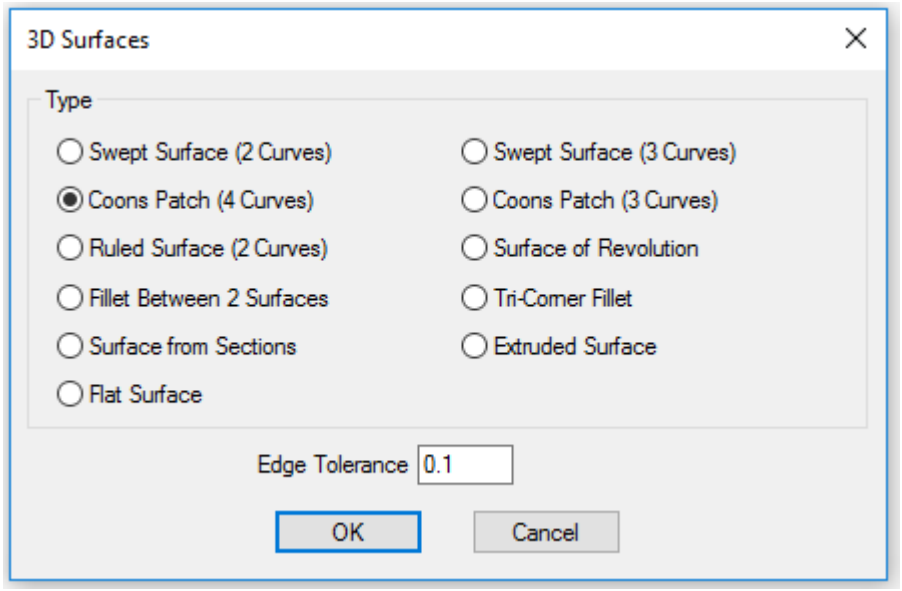


Figure 143 - Surface creation options

Select **Coons Patch (4 curves)** then **<LClick> [OK]**.
When prompted, select the sections **A-A** and **B-B** and the 200mm radius base profile curves connecting them. The prompt will ask for Edge1, Edge2, Edge3 and Edge 4, in this particular case the order is not important as the shape of the cross sections are uniform, however, the order of choice can have an impact on the finished surface.

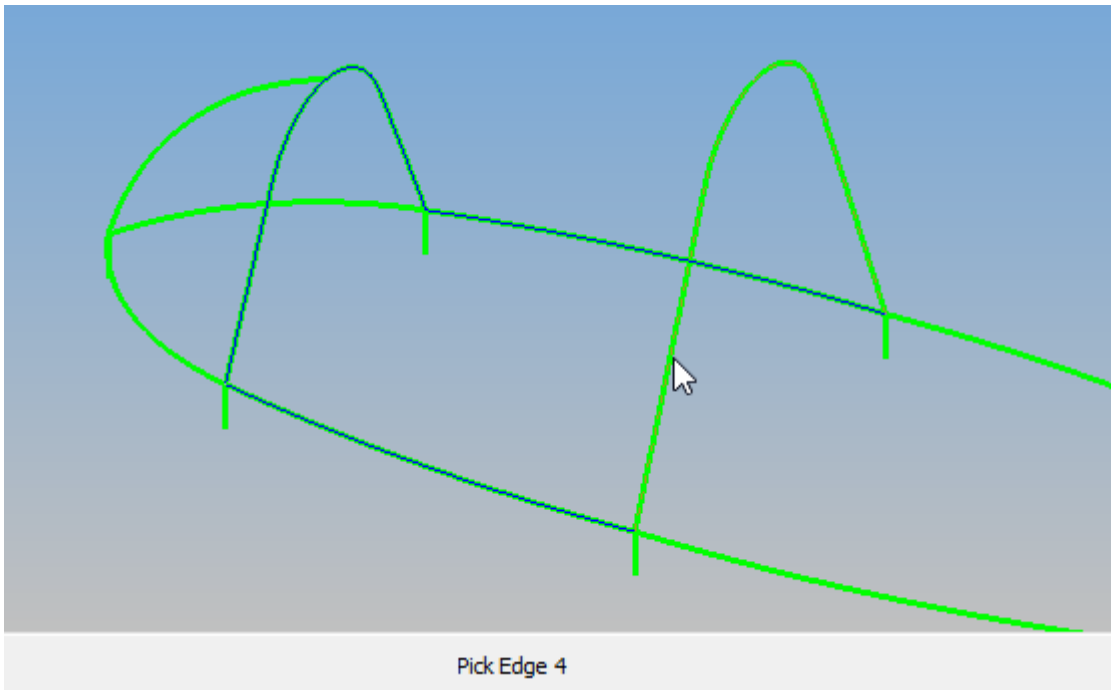


Figure 144 - Selected geometries for patch surface

Once you have chosen the four edges, the surface precision option is displayed.
The default options will always create a suitable surface, but if you need more accuracy for machining, then having more control lines is beneficial.

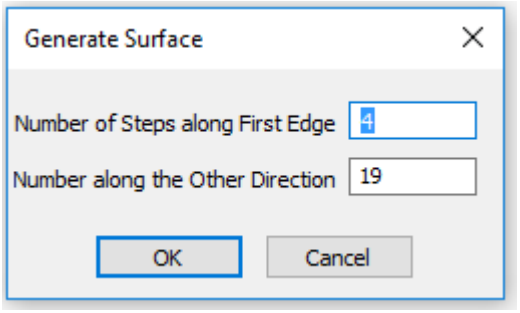


Figure 145 - Surface quality options

<LClick> [OK] to complete the process.

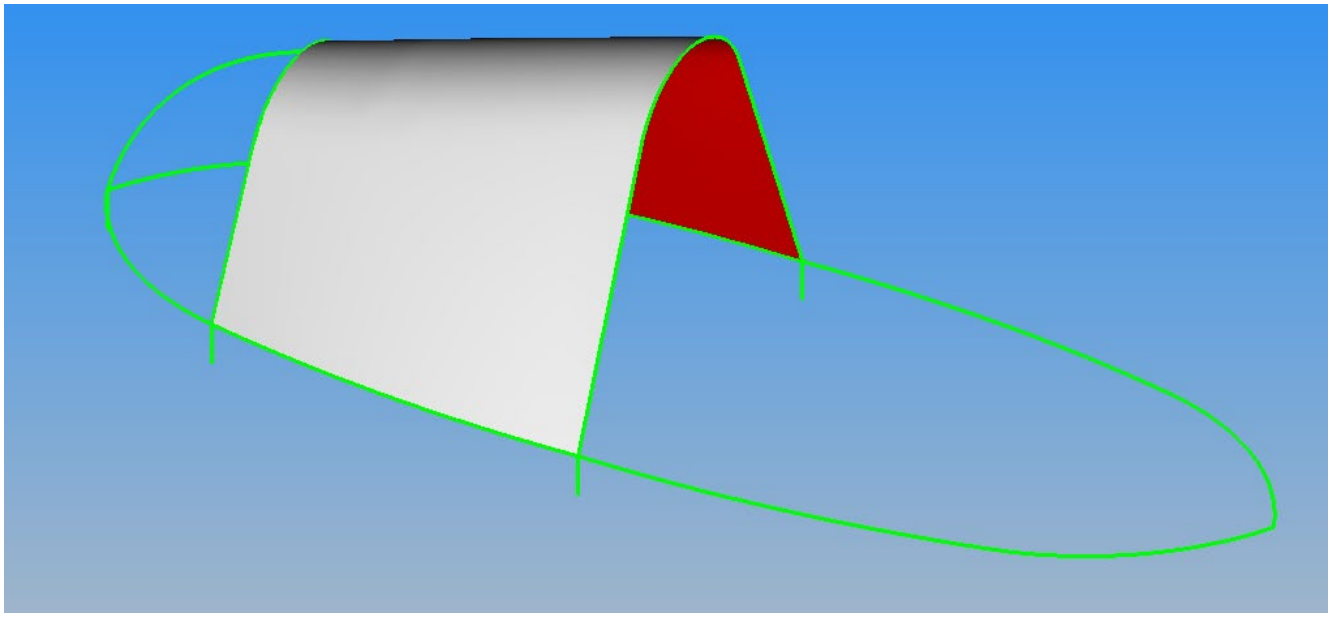


Figure 146 - Completed first surface

Making suitable profiles for section C-C surfaces

To create the end surface sections, we will be using the **Coons Patch (3 Curves)**.
To make the surfaces correctly we will need to manipulate the previous geometries to suit a three sided surface, this will involve breaking the **B-B** cross section geometry into two sections at the mid-point of the arc and also making sure that the base profile 37mm arcs are two separate entities.

Using **EDIT > Break**  and setting the options to ⊙ **Geometry** and ⊙ **Individual**.

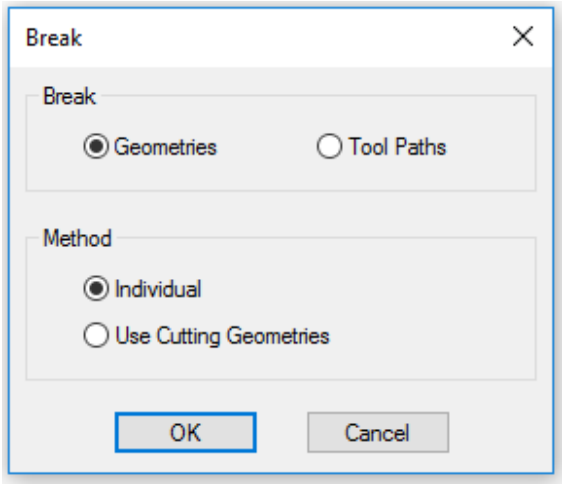


Figure 147 - Break command options

Using **MID-point of <F7> <LClick>** the mid point of the arc on cross section **B-B**.

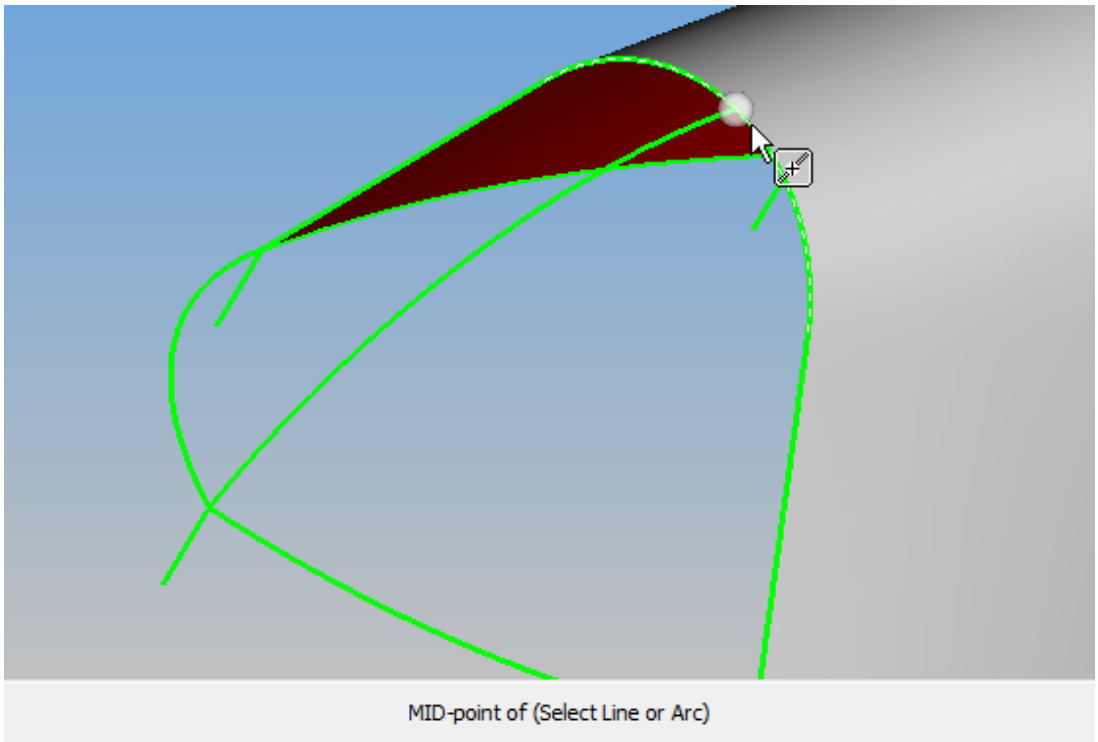


Figure 148 - Breaking the arc mid point

Then using **END-point of (F6) <LClick>** the end of the 37mm front arc. **<RClick>** to complete.

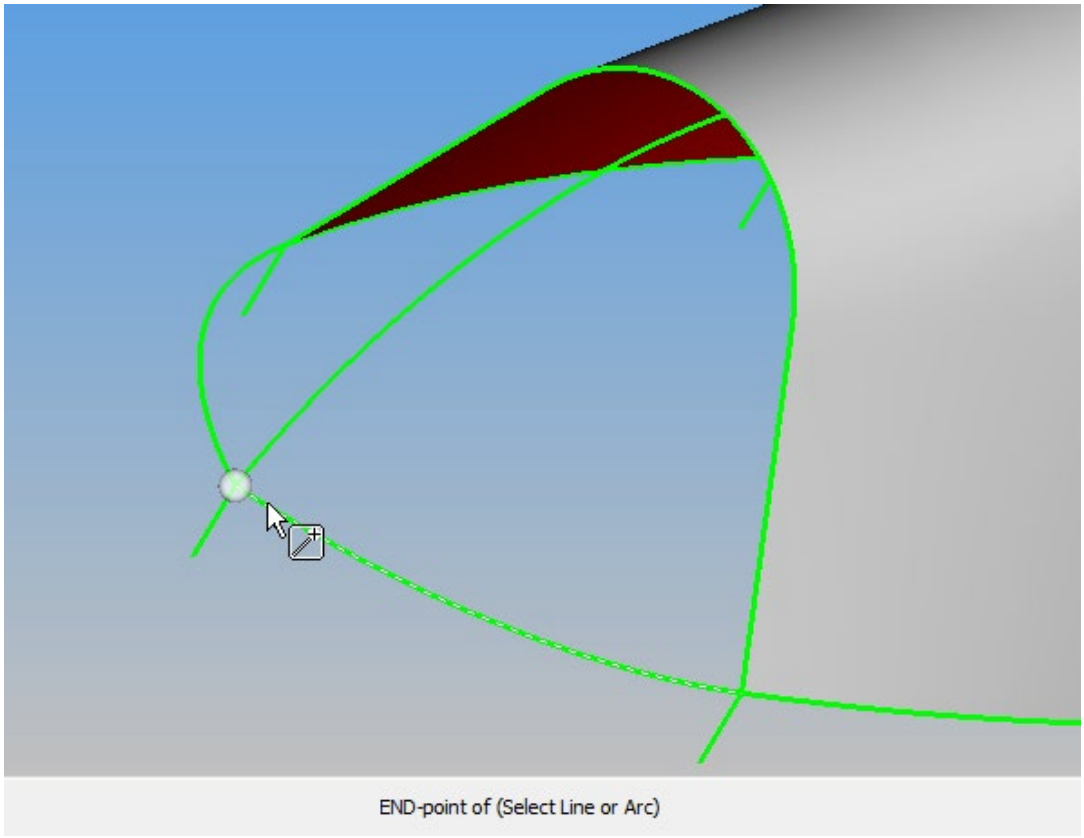


Figure 149 - Breaking the 37mm arc end point

Surface between section C-C and the lower arcs

Select **GEOMETRY > 3D Surfaces** and the surface creation dialogue is displayed.

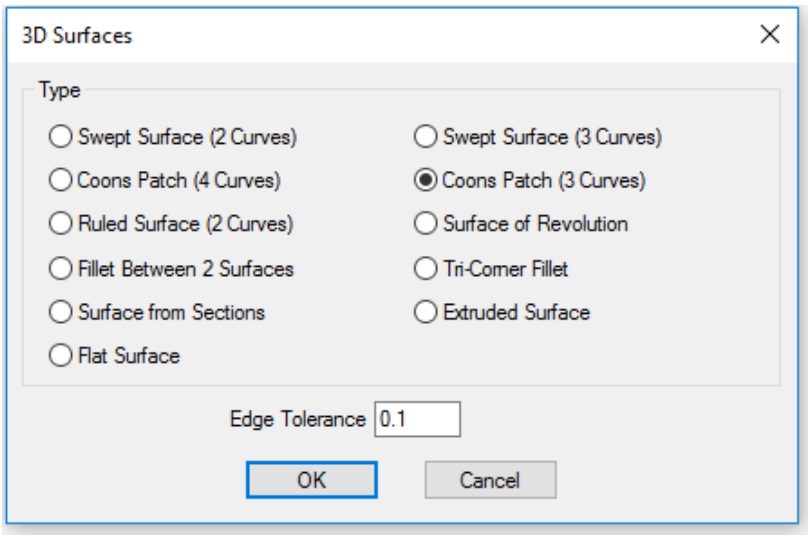


Figure 150 - Surface creation options

Select **Coons Patch (3 curves)** then **<LClick> [OK]**.
When prompted, select cross section **C-C**, then one half of cross section **B-B** and finally the 37mm base radius.

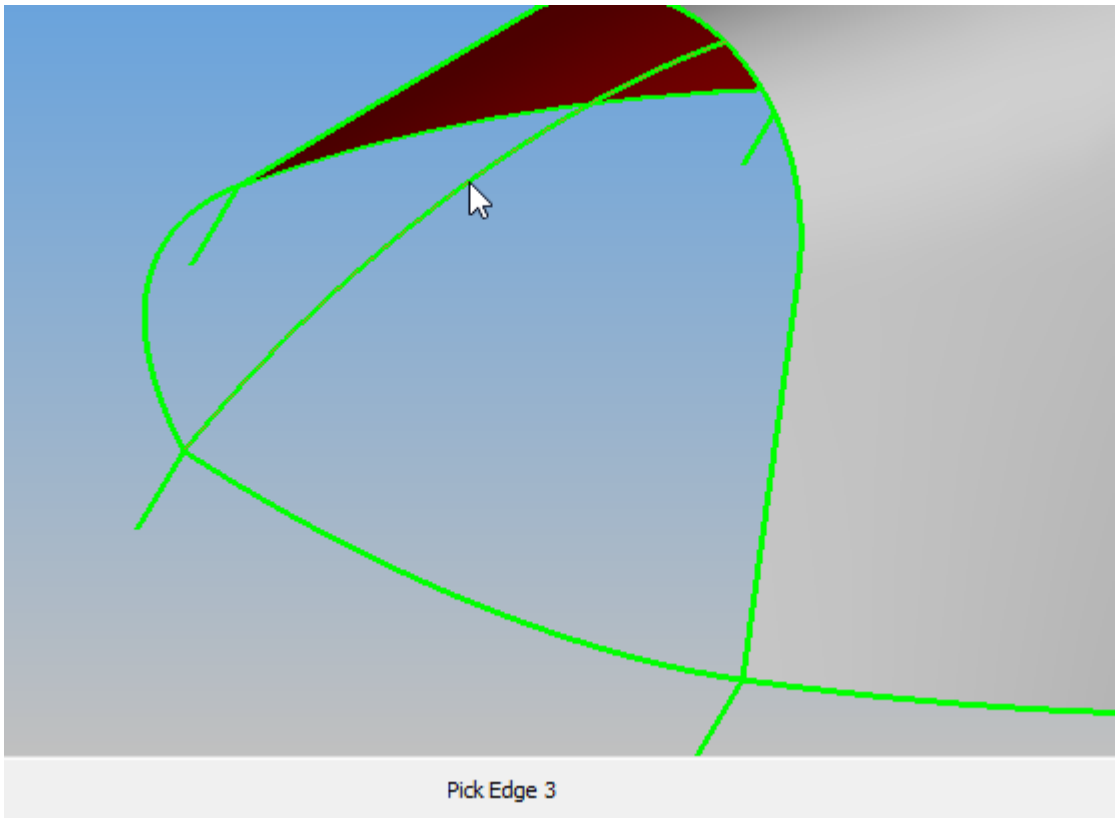


Figure 151 - Selecting the edges for the surface

Once you have chosen the three edges, the surface precision option is displayed.
The default options will always create a suitable surface, but is you need more accuracy for machining, then having more control lines is beneficial.

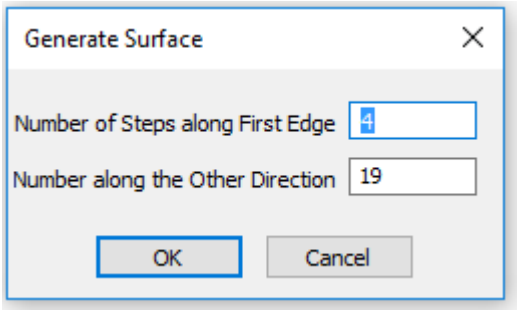


Figure 152 - Surface quality options

<LClick> [OK] to complete the process.

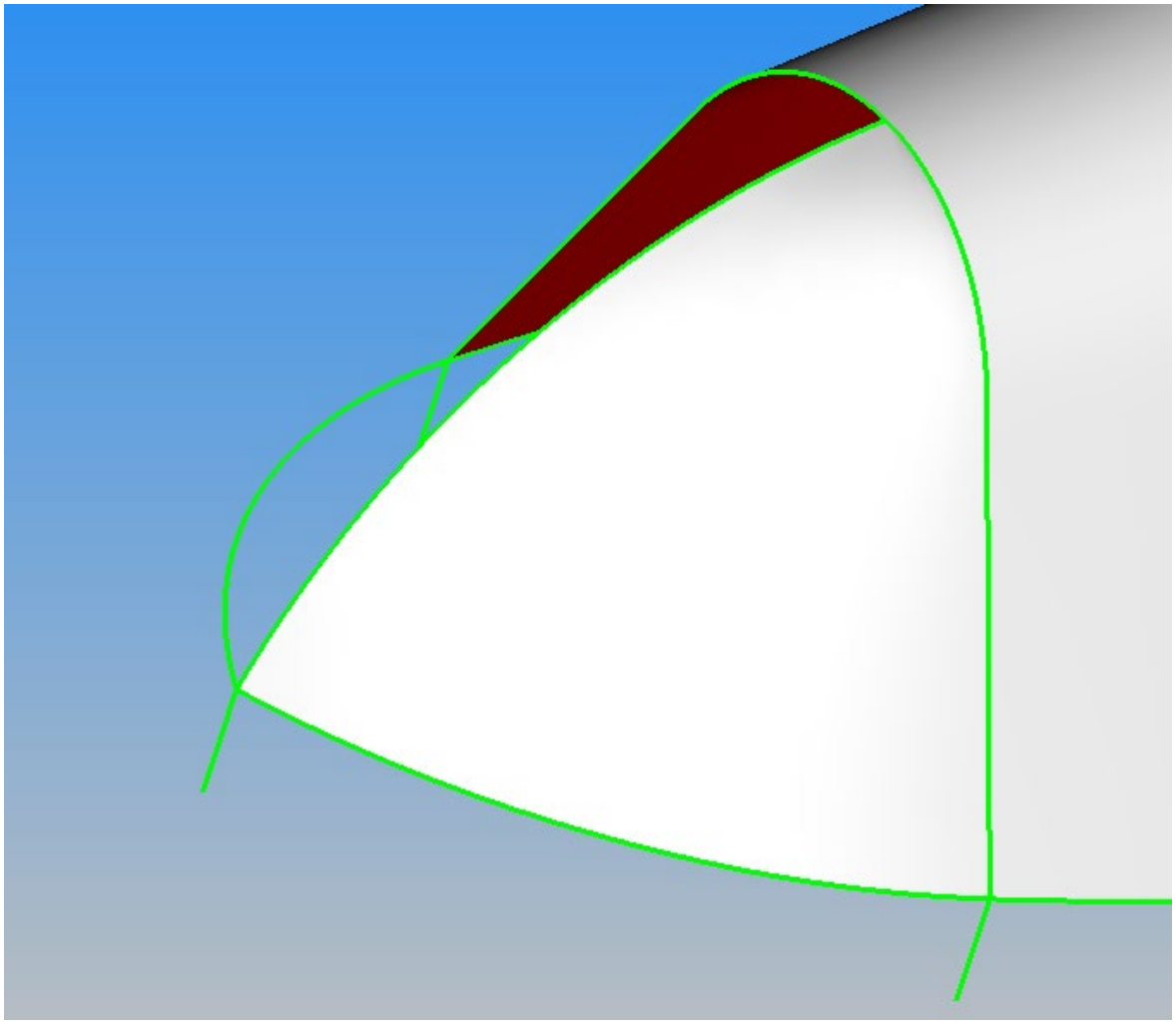


Figure 153 - Completed second surface

Repeat the process on the opposite side to create the complete end form.

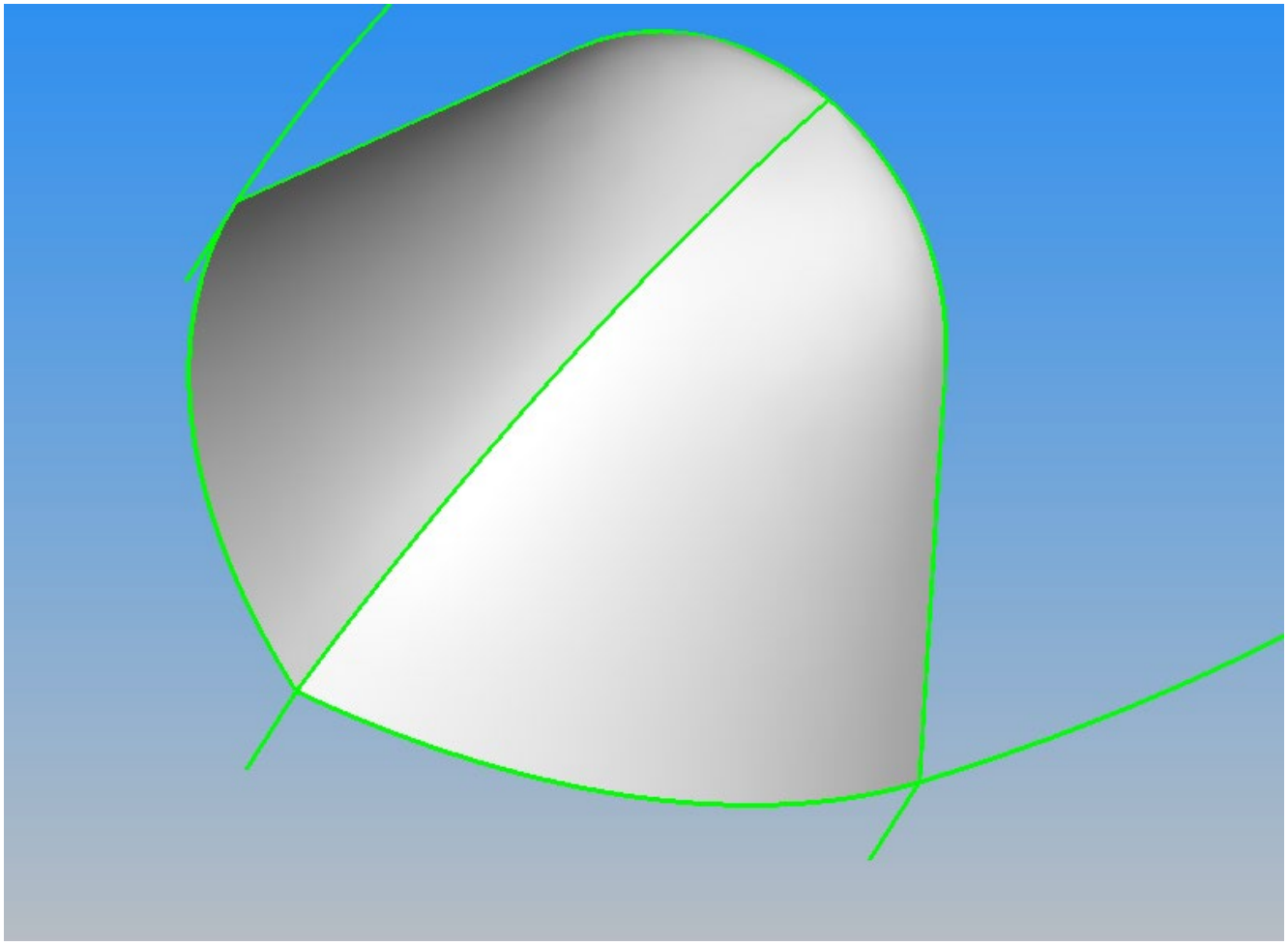


Figure 154 - Complete end form

Note that in some instances the order in which the geometries are chosen can have an impact on the finish of the surface by causing “ripples”.

If this does happen, Undo and select the elements in a differing order until you achieve the required finished form.

Creating the opposite half using Mirror

To generate the opposite surface sections, we can use **EDIT > Mirror**  .
<LClick> all three surfaces to select them.

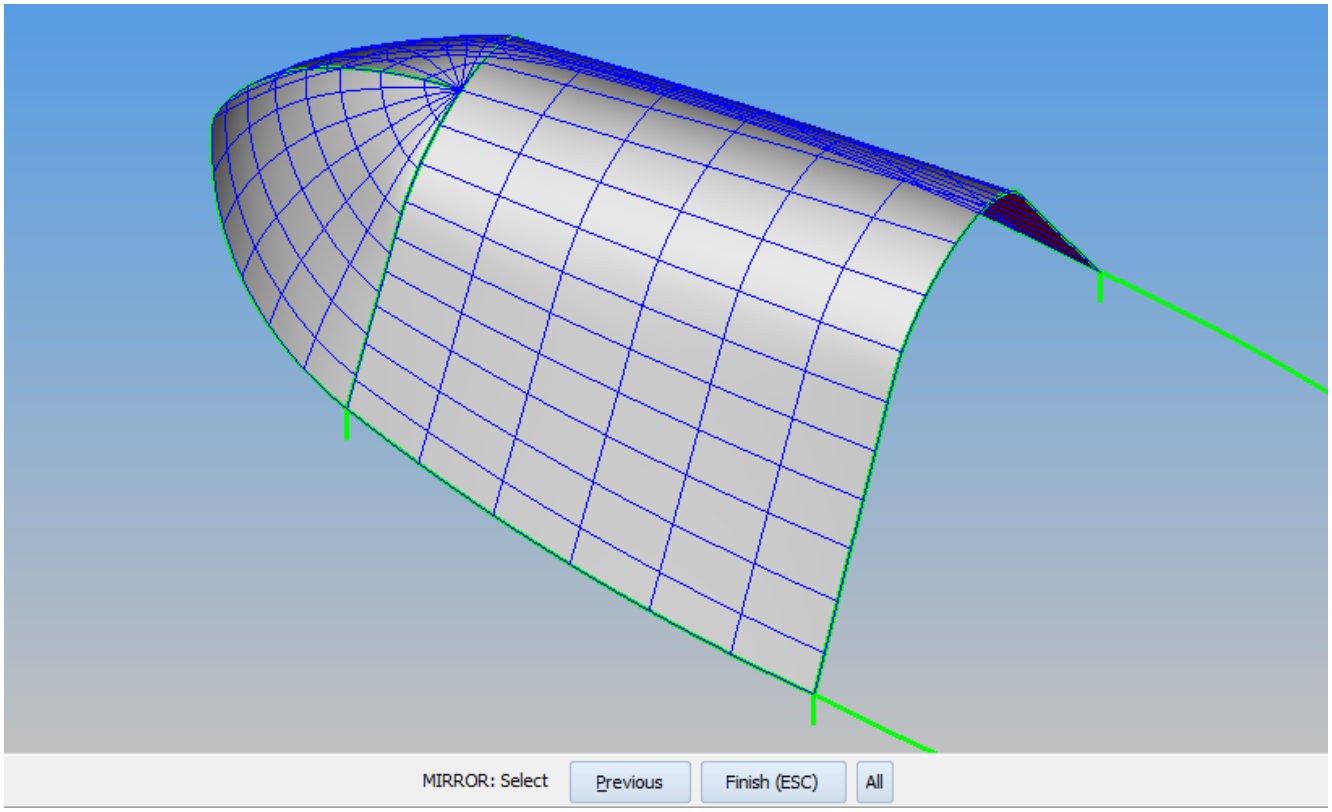


Figure 155 - Surfaces to mirror

<RClick> or **[Finish (ESC)]** to continue.
You are now asked for the reference position for the mirror. Unlike a 2D mirror command where you select two points in the same work plane, with these surfaces being in 3D space we have to select a work plane as the mirror reference.

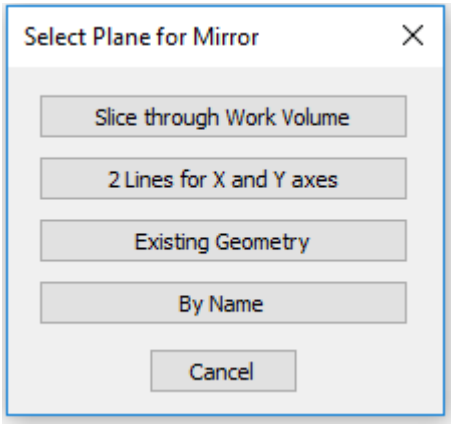


Figure 156 - Mirror plane reference selector

The easiest method on this example is to use the **[By Name]** option.

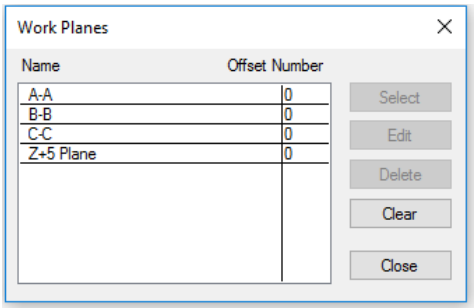


Figure 157 - Mirror reference By Name

Having unique names for our Work Planes makes the choice simple, **<LClick>** the **A-A** option, then **<LClick>** **[Select].**

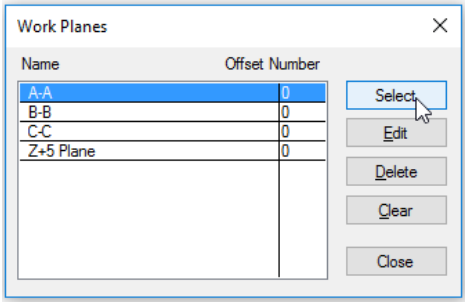


Figure 158 - Using a specific work plane

Answer **[Yes]** to keep the original surfaces. **<RClick>** to complete.

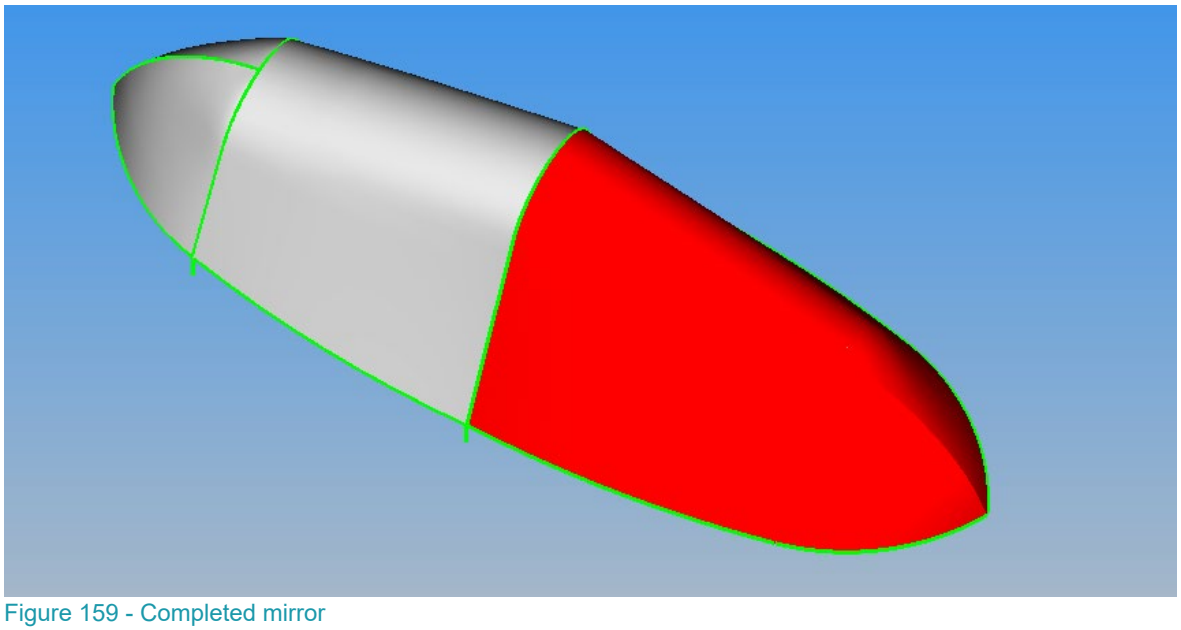


Figure 159 - Completed mirror

When surfaces are mirrored, they will be turned inside out and use of the

3D > Reverse Tool Side  option will be required to set the correct side to face out.
There is nothing wrong in using this method if you prefer, but you may have more work on a complex part.

Creating the opposite half using Rotate

Another method would be to use the **EDIT > Rotate** command.
To rotate the surfaces in 3D it is necessary to draw an axis about which the rotation will take place.

Using **GEOMETRY > 3D Polyline** , create the axis of rotation polyline from X**100, Y75, Z0** to X**100, Y75, Z70**.

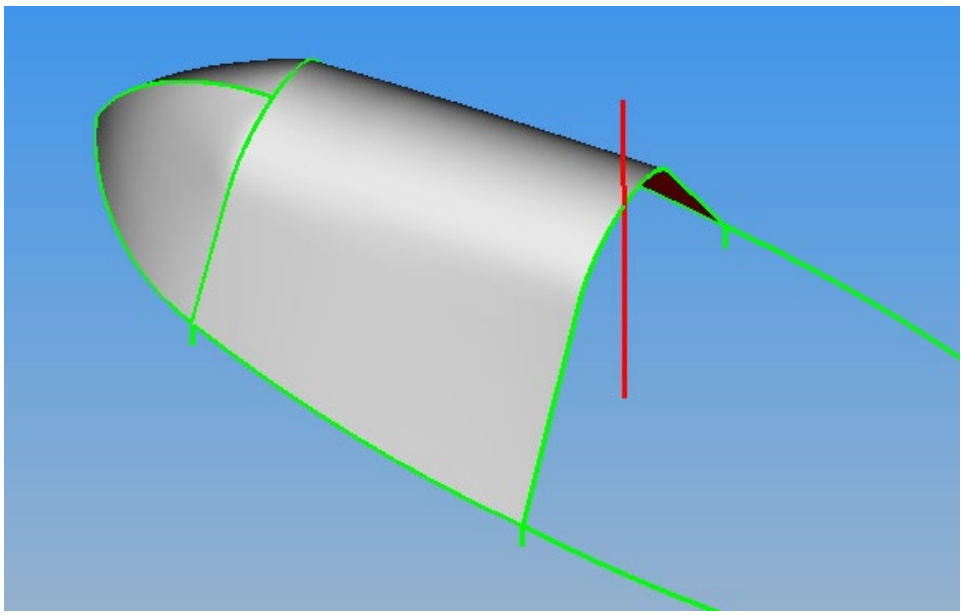


Figure 160 - Polyline axis for rotation

Select **EDIT > Rotate**
For the items to rotate, **<LClick>** the three surfaces.

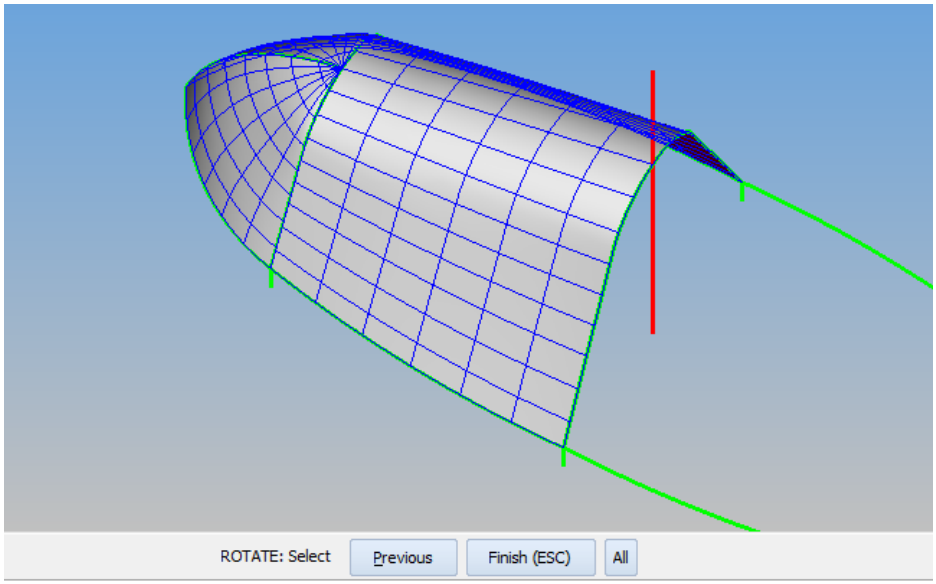


Figure 161 - Items to rotate

<RClick> or **[Finish (ESC)]** to continue.

For the Axis of Revolution, **<LClick>** the polyline.

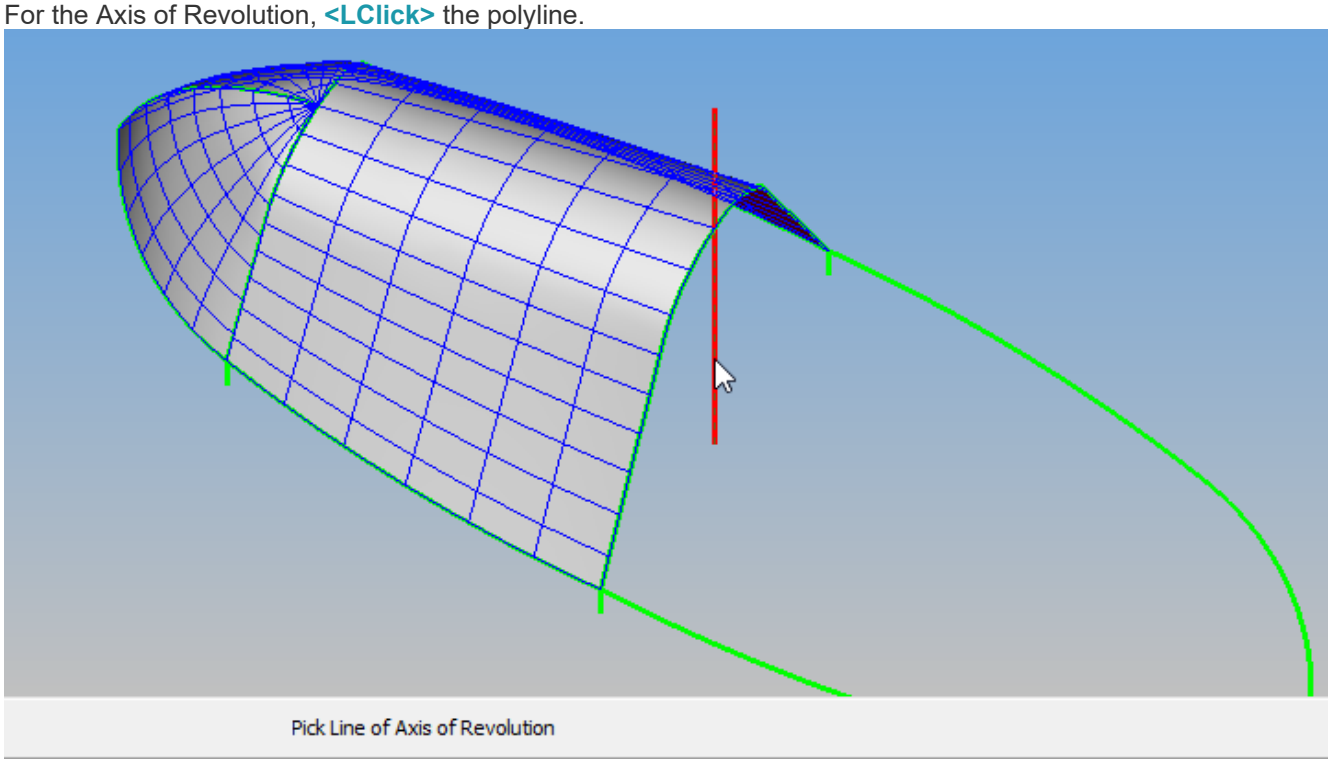


Figure 162 - Selecting the Axis of Revolution

For the angle option, enter 180.

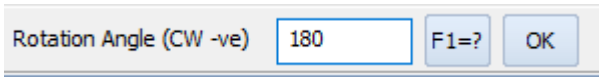


Figure 163 - Rotation angle options

<LClick> [OK].

For the number of copies, enter 2.

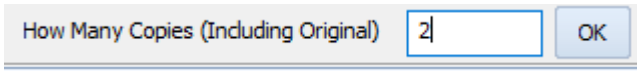


Figure 164 - How many to create

<LClick> [OK], **<RClick>** to finish.

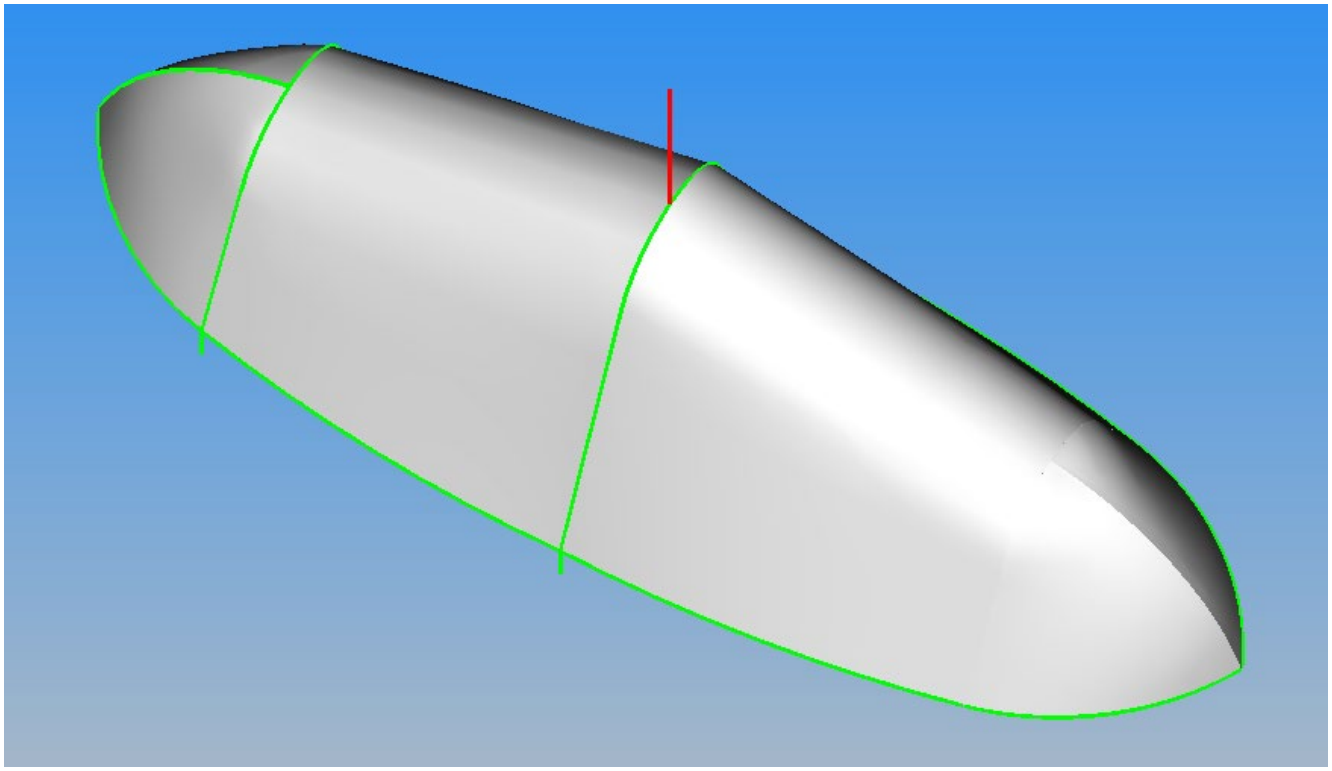


Figure 165 - Completed Rotate option

Using the Rotate option removes the need to reverse any surface sides to create a correct looking model.

Both methods are viable and will work in most scenarios, it all depends on how complex the part is, as to which is the more suitable to use.

Creating the lower vertical side surfaces

This next process has many different methods that could be used to create the side surfaces, Ruled, Extruded, Patch, Swept, it all depends on the requirements and complexity of your actual parts as to which method is the best for the part.
The final part of the tutorial will just show you how to create a single Ruled Surface between two pieces of geometry on the upper and lower base profiles. How you create the entire run of surfaces is up to you.

Ensure that all the geometries you wish to work with are visible.

Using **CAD > Show All**  is the easiest method.

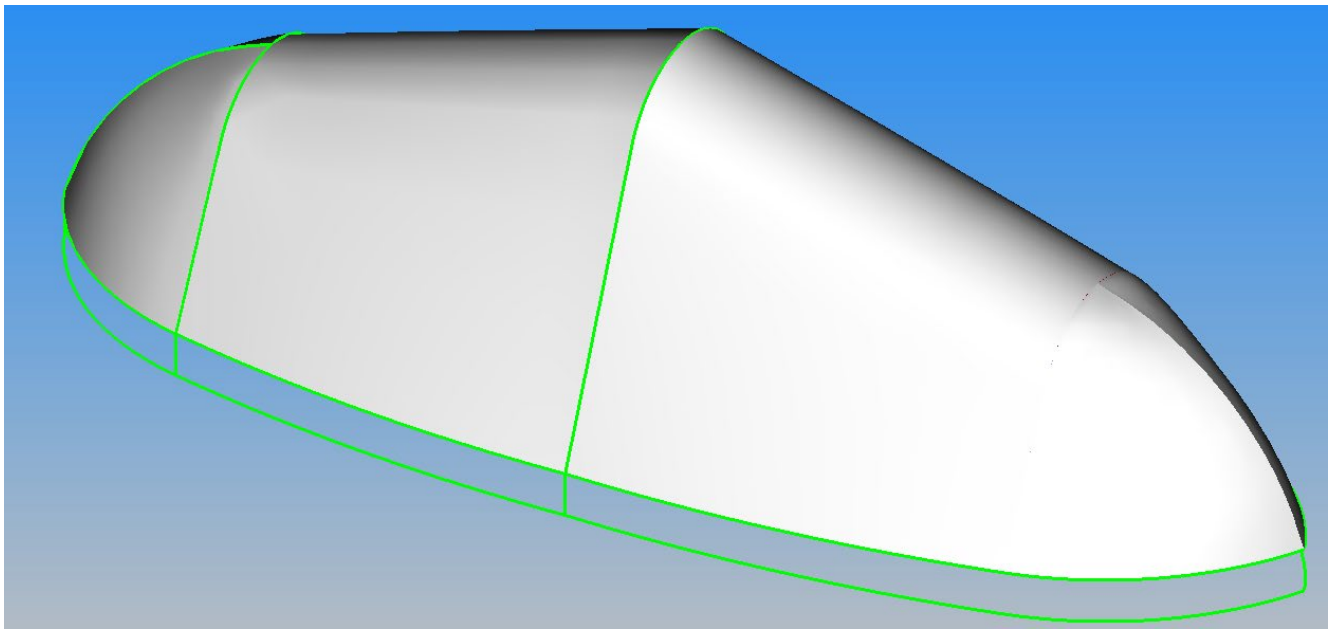


Figure 166 - Geometries for vertical surfaces

Select **GEOMETRY > 3D Surfaces** and the surface creation dialogue is displayed.

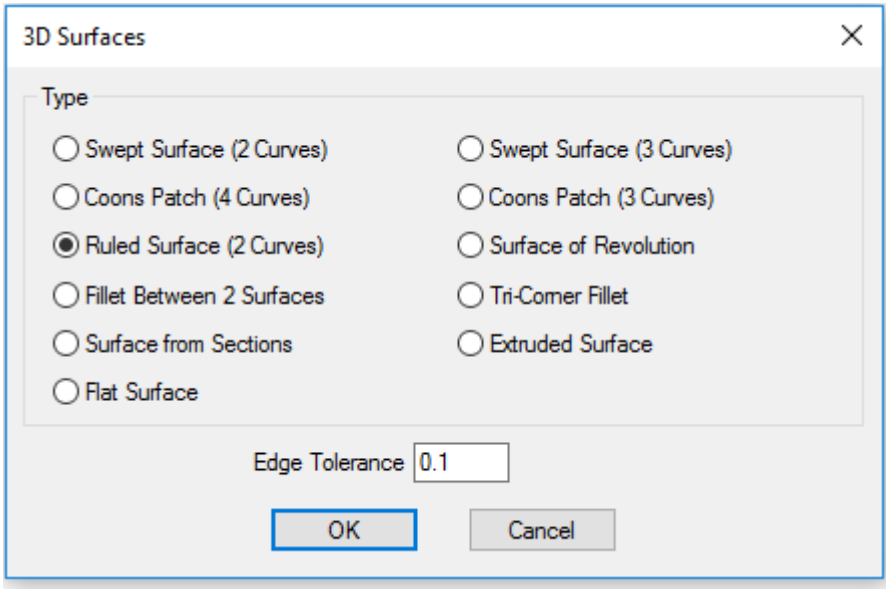


Figure 167 - Surface creation options

Select **Ruled Surface (2 curves)** then **<LClick> [OK]**.
When prompted, select two lines as shown below.

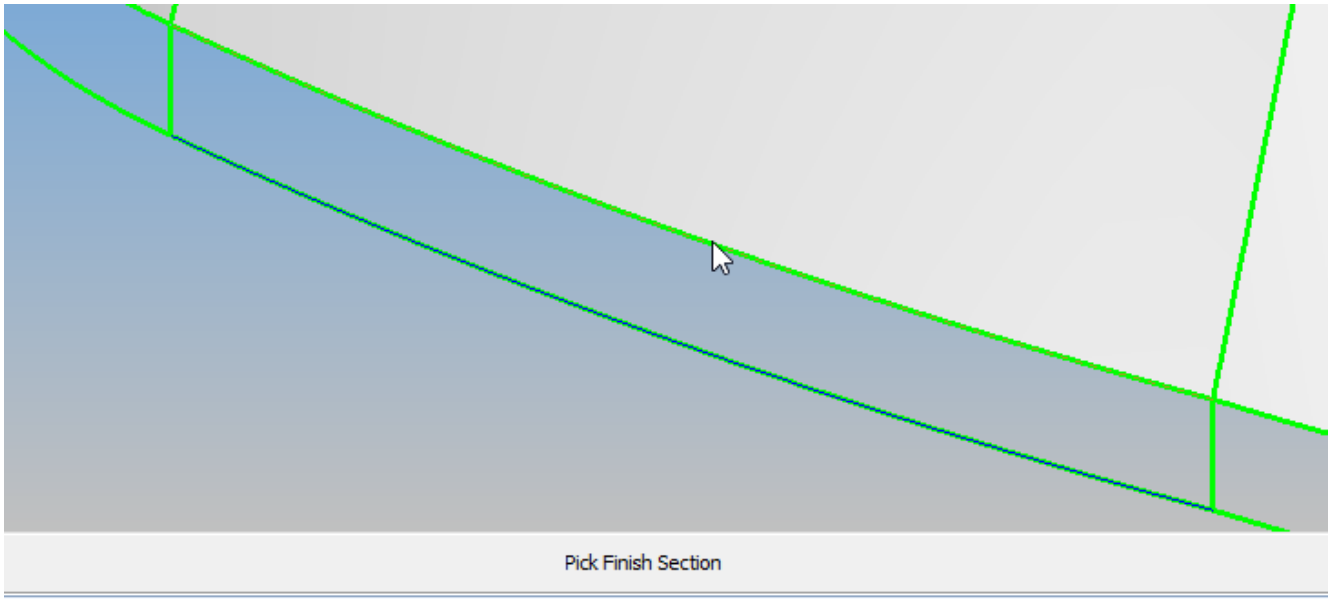


Figure 168 - Geometries for vertical surfaces

Once you have chosen the, the surface precision option is displayed.
The default options will always create a suitable surface, but is you need more accuracy for machining, then having more control lines is beneficial.

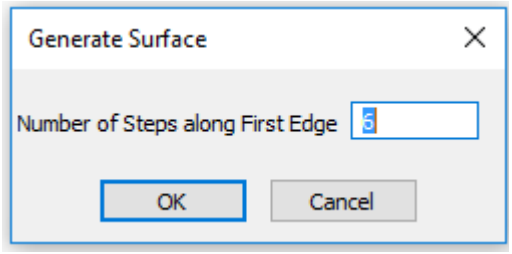


Figure 169 - Surface quality options

<LClick> [OK] to complete the process.

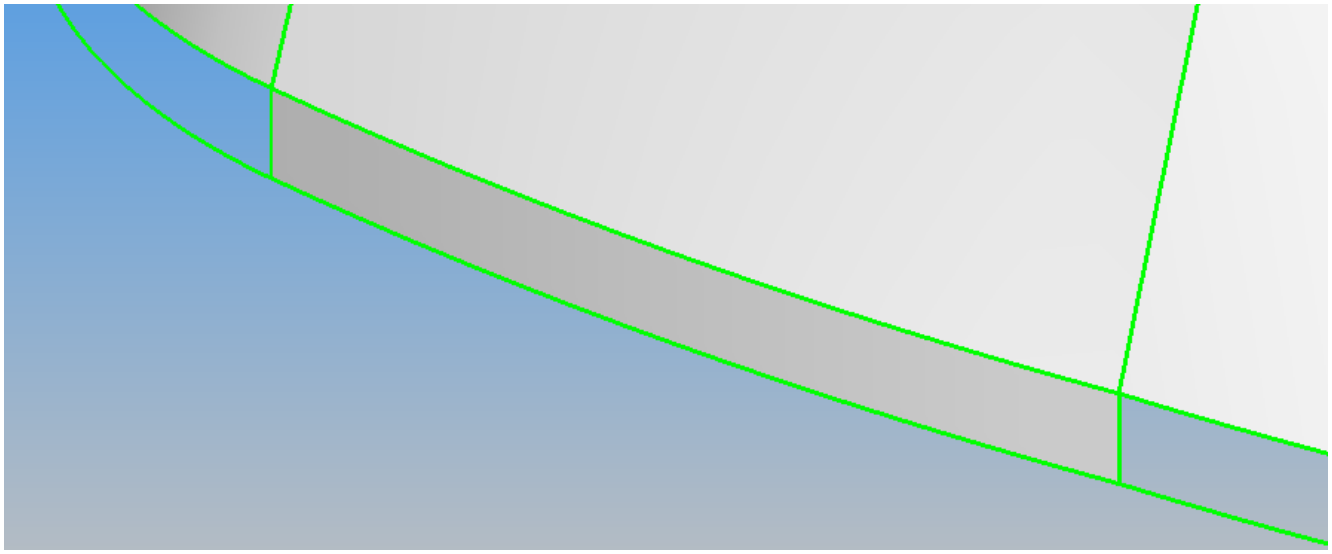


Figure 170 - Complete vertical surface

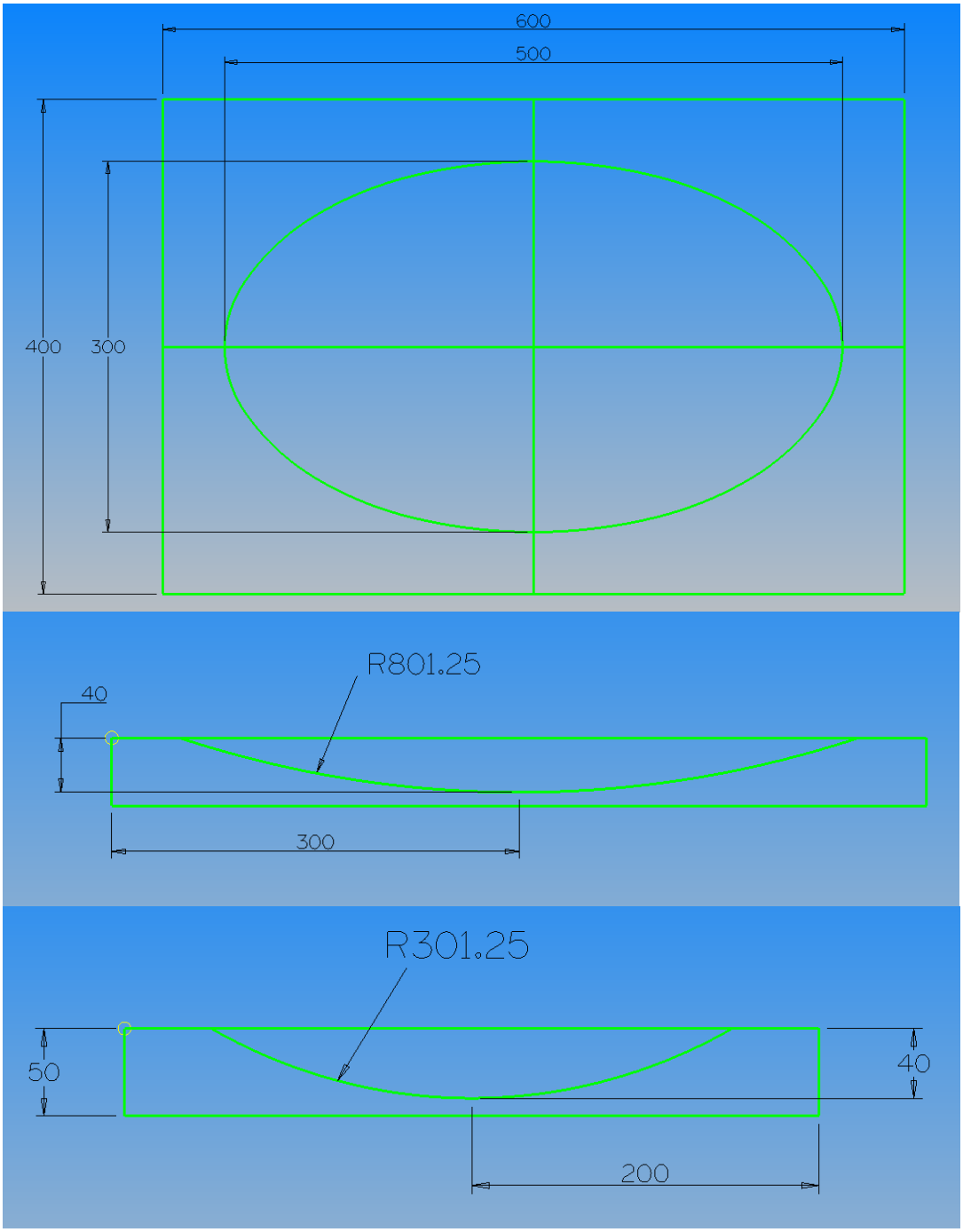
Complete the remaining sections of the geometry, remember that you may have to adjust your geometry or method depending on the situation.

You may also need to use **3D > Reverse Tool Side**  **to set the correct outside faces.**

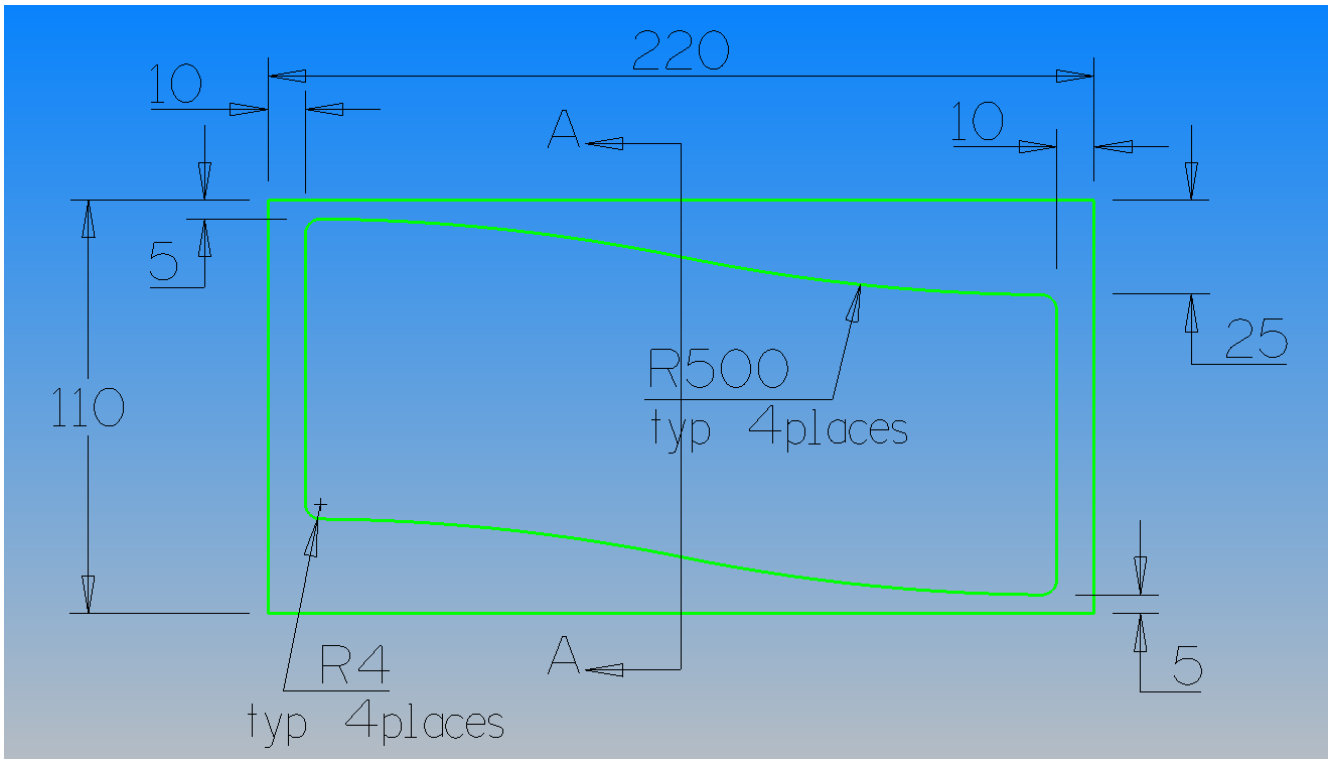
Save your completed part with a suitable name.

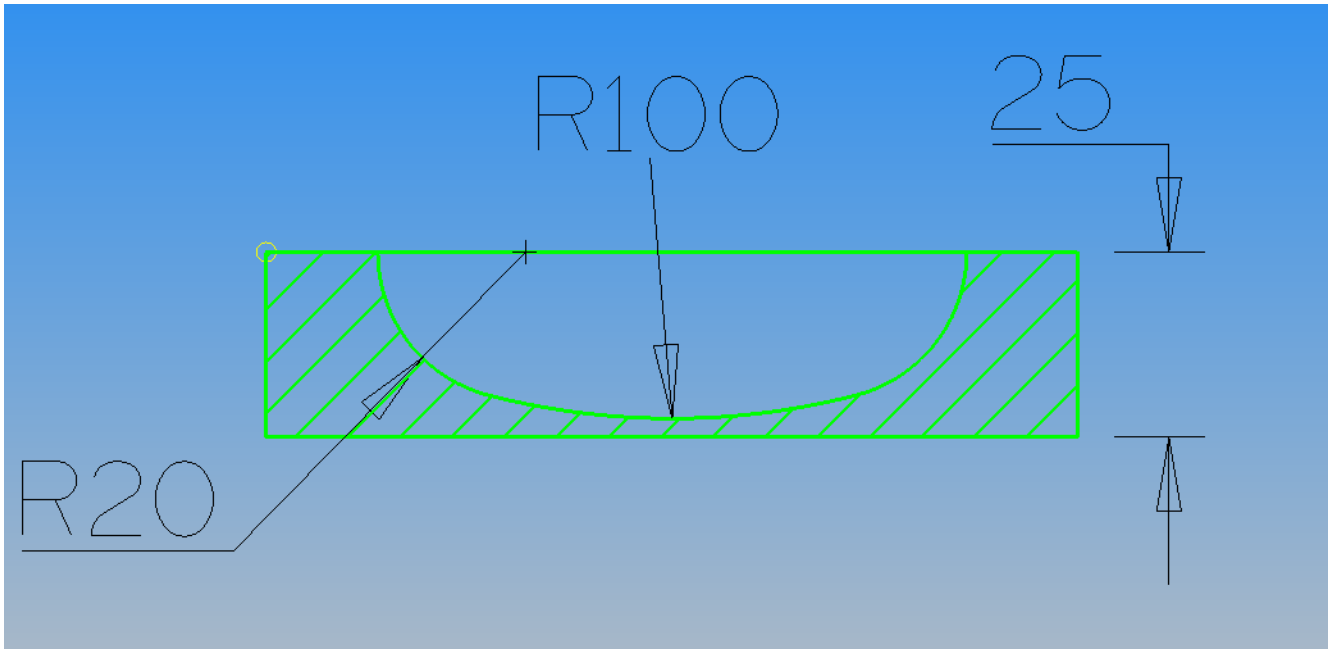
Examples

Elliptical Dish



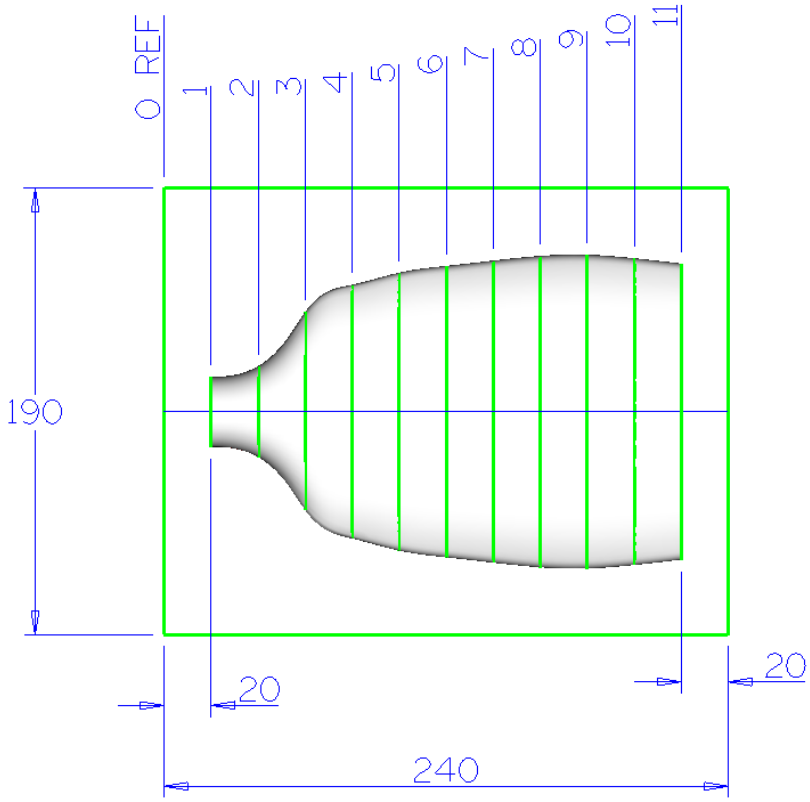
Curved Dish


220
10
10
A
5
25
110
R500
typ 4places
R4
typ 4places
A
5

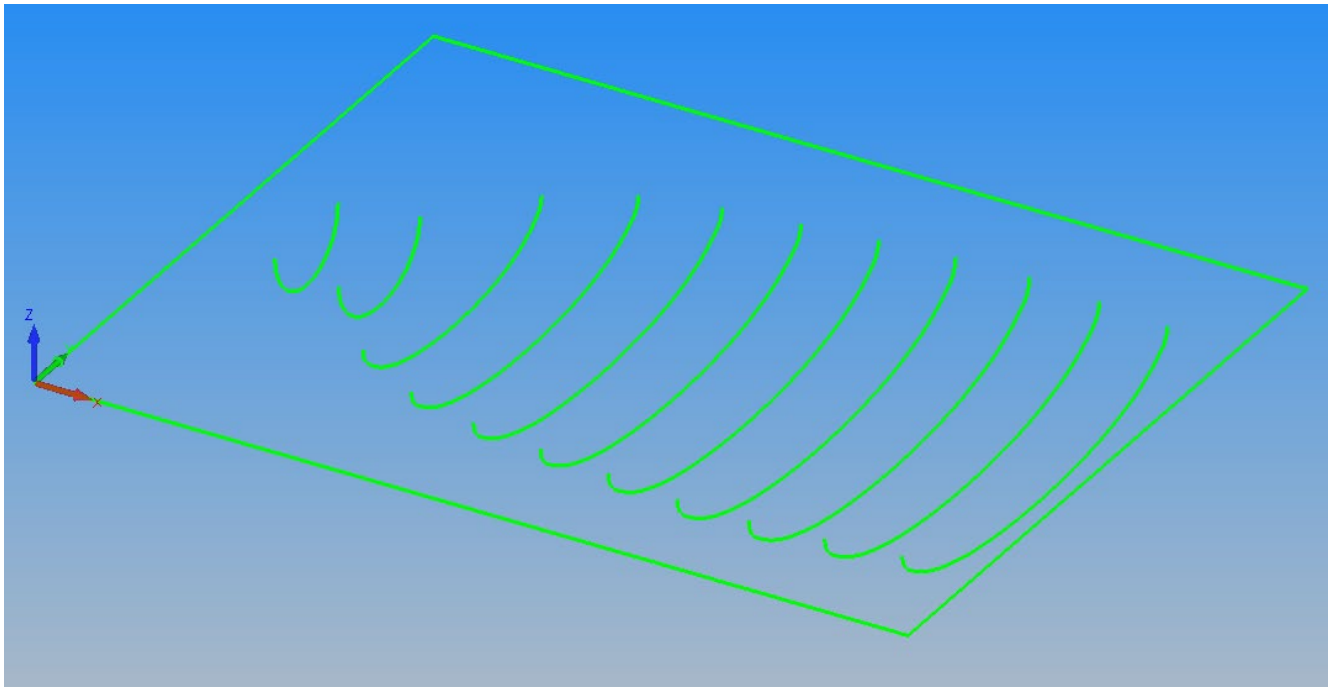

R100
25
R20

Surface from Sections

Sections at 20mm Centres



1 15 Radius Circle
2 38.7x30 ellipse
3 85x30 ellipse
4 108x32 ellipse
5 118.3x34 ellipse
6 124x36 ellipse
7 128.5x38 ellipse
8 132.5x40 ellipse
9 133.5x40 ellipse
10 130.5x38 ellipse
11 126x35 ellipse



Version amendments

V	Amendment Description	A	Software Version	Amended Date
12	Minor test updates	1	2020.1	10/10/2019
12	Template altered to Hexagon branding	0	2020.0	15/03/2019

HEXAGON
MANUFACTURING INTELLIGENCE

Alphacam 2020.1
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