HEXAGON
MANUFACTURING INTELLIGENCE

ALPHACAM 2020.1
HOUSING PLATE

Copyright

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.

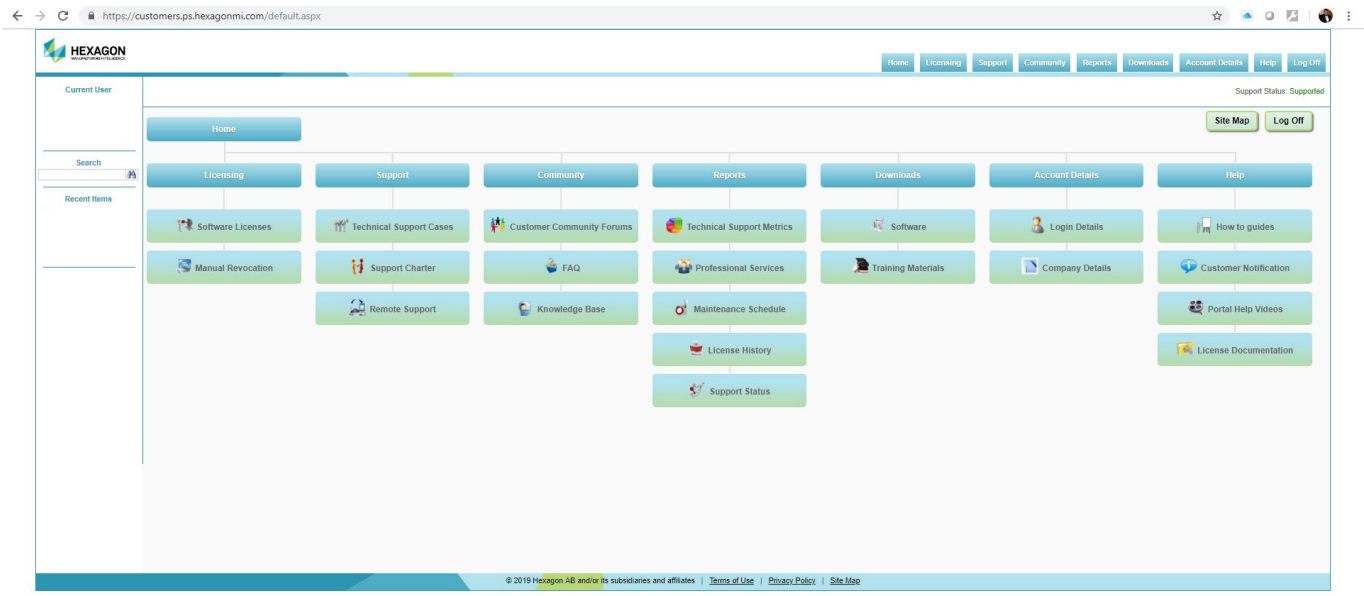


Figure 1 - Customer Support Portal

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.

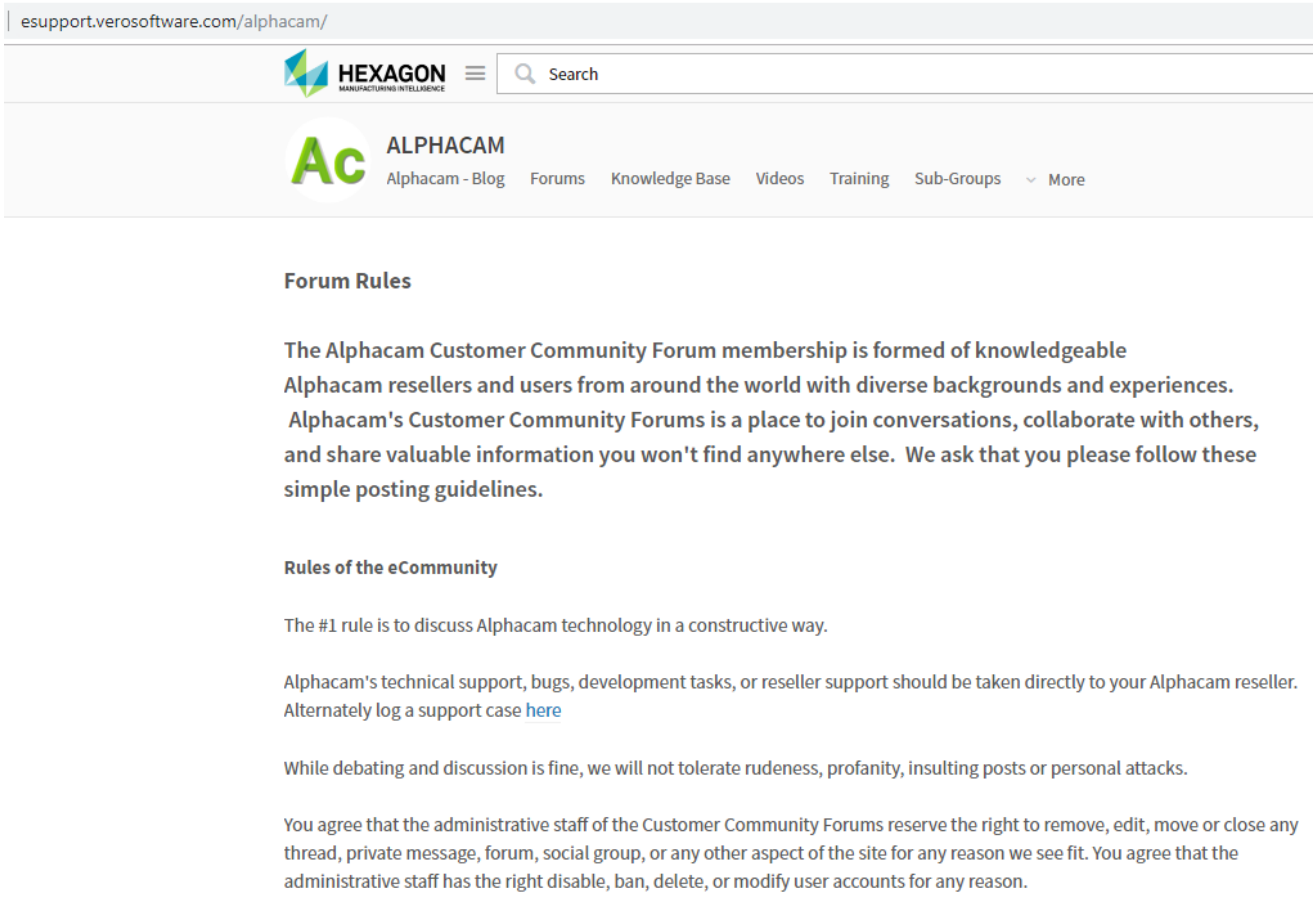


Figure 2 - esupport page

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.

This drawing is the part you are going to create.

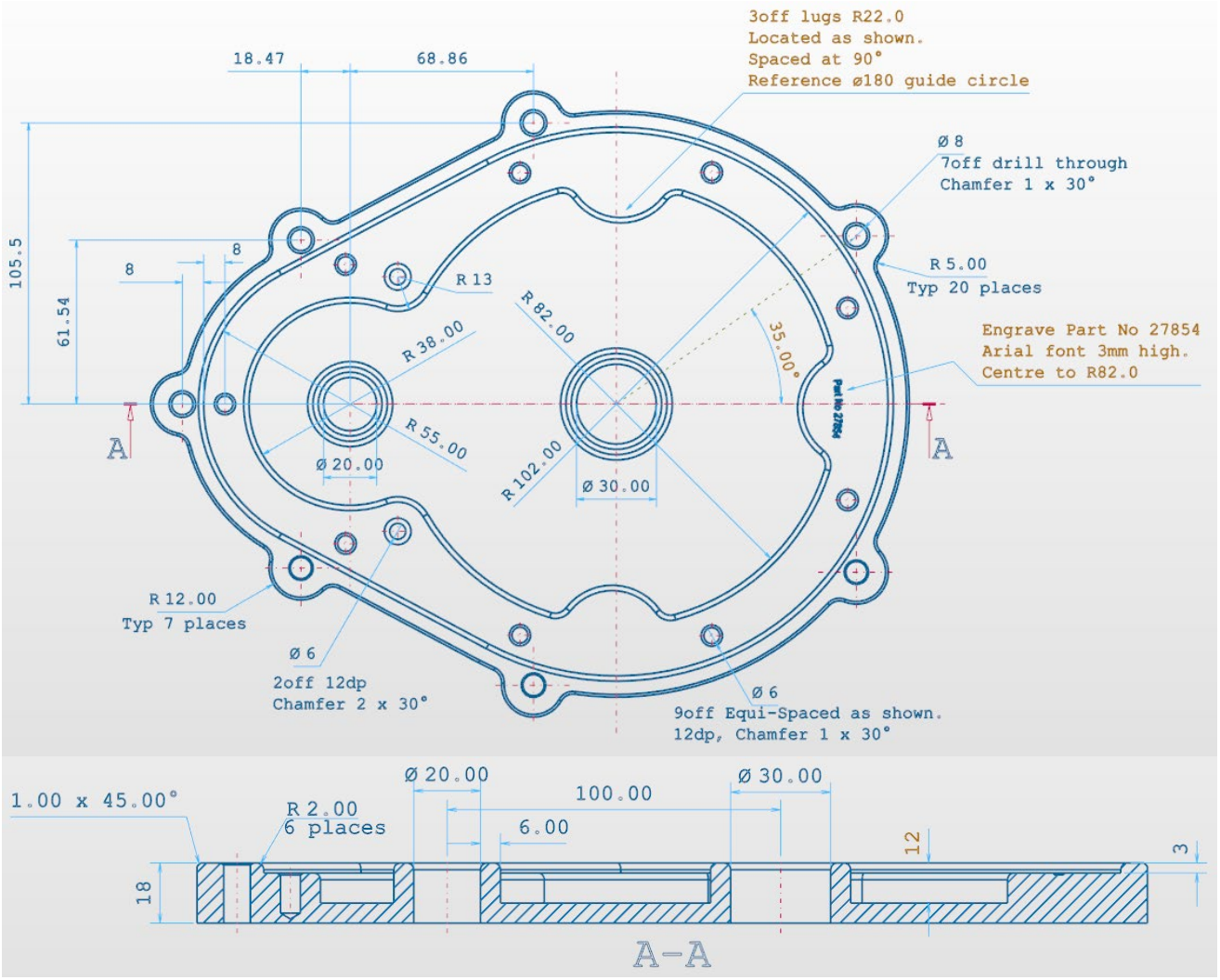


Figure 3 - Housing Plate Main Drawing Details

ALPHACAM provides many ways of creating part geometry, however for the purposes of these notes we will be using only CAD style geometry creation, whereby individual geometric features are created and then trimmed to create geometric contours. Toolpaths are then applied to the geometric contours from which the NC program is produced.

CAD style geometry creation mainly utilises the Line, Arc, Circle and Rectangle commands; however, ALPHACAM also includes “Special Geometries” which automatically produces standard geometric shapes.

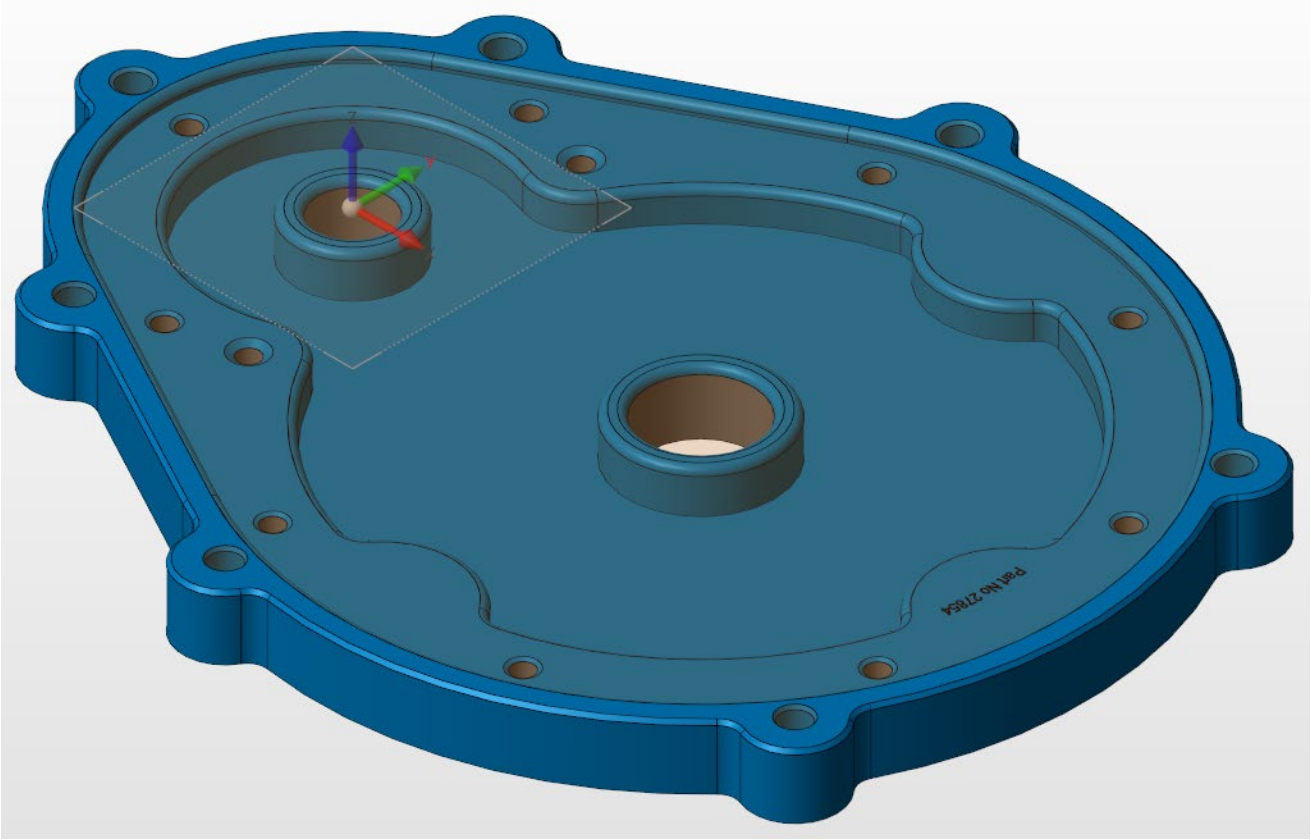


Figure 4 - Housing Plate reference model

THE PROCESS PLAN

As per the job process you should create a machining process plan prior to creating the geometric profiles. It is the process plan that determines which geometry is required to control the machining. In this example, we can assume that all the geometry is required for machining.

Prior to creating the geometry, it is good practice to produce outline notes about the geometry to be defined. In this case for clarity we will number the profiles to be drawn.

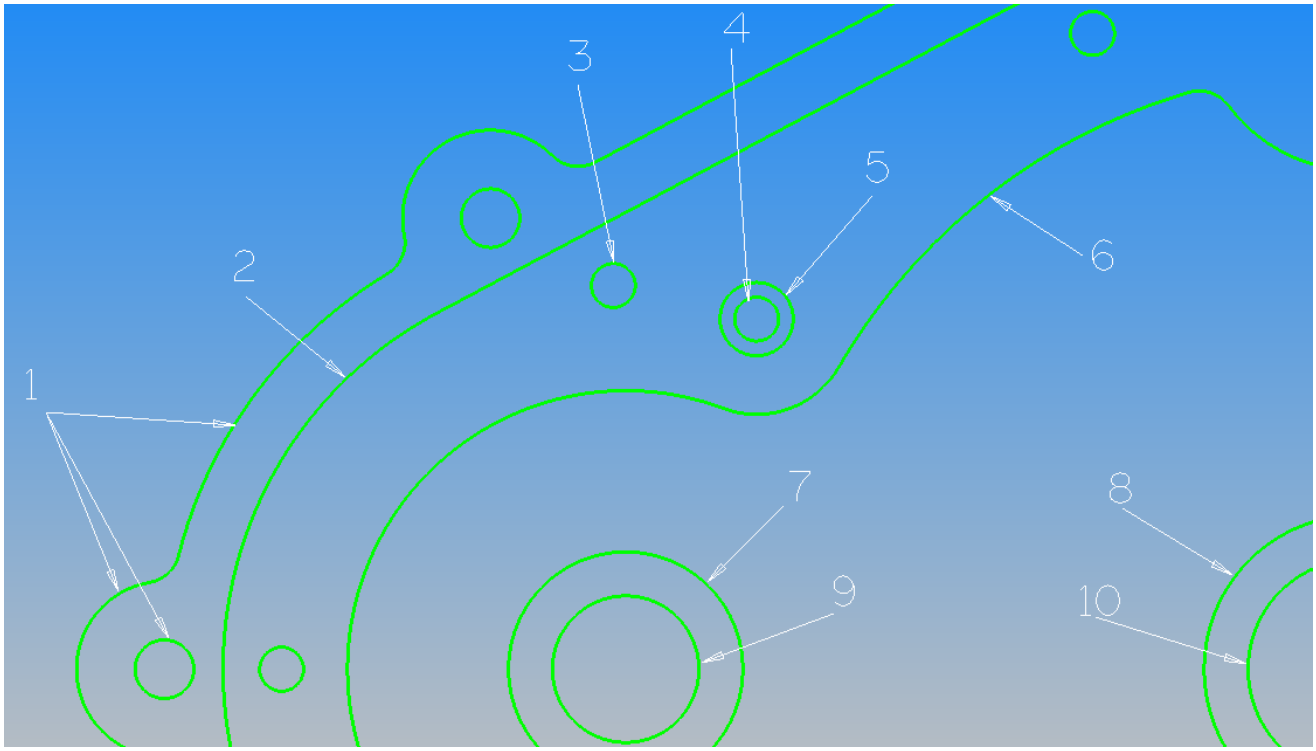


Figure 5 - Housing Plate Geometry Constituent Parts

Drawing Layer Structure

To allow for ease of manipulating the final drawing and adding machining we will create a structure of **User Layers** for the geometry to be created on.

User Layers are extremely useful as your drawings progress in aiding you visually using differing colours for different items as well as making the hiding and showing of geometries easier.

Constituent parts

1. Datum Position centre of circle profile 9.
2. Outer Profile same as profile 2 but 8mm offset larger, with Ø8 holes and R12 lugs with 5mm fillets.
3. Closed profile made up of 2 circles R55 and R102 and 2 tangent lines.
4. 9 holes Ø6, equally spaced on profile 8mm offset inside of profile 2 starting at left most quadrant point.
5. Ø10mm chamfer circle at centre of 13mm fillet.
6. Ø6mm circle at centre of 13mm fillet.
7. Figure of 8 with circles R38 & R82, with 3off R22 indents on Ø180 circle, 5mm fillets to indents plus 2off 13mm radius fillets.
8. Circle offset 6mm bigger than circle profile 9.
9. Circle offset 6mm bigger than circle profile 10.
10. Circle Ø20.
11. Circle Ø30.
12. 3mm Arial Font text “Part No 27854” centred on R82 radius guide line.

Order of Production

Draw the Profiles in the following order;
9 & 10, 7 & 8, 6, 2, 3, 1, 5, 4, 11

The order listed for drawing is based on the given information on the drawing in that you are unable to create certain geometry items until others have been drawn first.

CREATING THE GEOMETRY

Profiles 9 & 10

2 off circles Ø20 and Ø30.

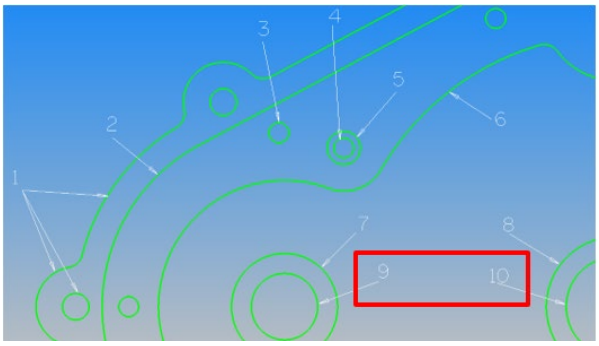


Figure 6 - Profile 9 & 10

Create and activate the User Layer

From the Layer page of the Project Manager, **<RClick>** on the entry User Layers.

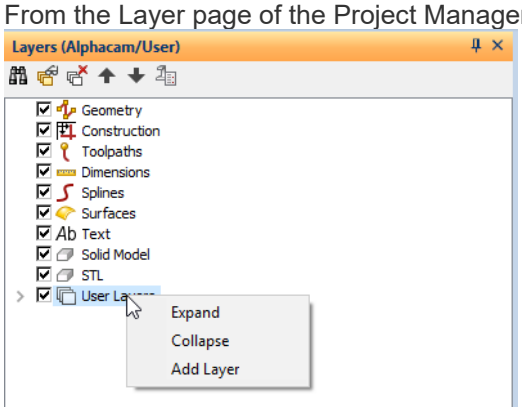


Figure 7 - Adding the first User Layer

From the context menu, select the option **Add Layer**.

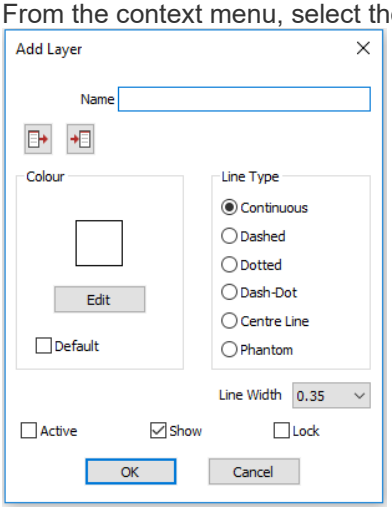


Figure 8 - Default User Layer Dialogue

In the dialogue, we will set a suitable name, chose a colour for the layer and set whether it is to be the active drawing layer we wish to use next.
For the Name, we will use **Shaft Holes.**

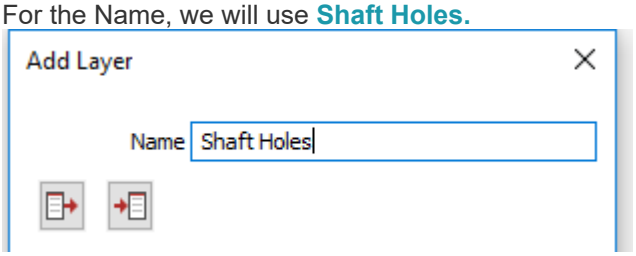


Figure 9 - Assigned name for the layer

The two buttons below the Name section allow for you to Save or Load a specific User Layer set up.
This will include the colour preference and Line Type which has previously been saved.
This can save you time when you want to generate a standard layout.

<LClick> on the **[Edit]** button to access the colour options.

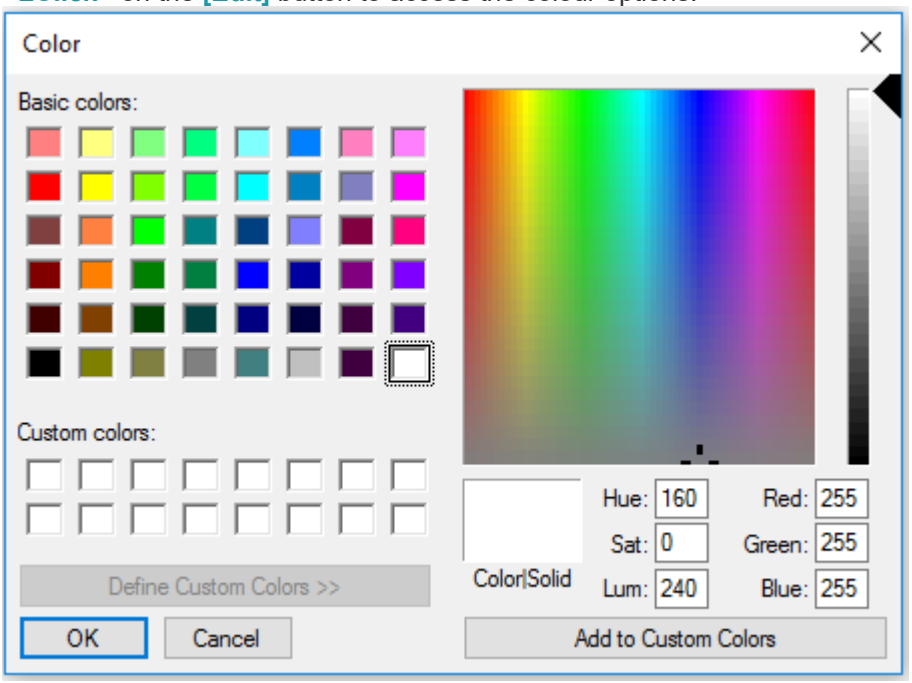


Figure 10 - Standard colour palette options.

You may select any of the predefined colours by **<LClick>** on the colour square, or you can select by clicking on the larger colour palette area.

Once you start to use this function more, you may wish to use the **Add to Custom Colours** to populate these for future use.

Select any colour you wish, with a word of caution.

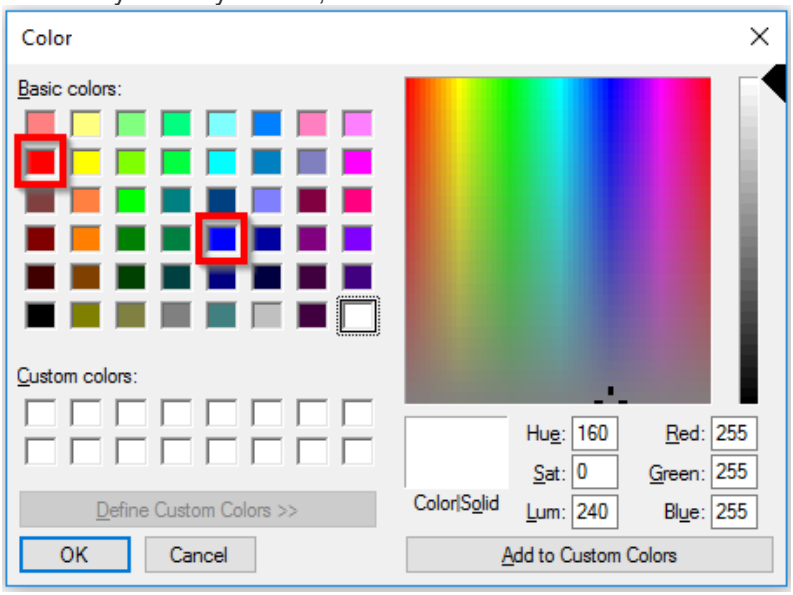


Figure 11 - Colours to avoid

> On a default install of ALPHACAM, the two colours indicated in the image above have specific functionality and you should not use them.

Once you have selected your colour, **<LClick> [OK]** to return to the User Layer set up dialogue.

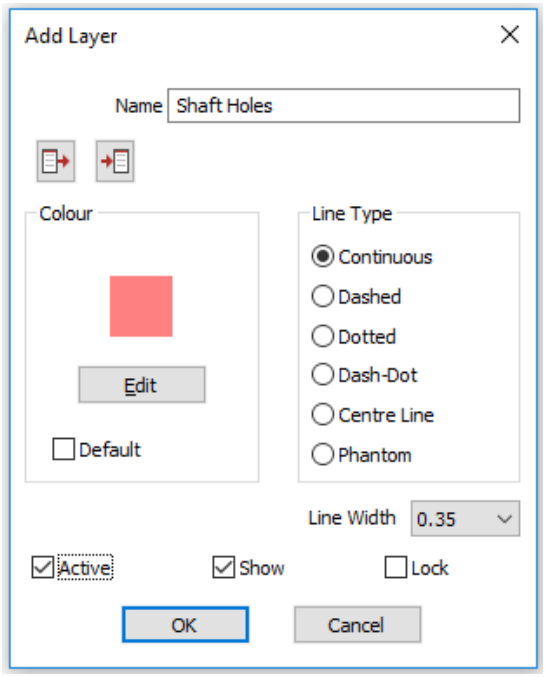


Figure 12 - Created layer dialogue set to Active

Lastly on the dialogue make the ☑ **Active** valid so that all your drawing creation will occur on this new layer. **<LClick> [OK]** to complete.

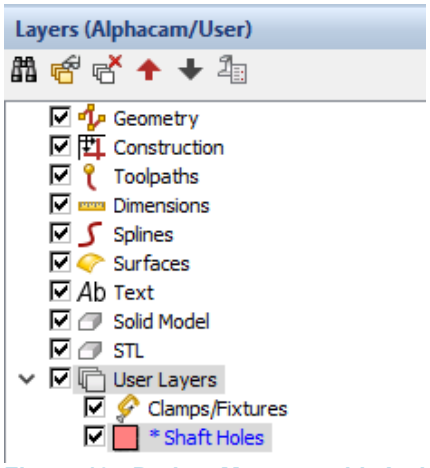


Figure 13 - Project Manager with Active User Layer

You can now see the new **User Layer** added, highlighted in blue and with a star to tell you this is your current active drawing layer.

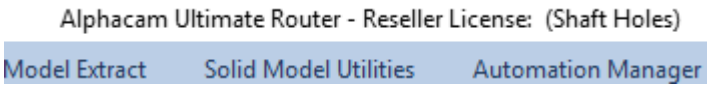


Figure 14 - ALPHACAM window Title Bar with Active Layer in brackets

> It is worth noting that the active User Layer will also be shown in brackets on the Title Bar of your ALPHACAM window, saving you the trouble of having to look at the Project Manager every time you need to check.

Create the geometry

Select **GEOMETRY > Circle > Centre + Diameter**

The command line will prompt you to enter the circle diameter. Type **20 <Enter>.**
Type **0,0** for the co-ordinates of the circle centre.
The 20mm diameter circle will be drawn.
The command line will again ask you for the circle diameter.
Type **30 <Enter>.**
Type **100,0** for the co-ordinates of the circle centre.
The 30mm diameter circle will be drawn. **<RClick>** to finish the command.

Figure 15 - First two circles on the User Layer

Profiles 7 & 8

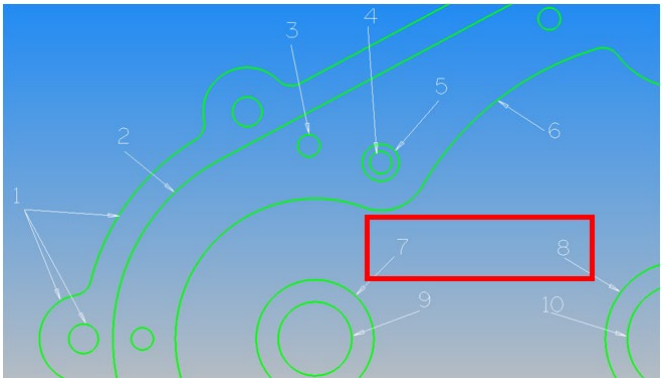


Figure 16 - Profiles 7 & 8

Create and activate the User Layer

These next two profiles are part of a different section of the drawing, so require a new User Layer.

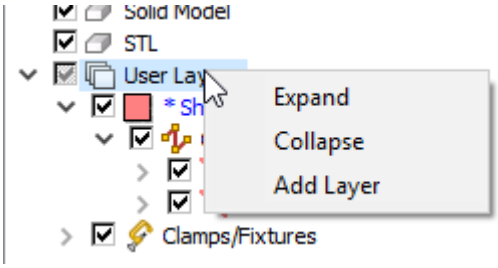


Figure 17 - <RClick> options for a new User Layer

Create the layer as shown below.

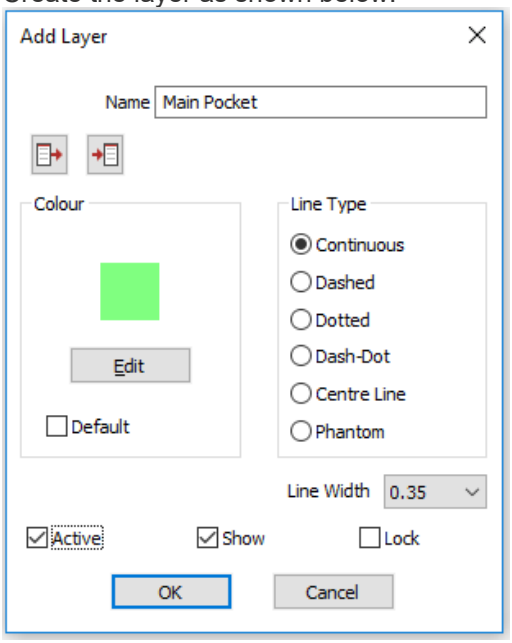


Figure 18 - User Layer Main Pocket

<LClick> [OK] to continue.

Create the geometry

2 off circles 6mm larger radially than profiles 9 & 10.
Create the walls around the two holes using the offset command.

Select **EDIT > Break Join etc. > Offset**

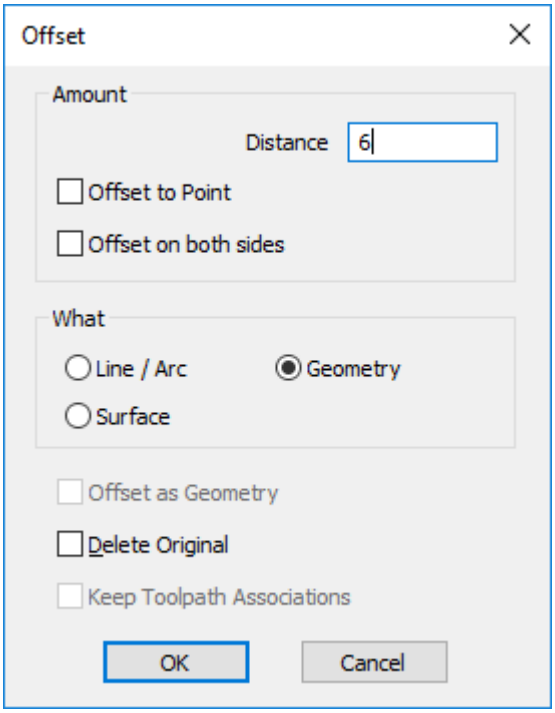


Figure 19 - Offset Dialogue

Set the options as shown then **[OK].**
<LClick> the Ø20 circle, then **<LClick>** anywhere on the outside, the new profile will be drawn.
<LClick> the Ø30 circle, then **<LClick>** anywhere on the outside, the new profile will be drawn.
<RClick> to finish the command.

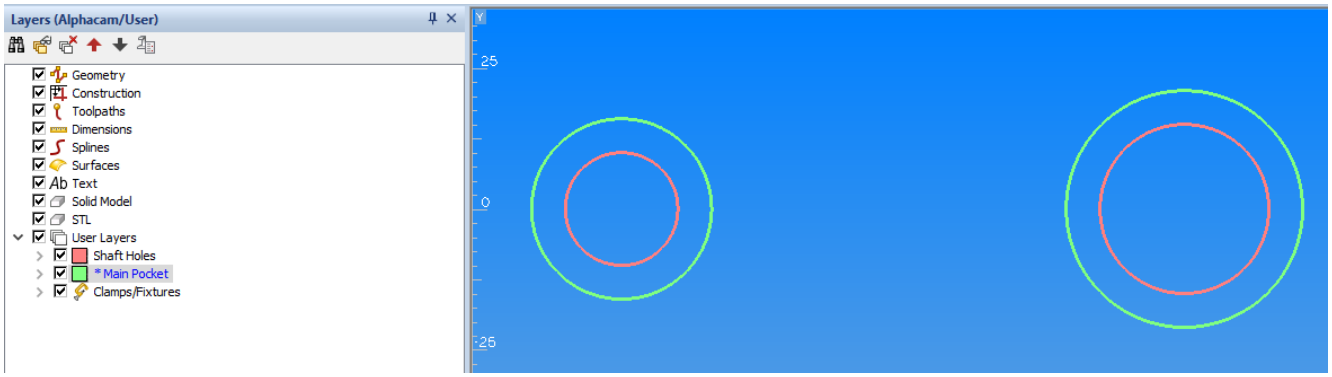


Figure 20 - Created geometry on Main Pocket layer

Worth noting is the use of the Geometry option in the What section.

Either options, Line/Arc or Geometry will create a circle, but the Line/Arc option will be set as a Construction element.

Draw profile 6

As this next section of geometry is also part of the **Main Pocke**t, we carry on using the already active User Layer.

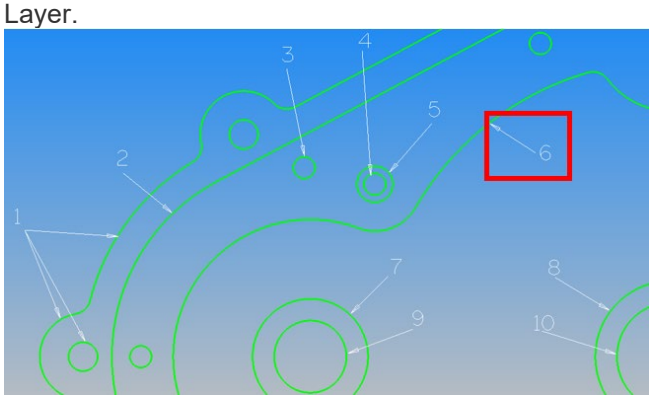


Figure 21 - Profile 6

Profile 6 is comprised of the following features;
Figure of 8 with circles R38 & R82, with 3off R22 indents on Ø180 reference circle, 5mm fillets to indents plus 2off 13mm fillets.

Draw the R38 and R82 circles

Prior to activating the next command ensure that **Auto Snap <F2>** is active.
(Is the word **Auto** highlighted at the bottom right of the screen?).

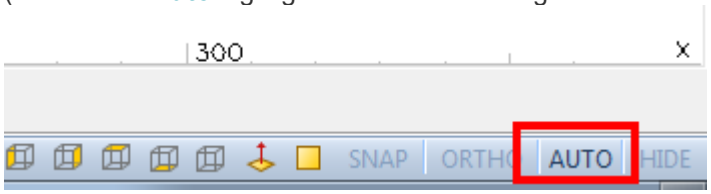


Figure 22 - Auto is shown at the bottom right of the screen

To activate Auto Snap press **<F2>.**

Select **GEOMETRY > Circle > Centre + Radius** 
The command line will prompt you to enter the circle radius. Type **38 <Enter>,** position the cursor over the centre of profile 9 (the snap selector will be displayed).

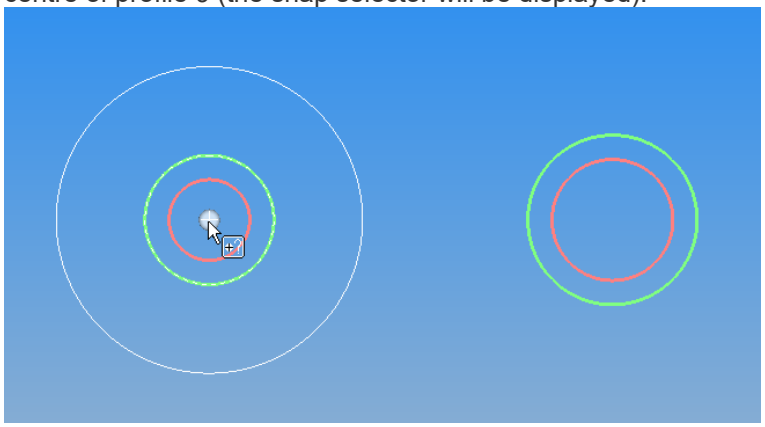


Figure 23 - Auto Snap marker to position the circle

<LClick> to select this position. The 38mm radius circle will be drawn.

The command line will again prompt you for the circle radius.

Type **82** into the dialogue, then position the cursor over the centre of profile 10 (the snap selector will be displayed) **<LClick>** to select this position.

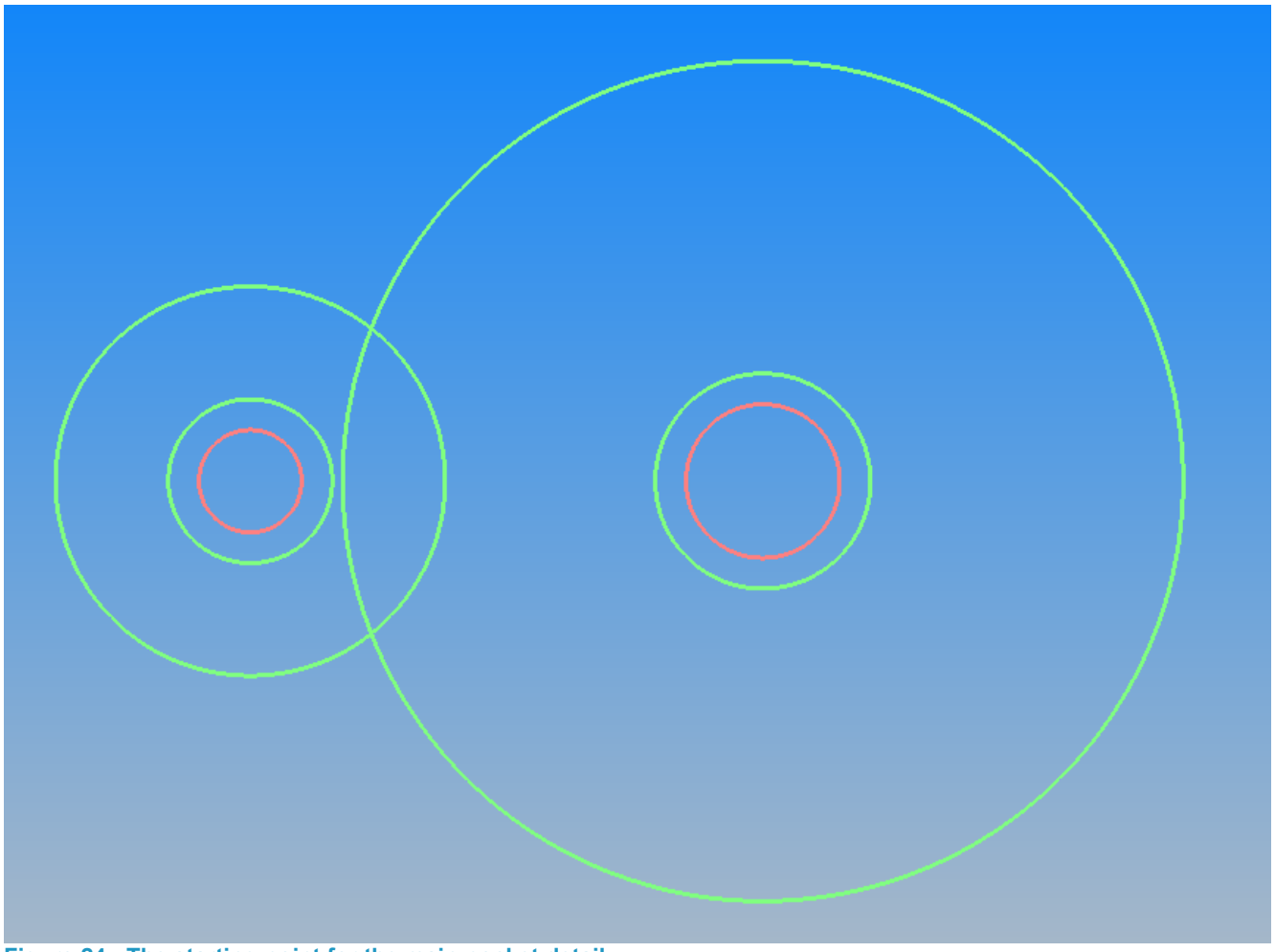


Figure 24 - The starting point for the main pocket detail

The 82mm radius circle will be drawn.
<RClick> to finish the command.

Main pocket Creation, Method 1

Using the Unite command

Select **EDIT > Fabricate > Unite** 

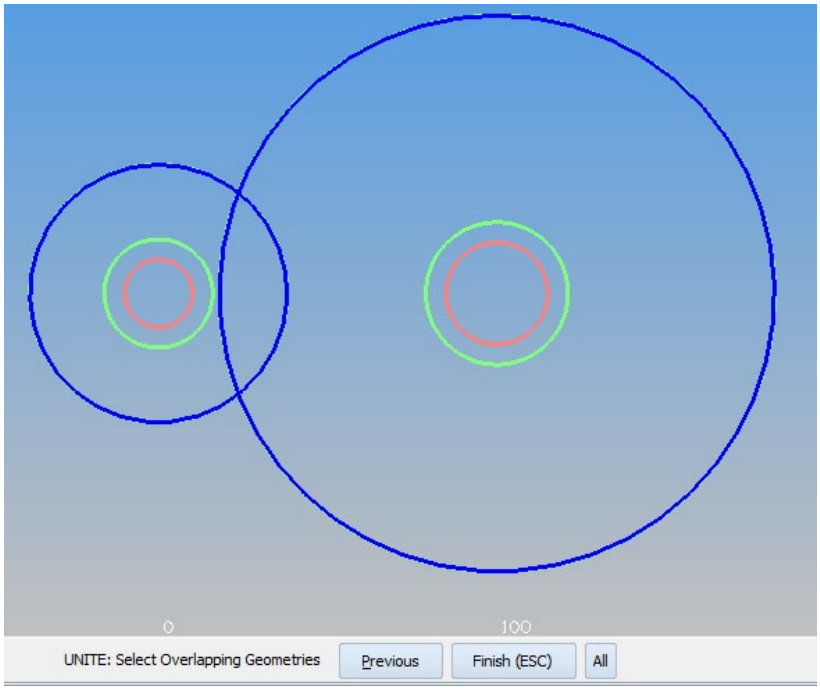


Figure 25 - Unite the two circles to complete the profile

<LClick> the two large circles to select them, then **<RClick>** to finish the process and create the new profile.

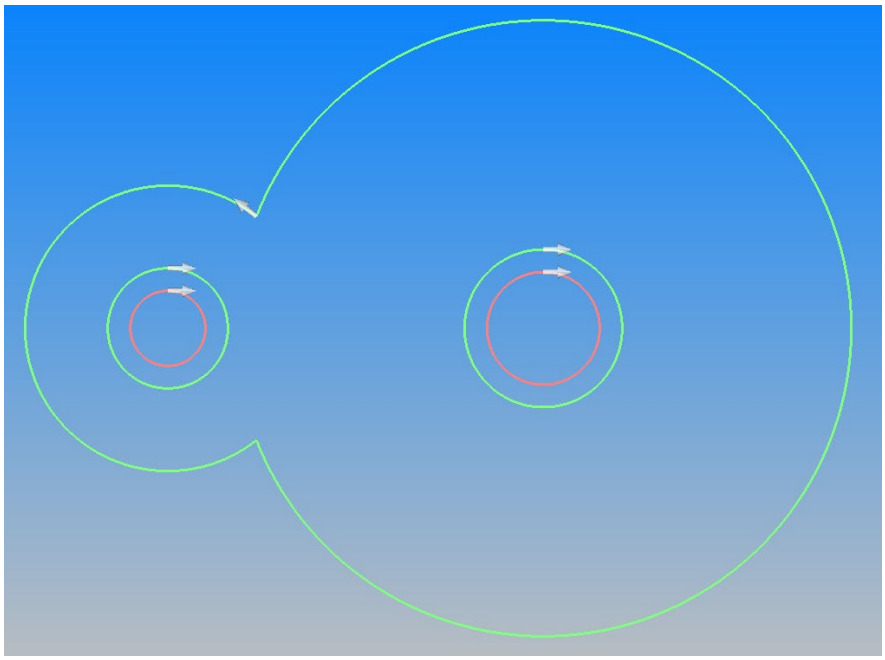


Figure 26 - Complete United profile as shown by a single Ghost Tool

Main pocket Creation, Method 2

Using the Trim command

Select **EDIT > Break Join etc. > Trim**
The system will prompt **TRIM: Select Cutting Geometries**, **<LClick>** both circles to select them.

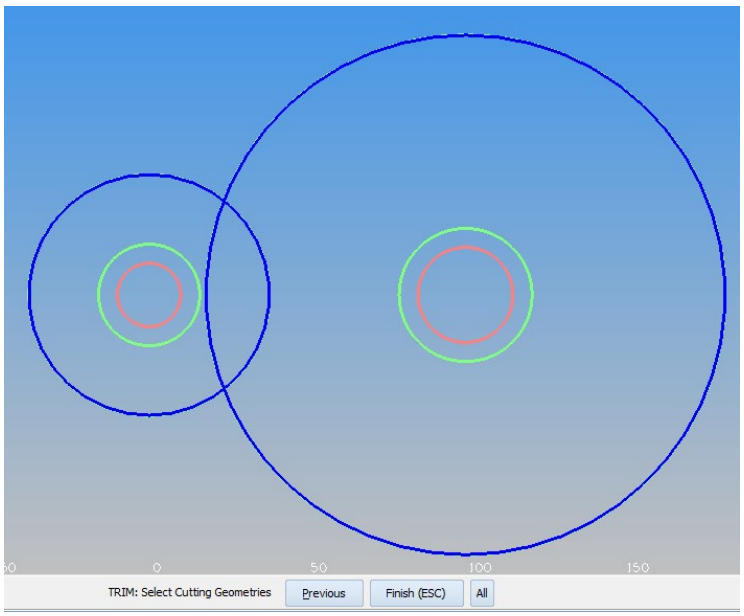


Figure 27 - Circles selected for trimming

<RClick>, **<Esc>** or **<LClick>** on the **[Finish (ESC)]** button to continue.
The system will prompt **TRIM: Pick Geometry/Toolpath to Trim**, **<LClick>** the cursor on the two sections of the circles that are not required.

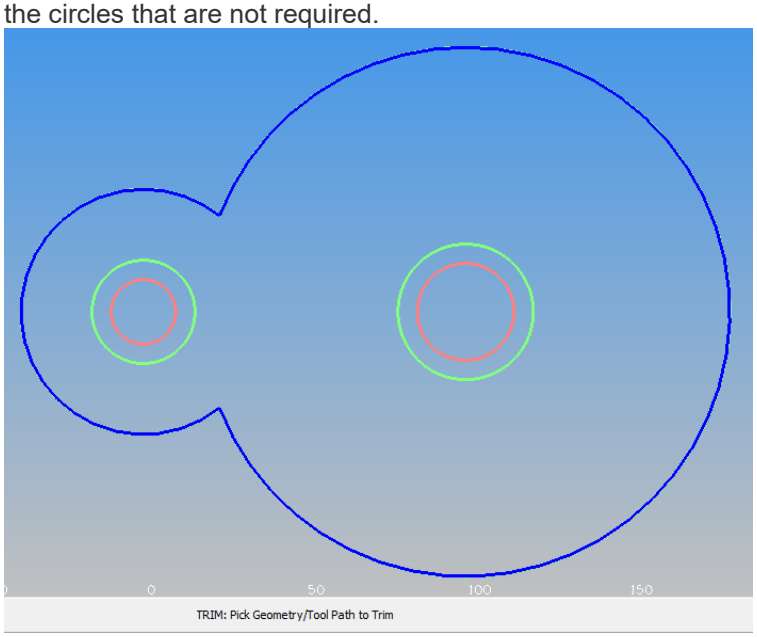


Figure 28 - Sections of circle not required Trimmed away

<RClick> to finish the command.

Join the profile

Select **<Ctrl> + <G>** or **VIEW > Display Options > Ghost Tools**  to display the ghost tools.

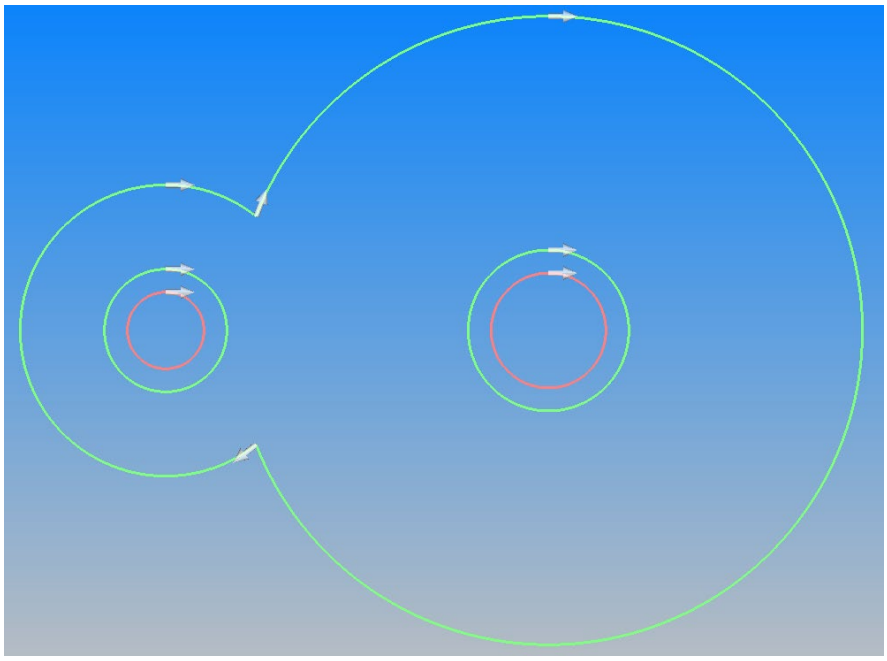


Figure 29 - Four ghost tools after using Trim

Select **EDIT > Break Join Etc. > Join** 
<LClick> on each part of profile 6 so that it is all (selected) blue, **<RClick>** or press **<Esc>** this will action the command and if drawn properly profile 6 will only be displayed with one ghost tool.

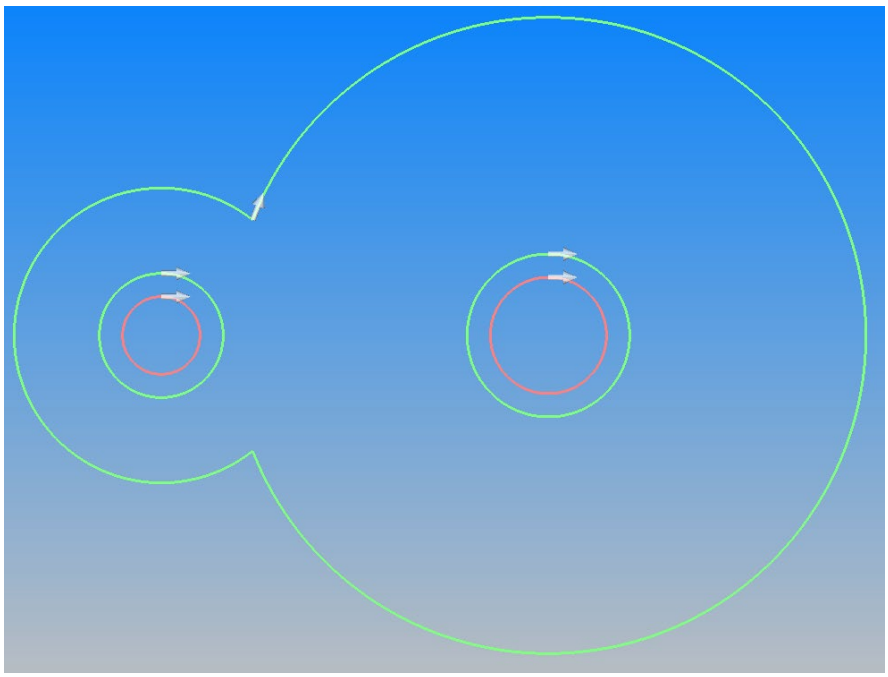


Figure 30 - Profile 6 joined.

Create the indents

To create the indent circles, we will use the **Bolt Hole Circle** command.
This creates a number of equally spaced circles around a specified construction circle.

Select **GEOMETRY > Special Geometries > Bolt Hole Circle**
The following dialogue is displayed. Complete as shown.

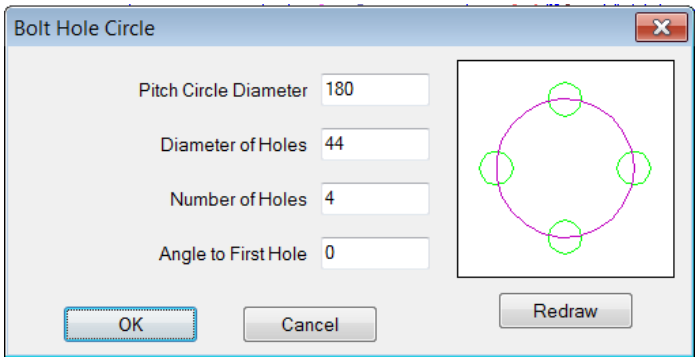


Figure 31 - Bolt Hole Circle Dialogue

For the Circle Centre select the centre of profile 10 at X100 Y0.

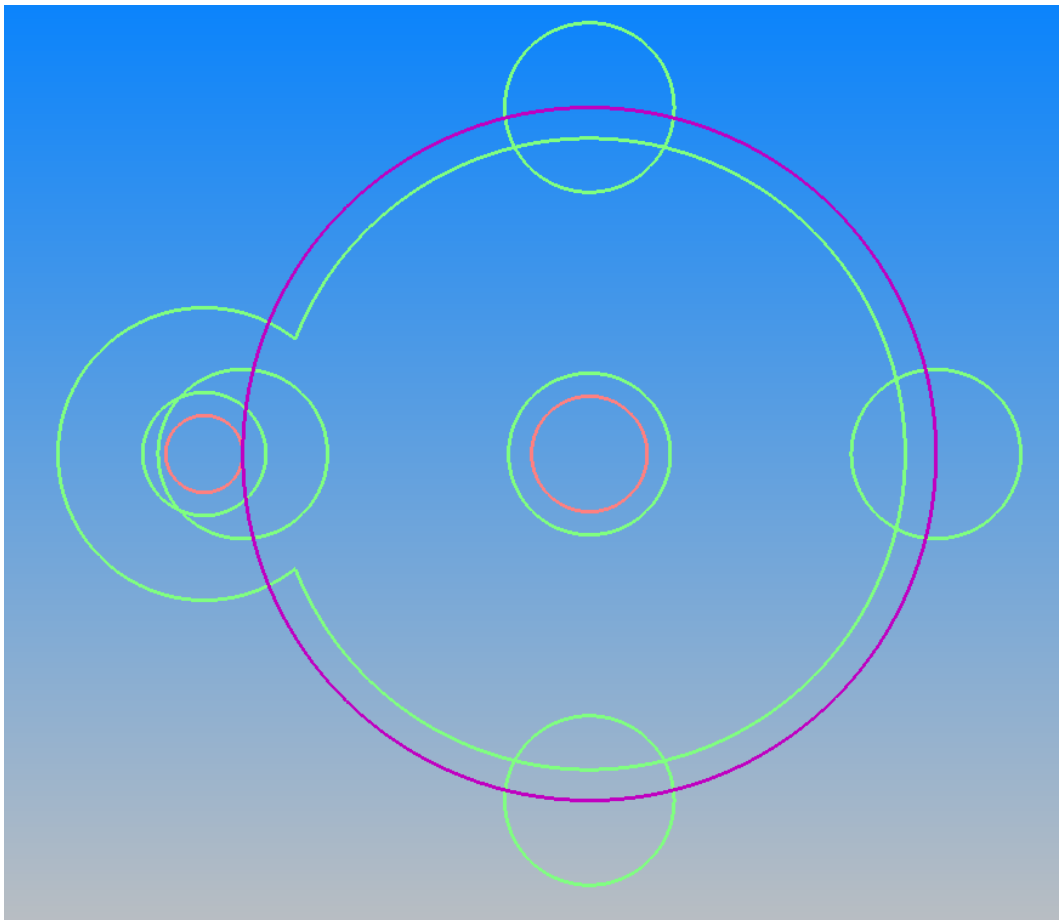


Figure 32 - Applied Bolt Hole Circles

Remove the circles from profile 6.

Select **EDIT > Fabricate > Subtract**
For the **Geometries to Subtract to, <LClick>** the bolt hole circle group.

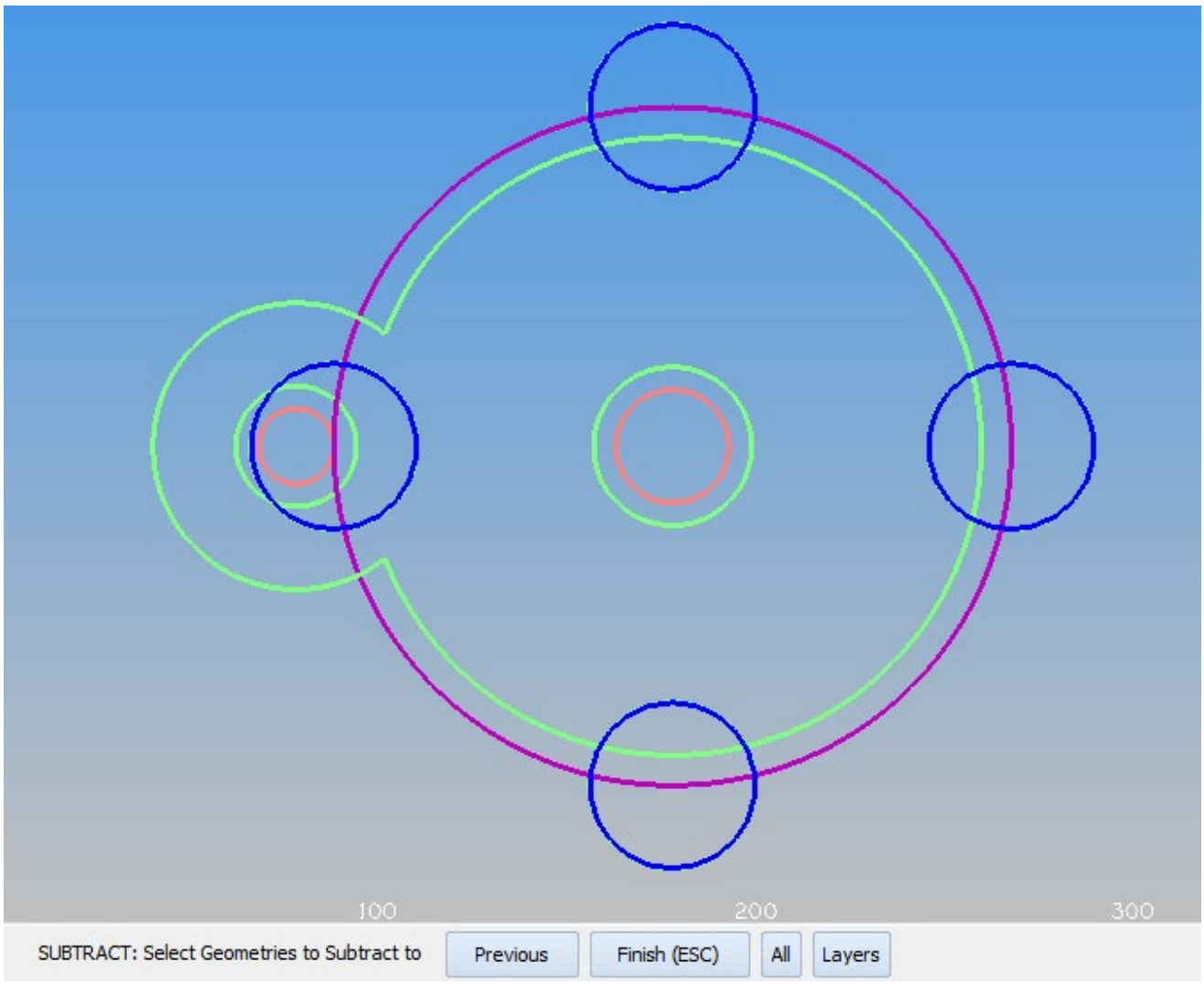


Figure 33 - Selecting the geometries to cut with

Note that creating the circles using the Special Geometries option automatically **Groups** the items to make selection easier in cases like this.

<RClick> to finish this section of the command.

For the **Geometries to Subtract from, <LClick>** Profile 6, the combined figure of eight shape.

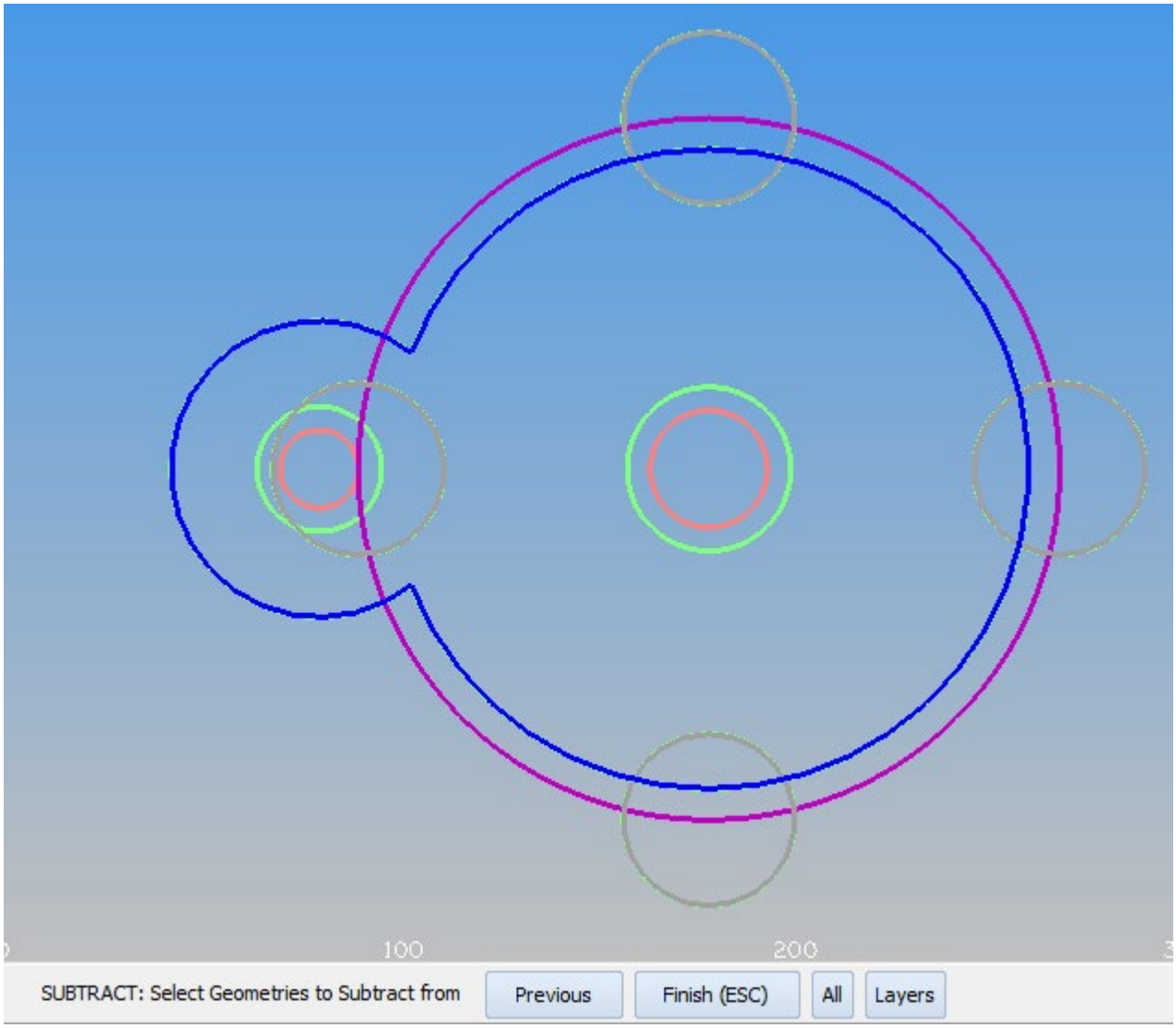


Figure 34 - Selecting the geometries to be cut

<RClick> to finish the command.

In the Project Manager **Layers** page.

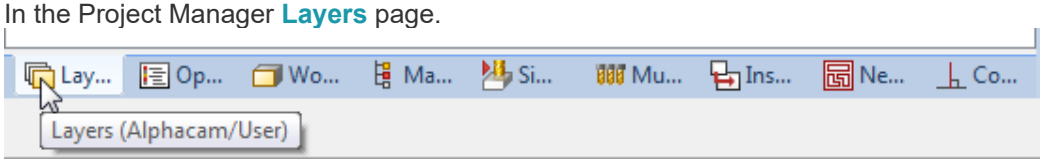


Figure 35 - Project Manager Layers

<LClick> the small symbol next to the User Layers to expand those.
Then **<LClick>** again on the small symbol next to *** Main Pocket**, then again on the small symbol next to the word **Geometry** to expand and make all the drawn items visible.

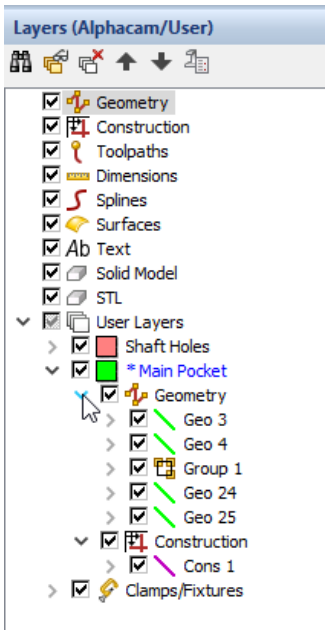


Figure 36 - Expand Geometry

<LClick> on **Group 1** and keep the left mouse button held down to allow you to drag the group so that the word **Construction** second down from the top is highlighted, then release the left mouse button to move the geometries to the **Construction** layer.

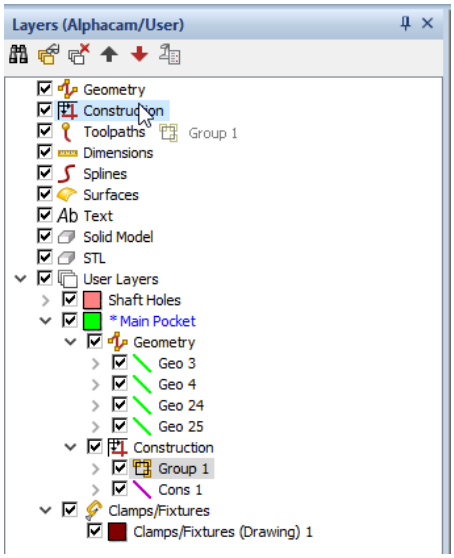


Figure 37 - Drag & Drop

There is nothing wrong in leaving the group on the main **Construction** layer if you wish.
To place it onto the current active **User Layer**, you will need to perform the drag and drop function again to place it onto the required user layer.

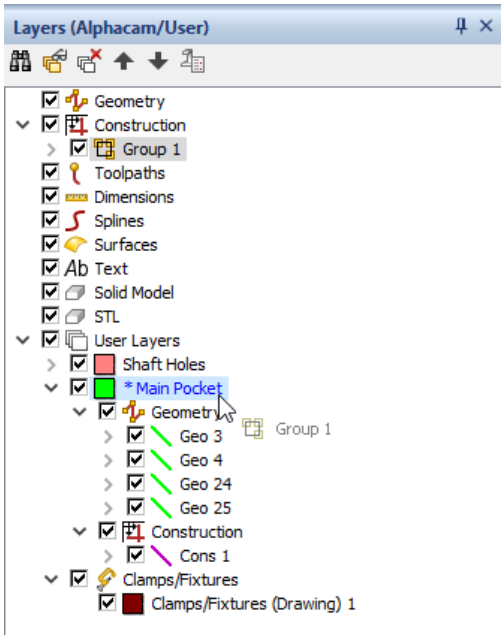


Figure 38 - Relocate form the Primary Construction layer

<LClick> on **Group 1** and keep the left mouse button held down to allow you to drag the group so that the word *** Main Pocket** is highlighted, then release the left mouse button to move the geometries to the **Construction** layer inside **Main Pocket**.

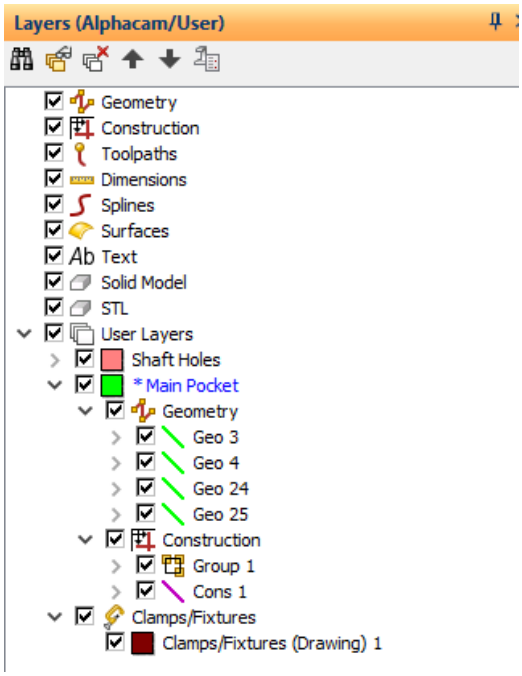


Figure 39 - Correct layout

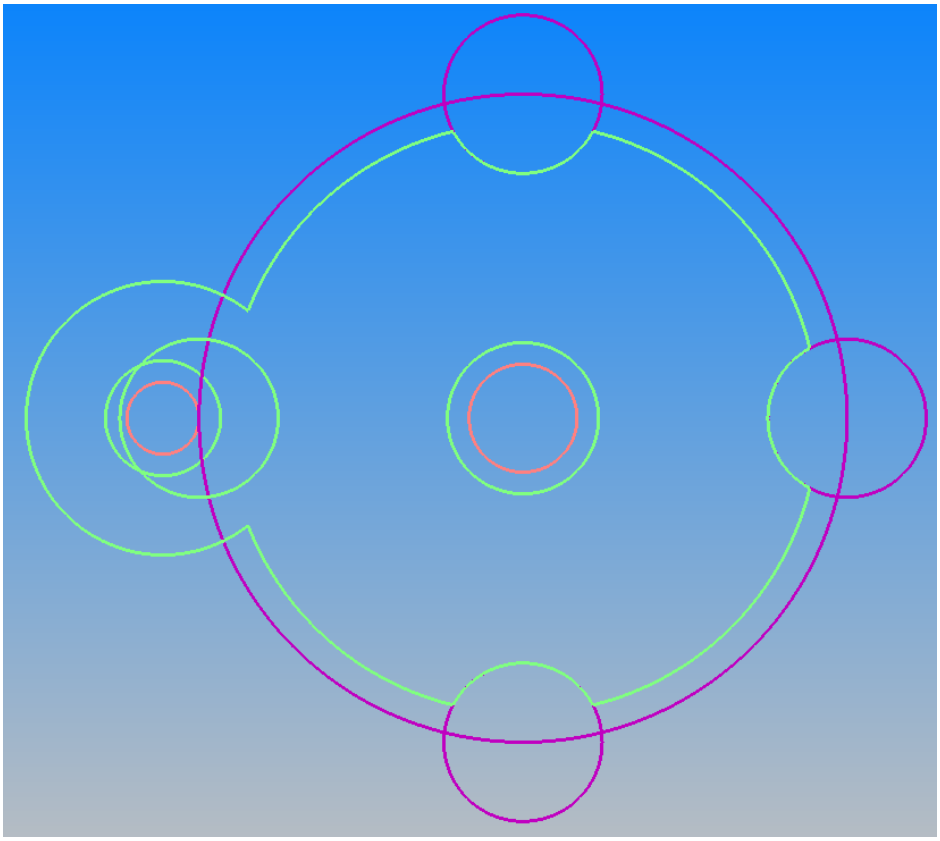


Figure 40 - Cutting circles as Construction entities

Untick the **Construction** layer to turn off the construction elements.

> It is good drawing practice to not delete items as you may need them for reference later. Moving them to different layers and then turning off those layers is a neater solution.

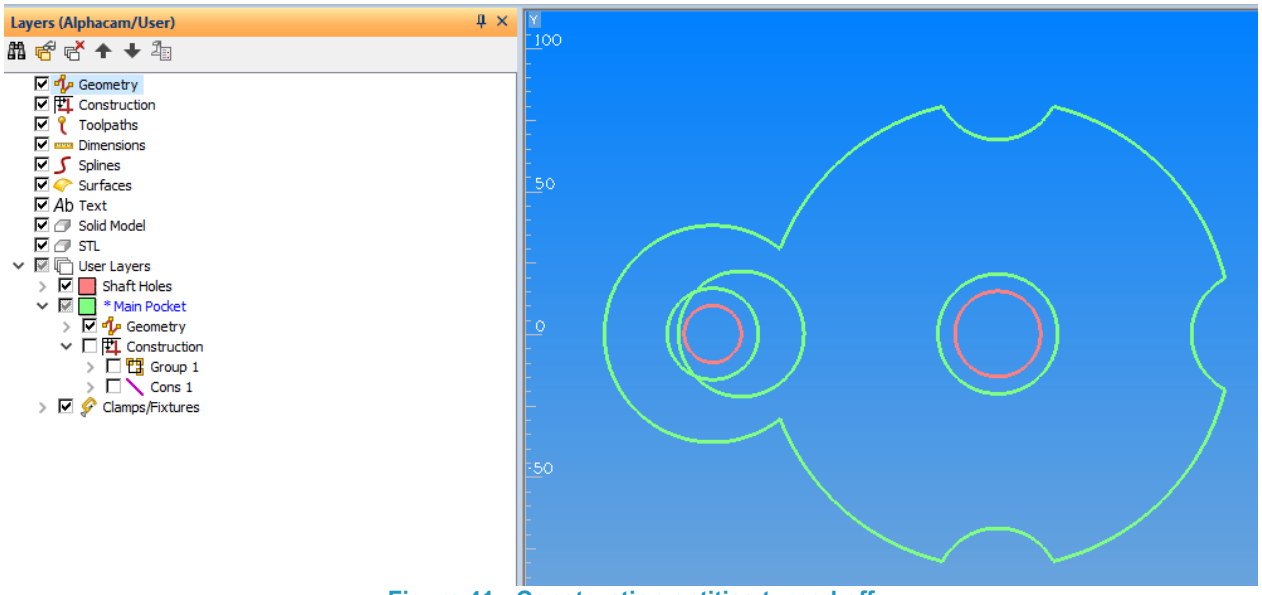


Figure 41 - Construction entities turned off

An extra Ø44 circle is created because a circle was subtracted from within the bounding profile.

Select **EDIT > Delete** and select the redundant circle. **<RClick>** or **<Esc>** to complete the selection process, then **[OK]** to confirm deletion of the extra geometry circle. Lastly, **<RClick>** or **<Esc>** to finish the Delete command completely.

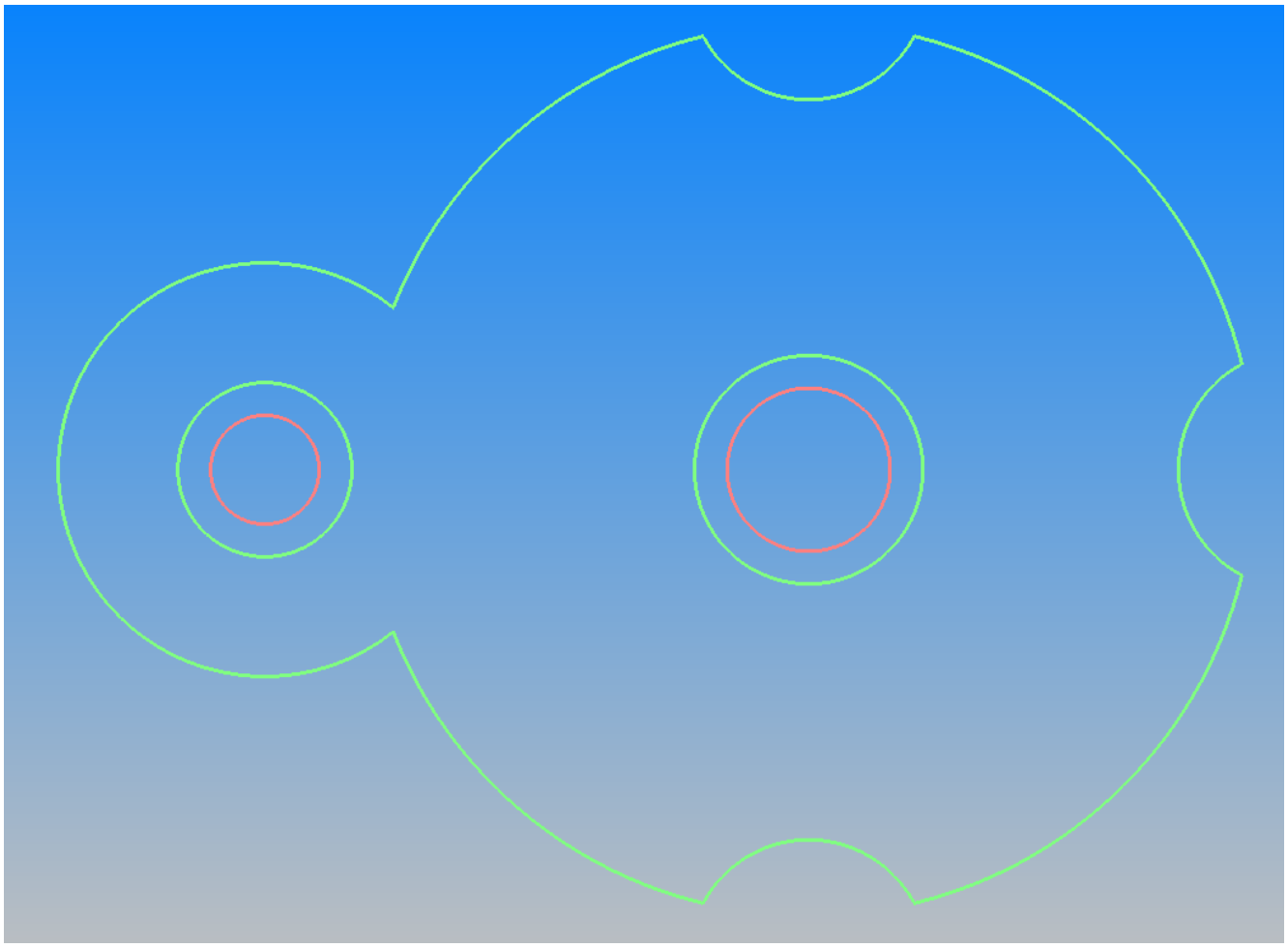


Figure 42 - Completed profile ready to fillet application

Finalising the profile fillets

Applying the main R5 Fillets

The Fillet command is a very flexible tool allowing not just the application of fillets via the dialogue, but also the modification of existing fillets on a profile.

Select **EDIT > Break Join etc. > Fillet**

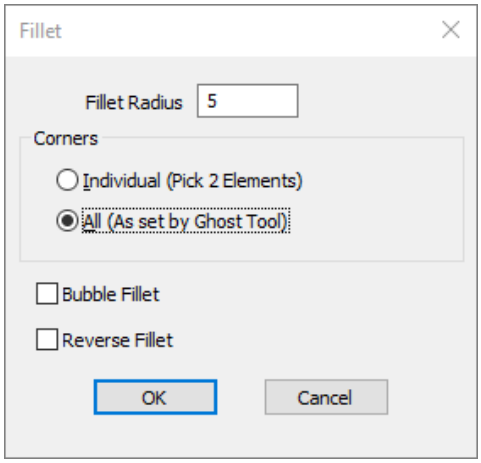


Figure 43 - Fillet Command Dialogue

Enter 5 for the radius and select **⊙ All (as set by Ghost Tool)** then **<LClick> [OK]**.
<LClick> the newly created profile as the item to apply the fillets to.
<RClick> to apply the fillets, **<RClick>** a second time to complete the command.

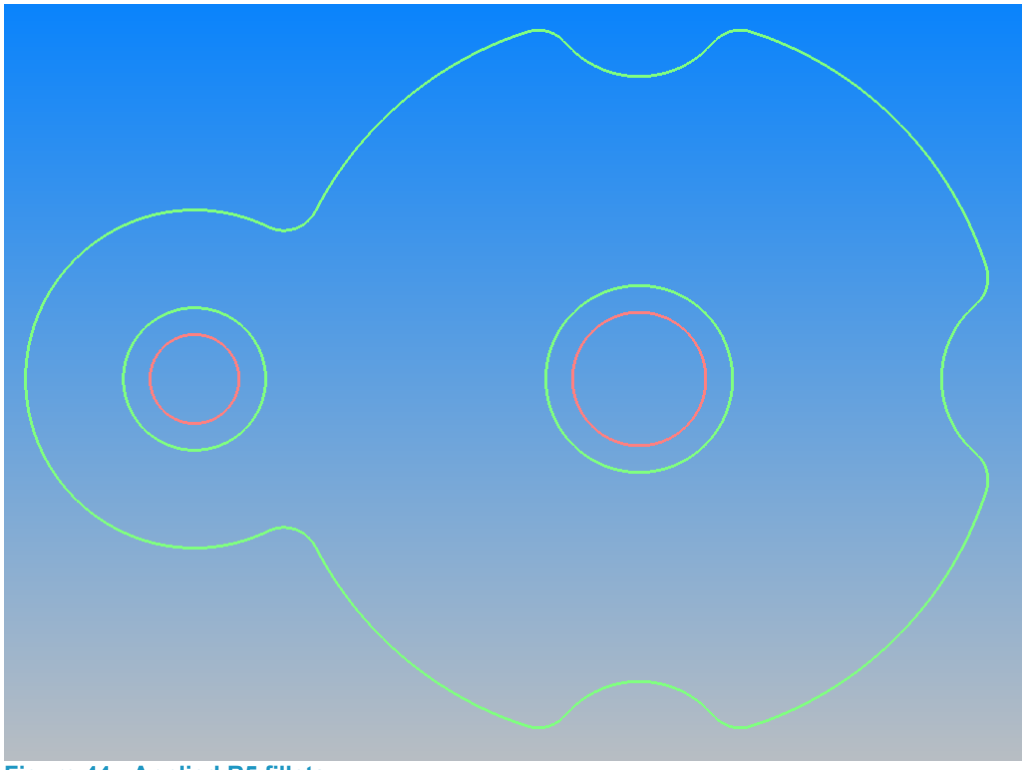


Figure 44 - Applied R5 fillets

Adjusting the two R13 Fillets

Select **EDIT > Break Join etc. > Fillet**

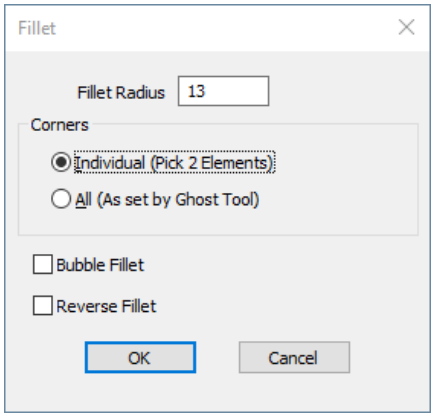
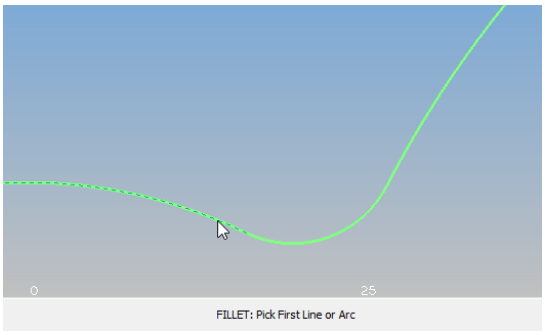


Figure 45 - Fillet command dialogue to adjust the R13 items

Setting the option to ⊙ **Individual (Pick 2 elements)** allows you to alter an existing fillet by **<LClick>** on the two elements each side of the existing filet and applying a new size.
<LClick> the two arcs on each side of one of the R5 fillets that need to be adjusted to R13.



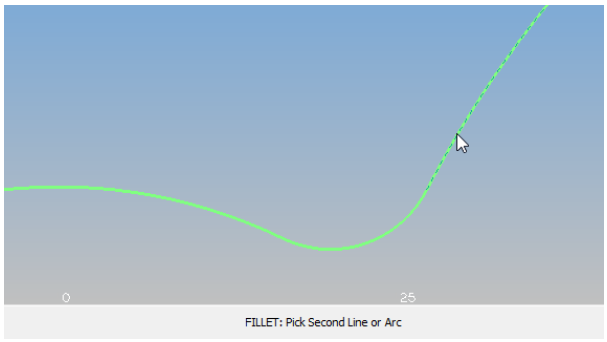


Figure 46 - Selecting the two elements

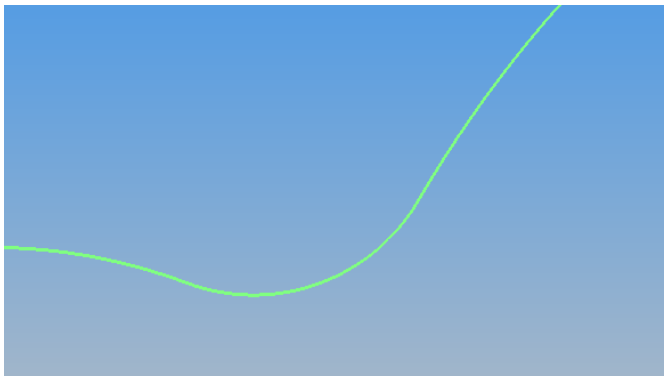


Figure 47 - Modified R13 fillet

Repeat the above operation for the second R13 fillet modification.

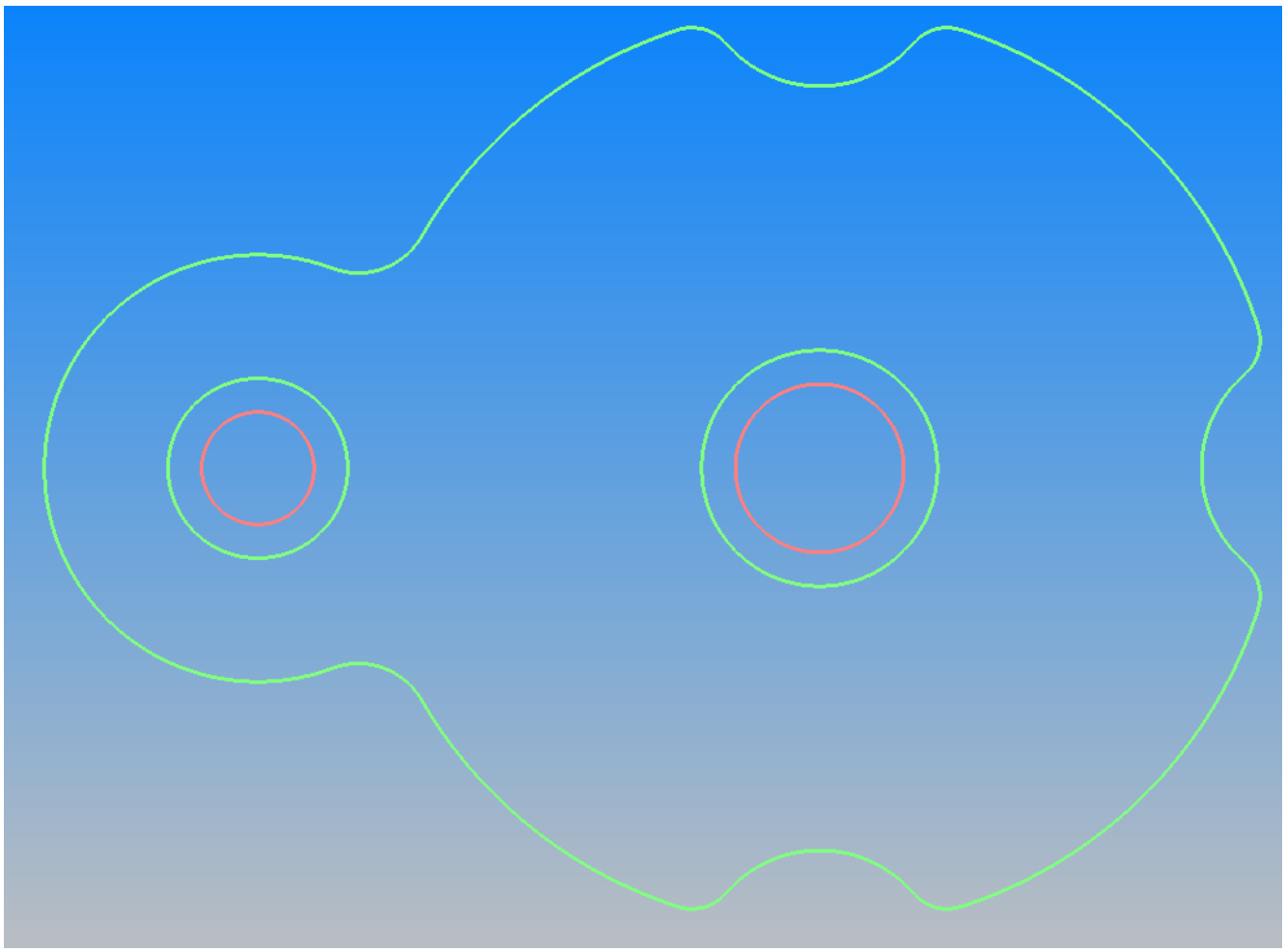


Figure 48 - Completed main pocket detail

Draw profile 2

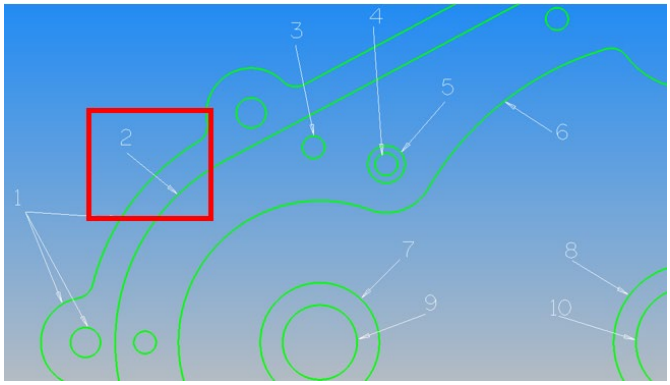


Figure 49 - Profile 2

Create and activate the User Layer

The next profile is part of a different section of the drawing, so requires a new **User Layer**.

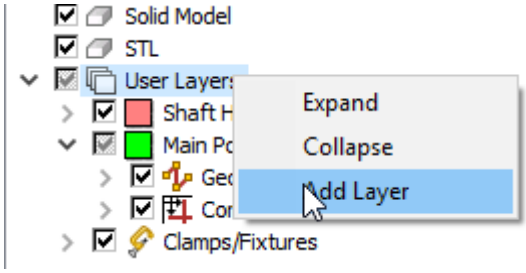


Figure 50 -<RClick> options for a new User Layer

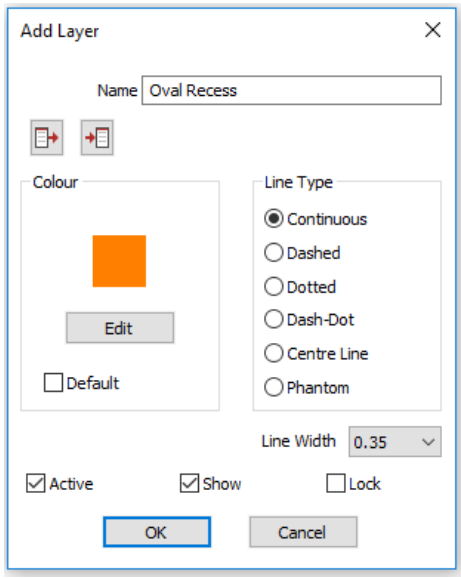


Figure 51 - User Layer Oval Recess

<LClick> [OK] to continue.

Draw the R55 and R102 circles

Prior to activating the next command ensure that **Auto Snap** is active (is the word **Auto** highlighted at the bottom right of the screen?). To activate Auto Snap press **<F2>**.

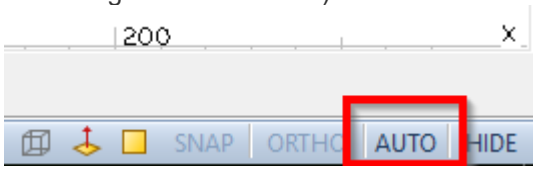


Figure 52 - Auto Snap active indicator

Select **GEOMETRY > Circle > Centre + Radius**

The command line will prompt you to enter the circle radius.
Type **55 <Enter>,** position the cursor over the centre of profile 1 (the snap selector will be displayed) **<LClick>** to select this position. The 55mm radius circle will be drawn.

The command line will again ask you for the circle radius.
Type **102 <Enter>,** position the cursor over the centre of profile 2 (the snap selector will be displayed) **<LClick>** to select this position. The 102mm radius circle will be drawn.
<RClick> to finish the command.

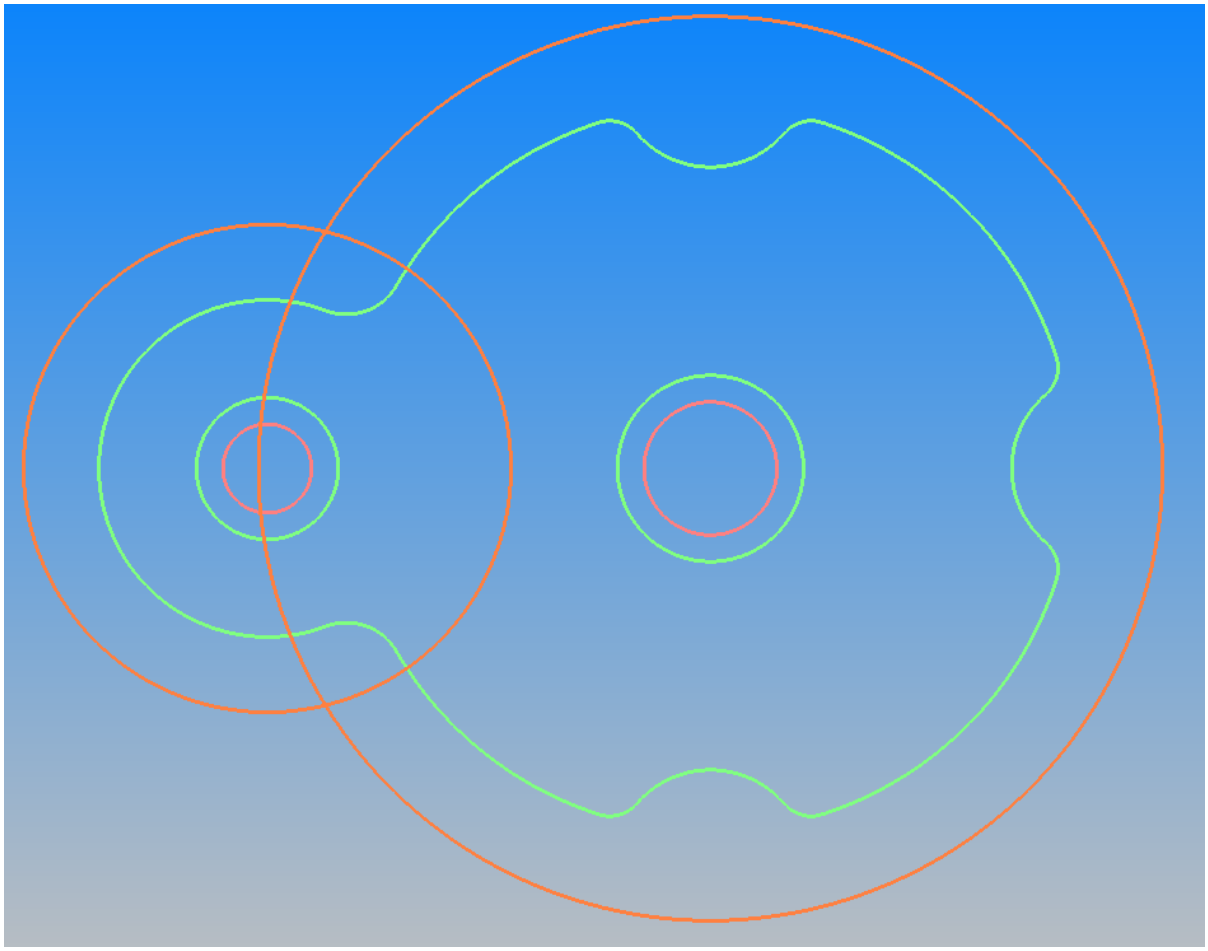


Figure 53 - Large Circles applied

Add Tangent Lines

Prior to creating the tangent lines ensure that Auto Snap is deactivated.
To deactivate Auto Snap press **<F2>**.

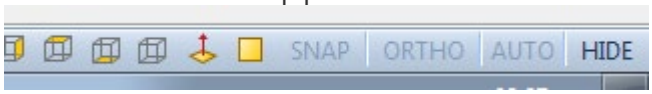


Figure 54 - Auto Snap off

> Although it is not mandatory to turn off Auto Snap we do so because the Auto Snap only snaps to End, Mid, Centre & Quadrant points and it can be confusing when the position to be selected is a tangent point that could be very close to one of these locations.

Select **GEOMETRY > Line**

For the **LINE From** point press **<F10>**, the prompt will change to select circle or arc.

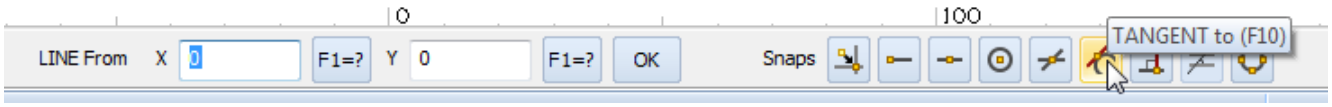


Figure 55 - Tangent Line From

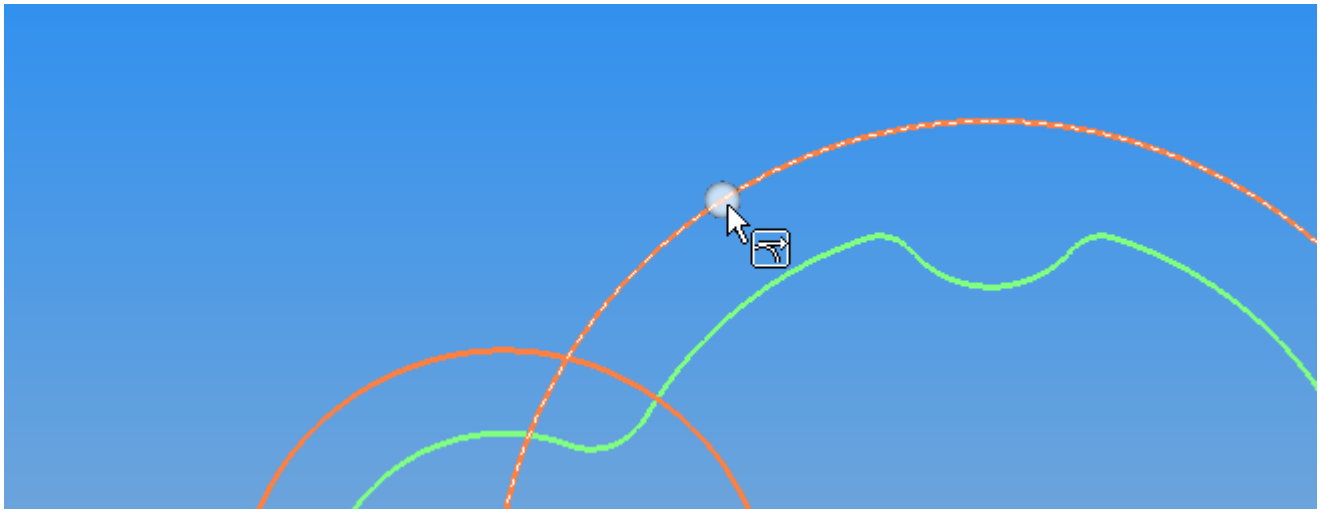


Figure 56 - Tangent from point

Select a position as shown above.

The elastic cursor line will be displayed, and this will remain tangent to the R55 circle as it is moved.

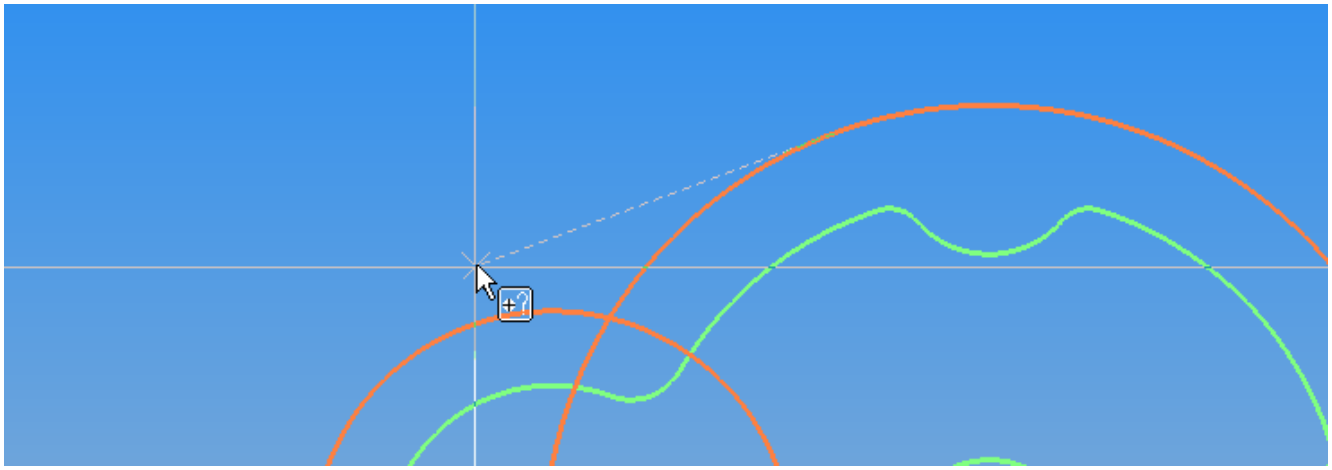


Figure 57 - Elastic Cursor Line

For the **LINE to** point press **<F10>** again.
The prompt will change to select circle or arc.

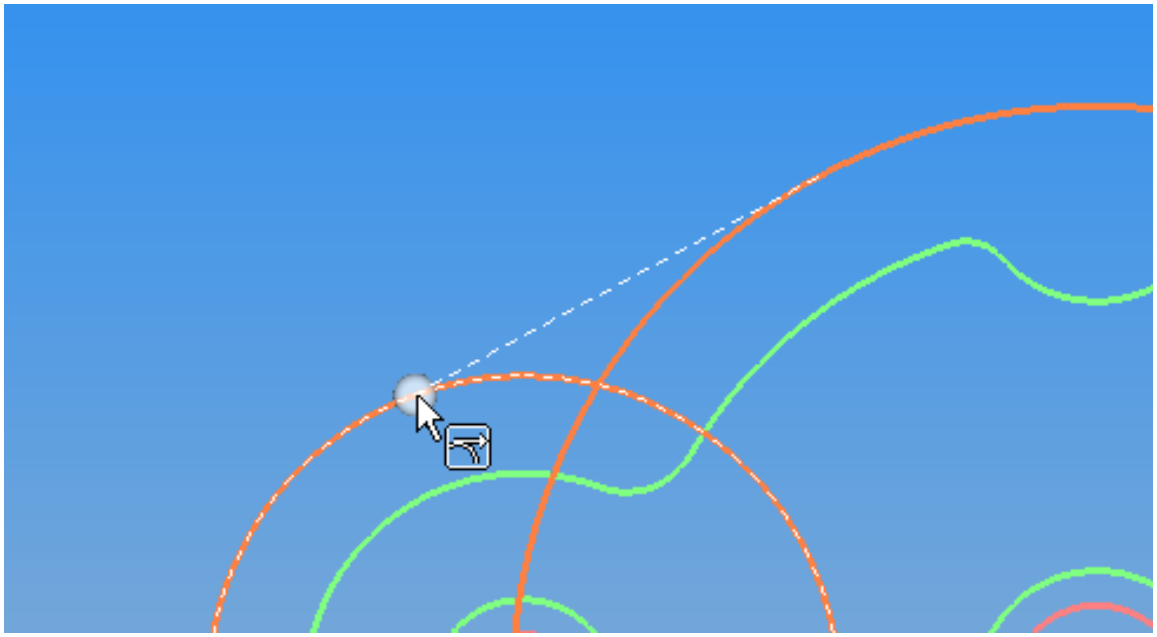


Figure 58 - Second selection point

Select a position as shown above.
The tangent line will be created **<RClick>** or press **<Esc>** to finish the line command.

Press **<Space>** to reactivate the Line command.

For the **LINE From** point press **<F10>** the prompt will change to select circle or arc. Select a position towards the bottom left of the R55 circle.

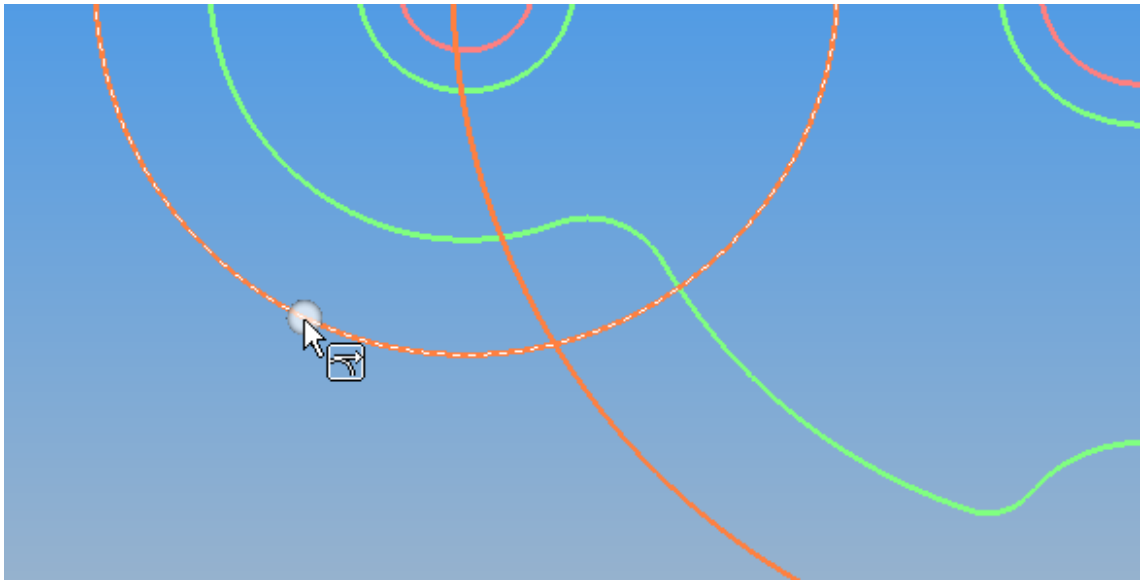


Figure 59 - Second Tangent Line start point

The elastic cursor line will be displayed, and this will remain tangent to the R55 circle as it is moved.
For the **LINE To** point press **<F10>** the prompt will change to select circle or arc. Select a position towards the bottom right of the R102 circle.

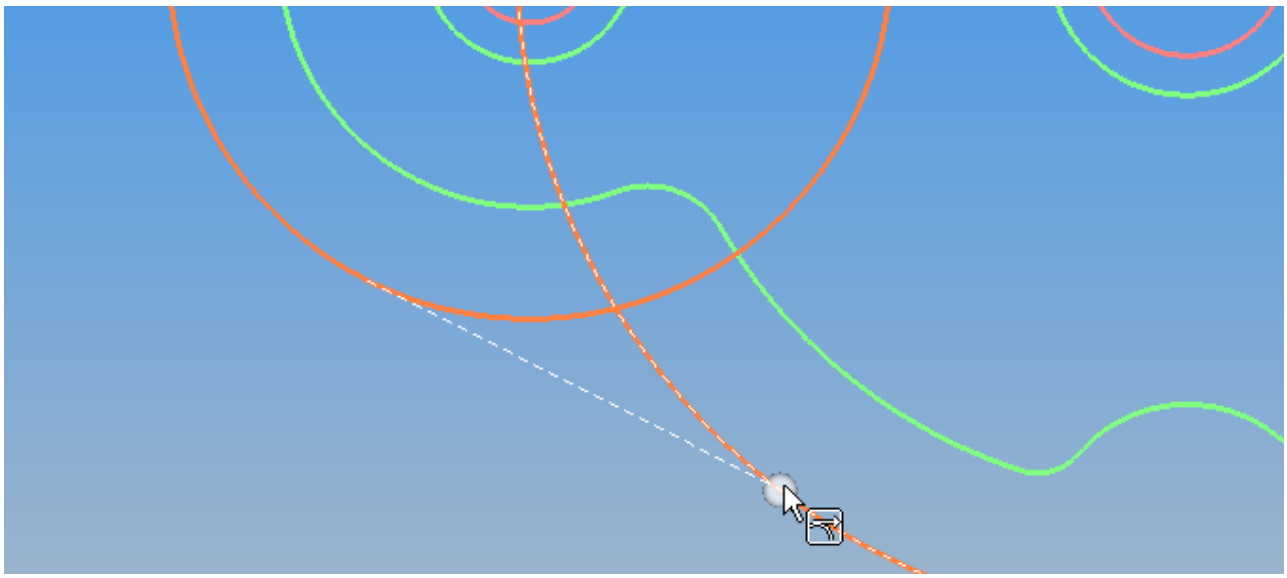


Figure 60 - Second Tangent Line end point

The tangent line will be created **<RClick>** or press **<Esc>** to finish the line command.

Note that the snap options are non-modal (single hits) and need to be selected each time you wish to use them.

Trim the Circles to the Lines

Select **EDIT > Break Join etc. > Trim** 
The system will prompt **TRIM: Select Cutting Geometries**, **<LClick>** both tangent lines (they should turn blue).

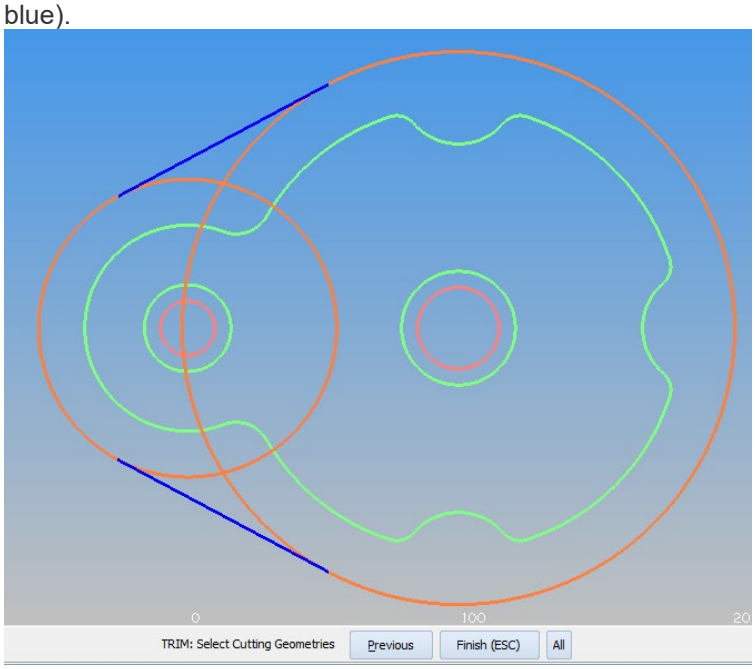


Figure 61 - Select Trimming Geometries

Press **<Esc>** or **<RClick>** or select the **[Finish (ESC)]** button.
The system will prompt **TRIM: Pick Geometry/Toolpath to Trim**, **<LClick>** the cursor on the portion of the circles to be removed.

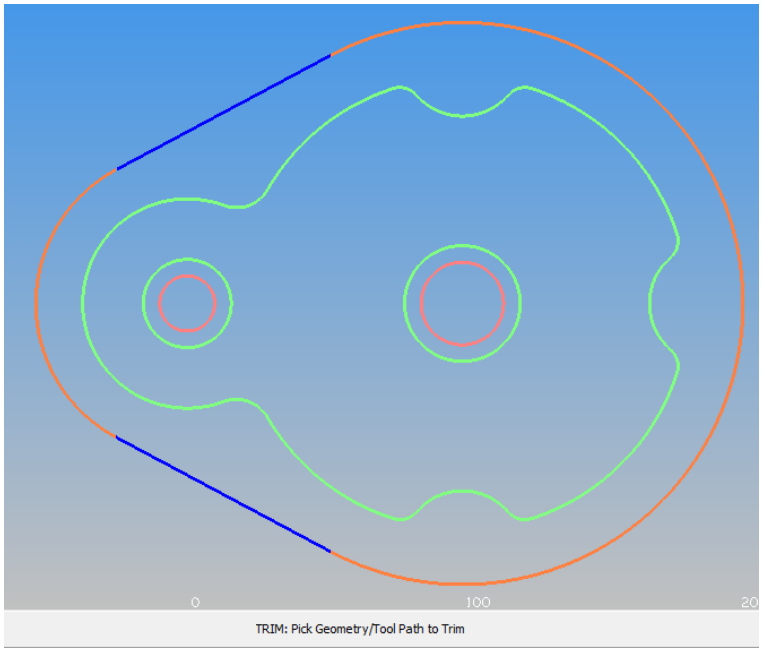


Figure 62 - Sections of unwanted circles removed

<RClick> to finish the command.

Join the profile

Select **<Ctrl> + <G>** or **View > Display Options > Ghost Tools**  to display the ghost tools

Select **EDIT > Break Join Etc. > Join** 

<LClick> on each part of profile 2 so that it is all (selected) blue, **<RClick>** or press **<Esc>** this will action the Join command and if drawn properly profile 2 will only be displayed with one ghost tool.

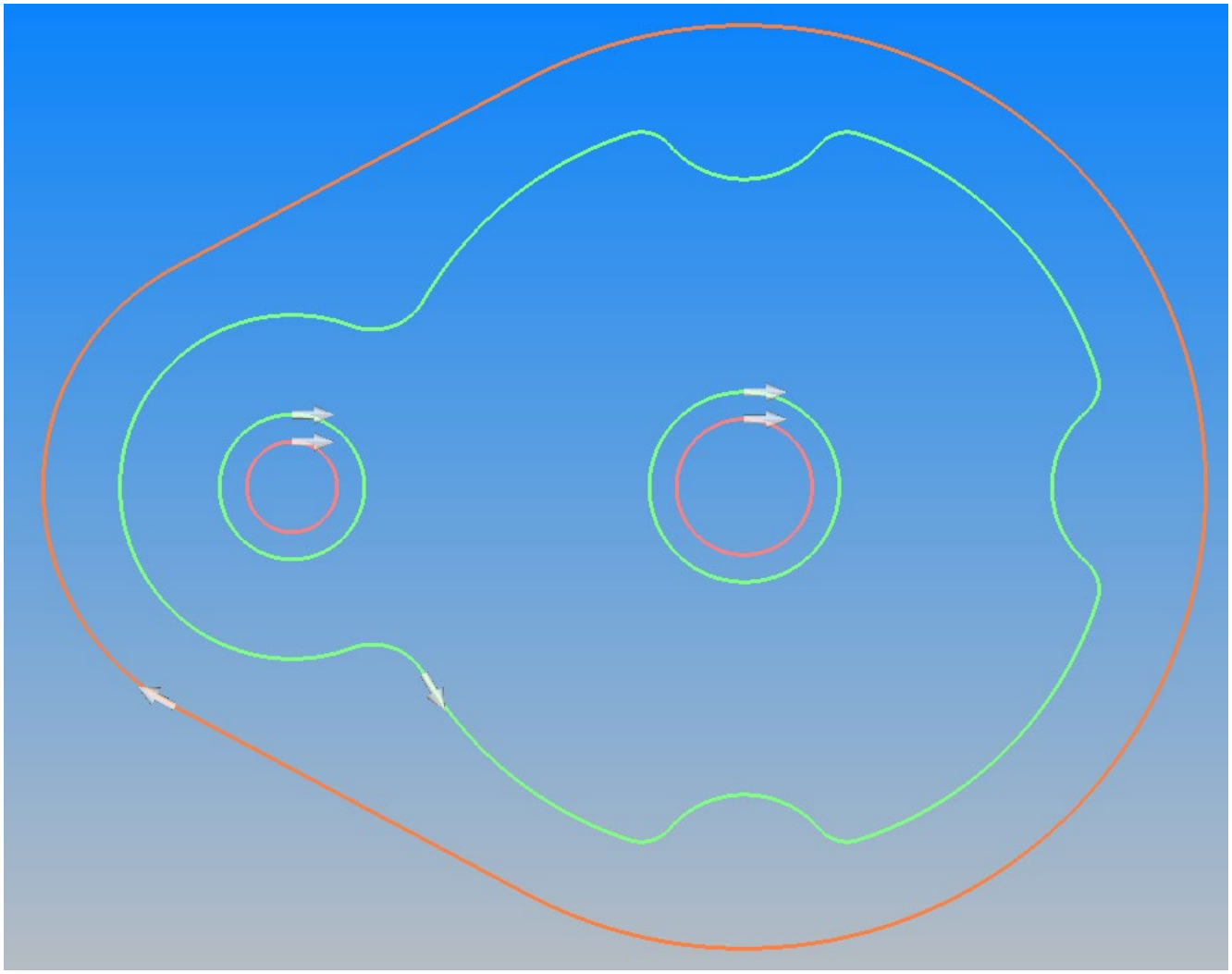


Figure 63 - Ghost tools display one complete profile to machine

Create profile 3

Profile 3 comprises of 9 holes of Ø6 equally spaced on a profile 8mm inside of profile 2 starting at left most quadrant point.

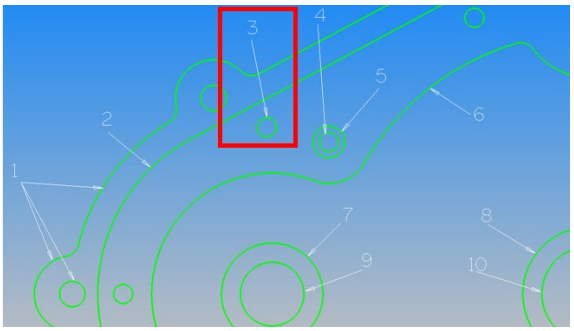


Figure 64 - Profile set 3

Create and activate the User Layer

The next profile is part of a different section of the drawing, so requires a new **User Layer**.

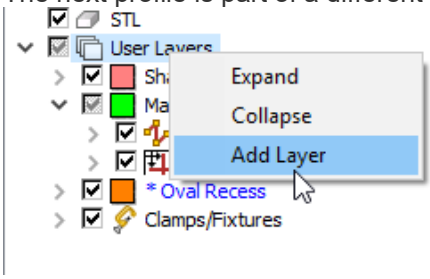


Figure 65 - -<RClick> options for a new User Layer

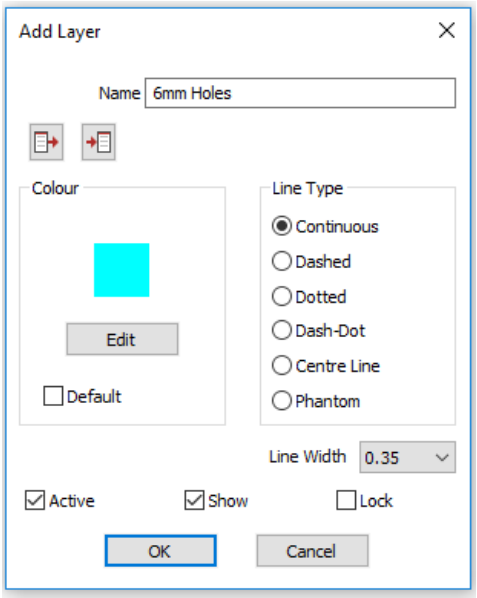


Figure 66 - User Layer 6mm Holes

<LClick> [OK] to continue.

Offset profile 2.

Select **EDIT > Break Join etc. > Offset** 

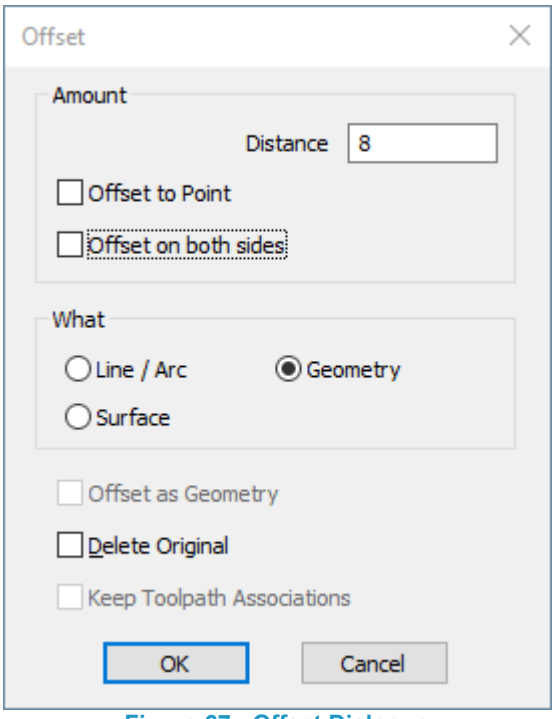


Figure 67 - Offset Dialogue

Set the options as shown then **<LClick> [OK]**.
<LClick> profile 2 then **<LClick>** a point on the inside, the new profile will be drawn.

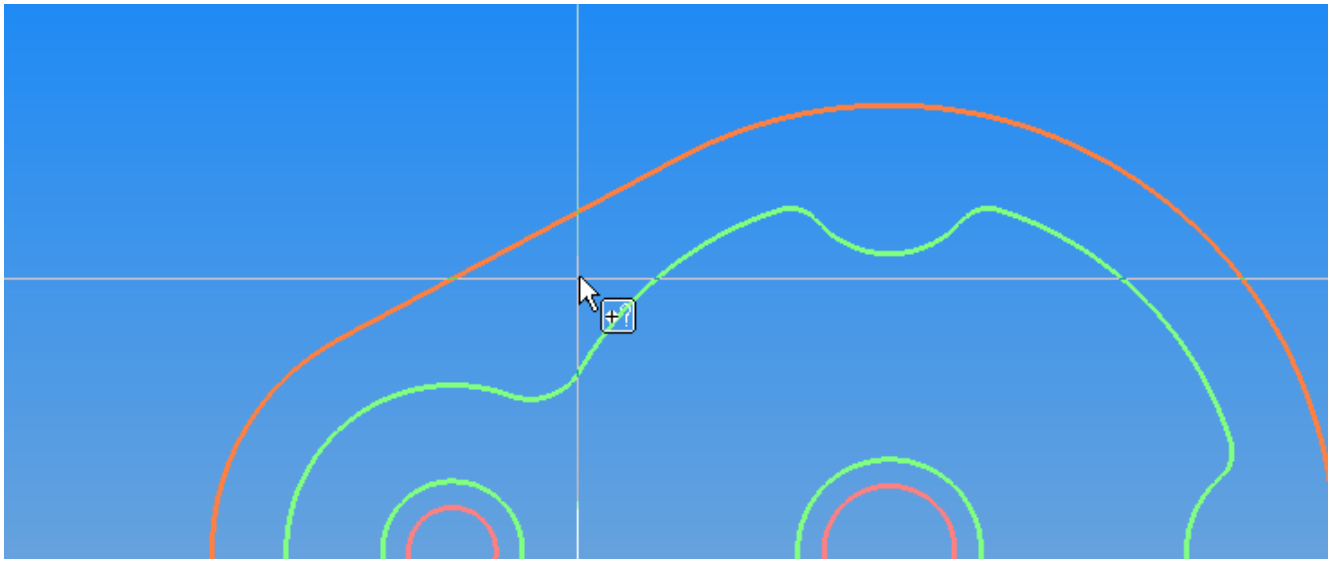


Figure 68 - Offset 8mm inside for the Equi-Spaced guide line

<RClick> to finish the command.

Reset the Start point

The Equi-Spaced Holes command uses the Start point of the selected geometry to place the First hole. To ensure the correct orientation of the holes we need to make sure that the **Start Point** for the last geometry we created is in the correct location.

Select **EDIT > Start Order > Start Pt** 

Ensure that Auto Snap **<F2>** is active then **<LClick>** the Left most quadrant point on the new profile.

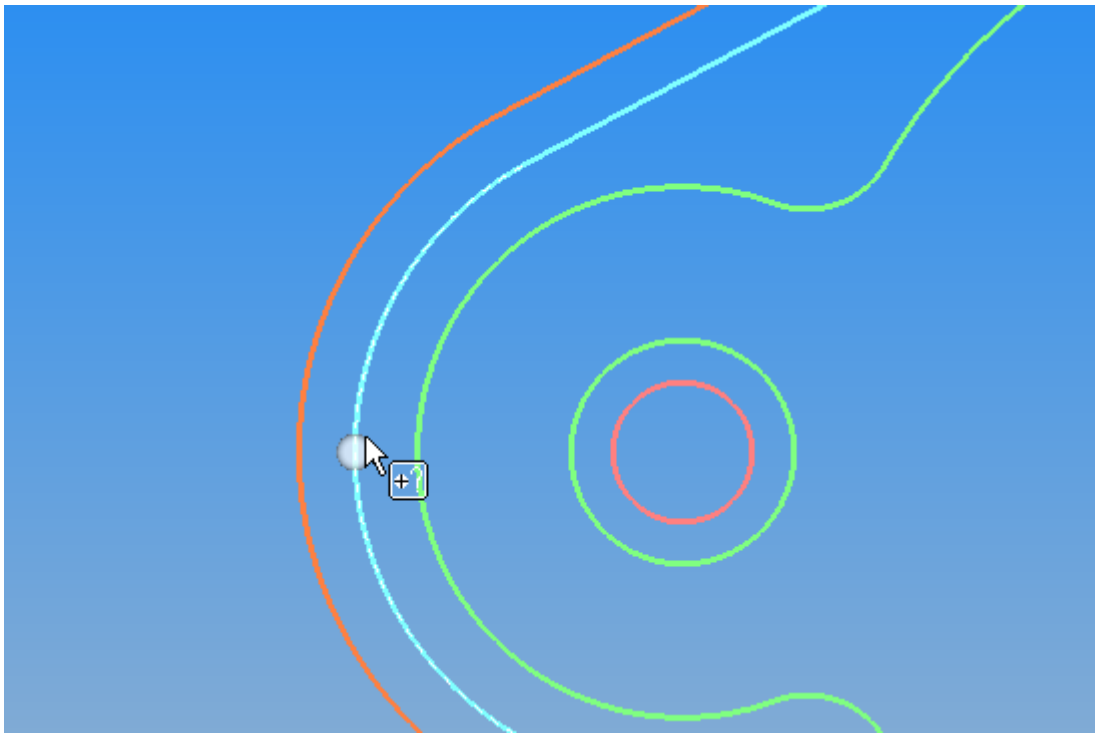


Figure 69 - Setting the correct Start Point for Equi-Spaced holes

<RClick> to finish the command.

Create the Equi-Spaced circles.

Select **GEOMETRY > Special Geometries > Equi-spaced holes** 

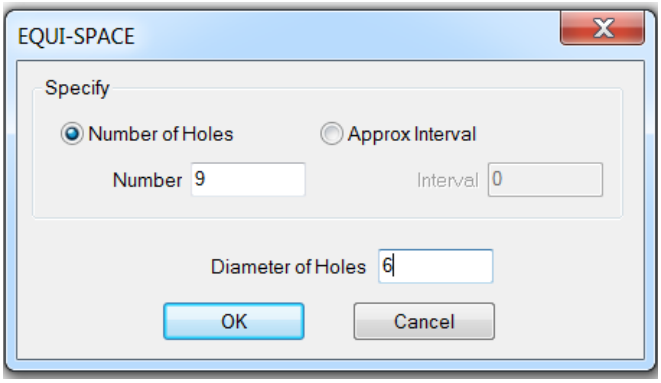


Figure 70 - Equi-Spaced Holes command Dialogue

Set the ⊙ **Number of Holes** to 9 and the **Diameter of Holes** to 6 then **<LClick> [OK]**.
<LClick> the new profile to select it, then **<RClick>** to action the command.

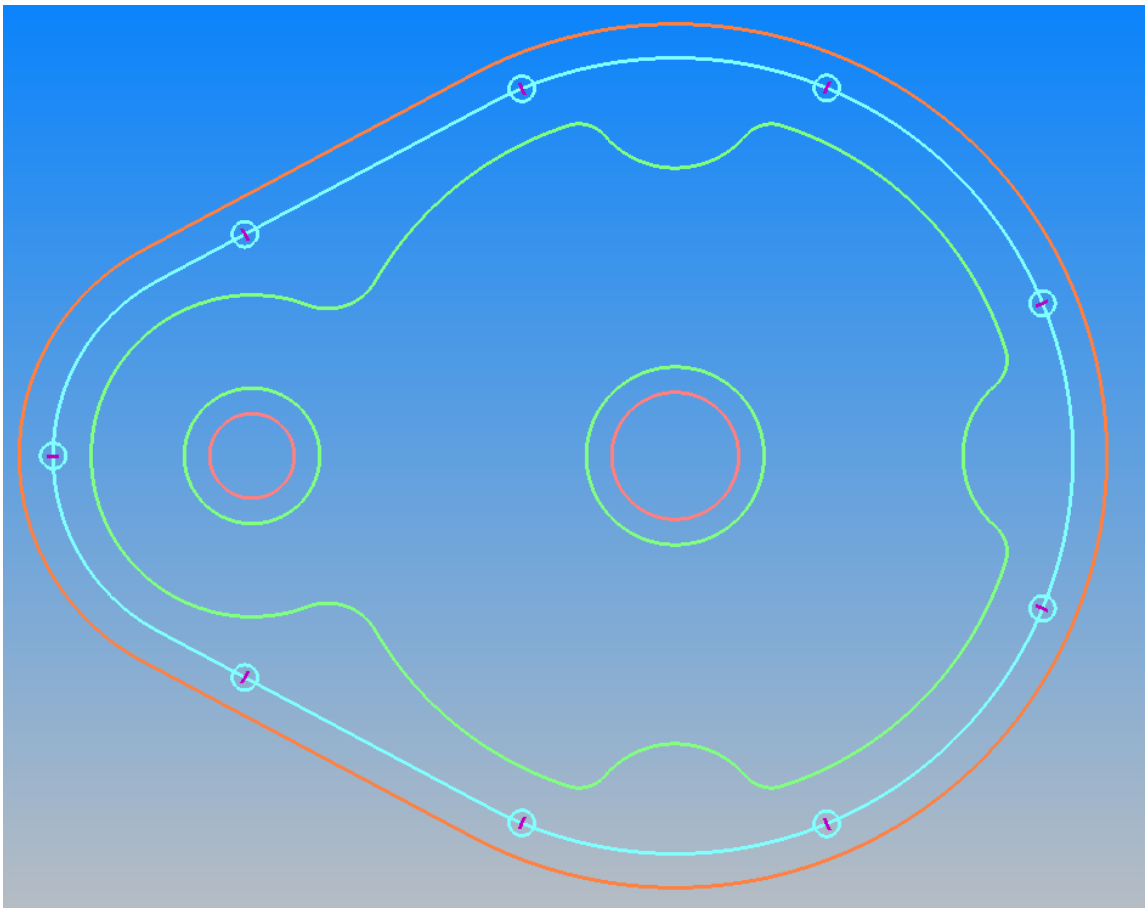


Figure 71 - Applied Equi-spaced circles

Managing the Drawing

Using **EDIT > Change** 

Set the options as follows.

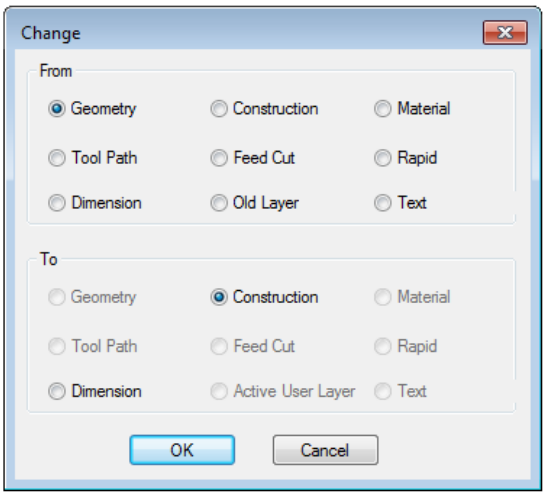


Figure 72 - Change options Dialogue

In the top section, select what the entity is at present, ⊙ **Geometry**.
In the lower section select what you would like the entity to alter into, ⊙ **Construction**.
<LClick> [OK] then, **<LClick>** the profile on which the holes were created.

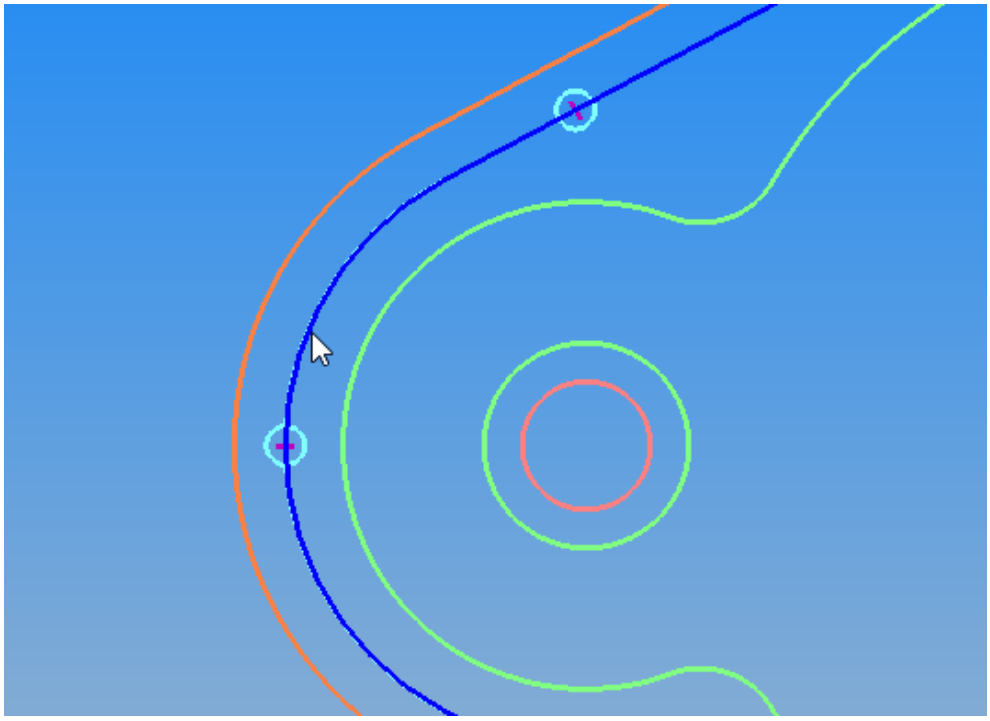


Figure 73 - Select the item to Change

<RClick> to complete the process.

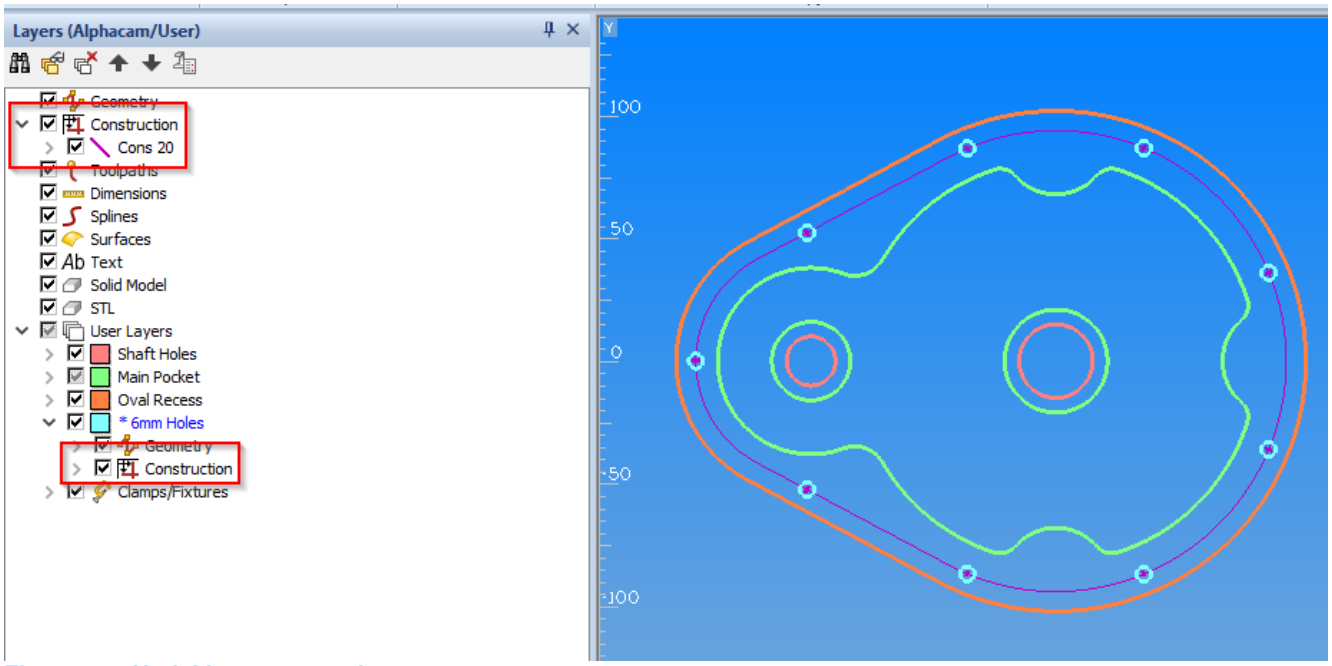


Figure 74 - Unticking construction

Untick the main **Construction** Layer to then hide the Changed geometry and untick the 6mm Holes layer **Construction** section to hide the small marker lines for the hole centres.

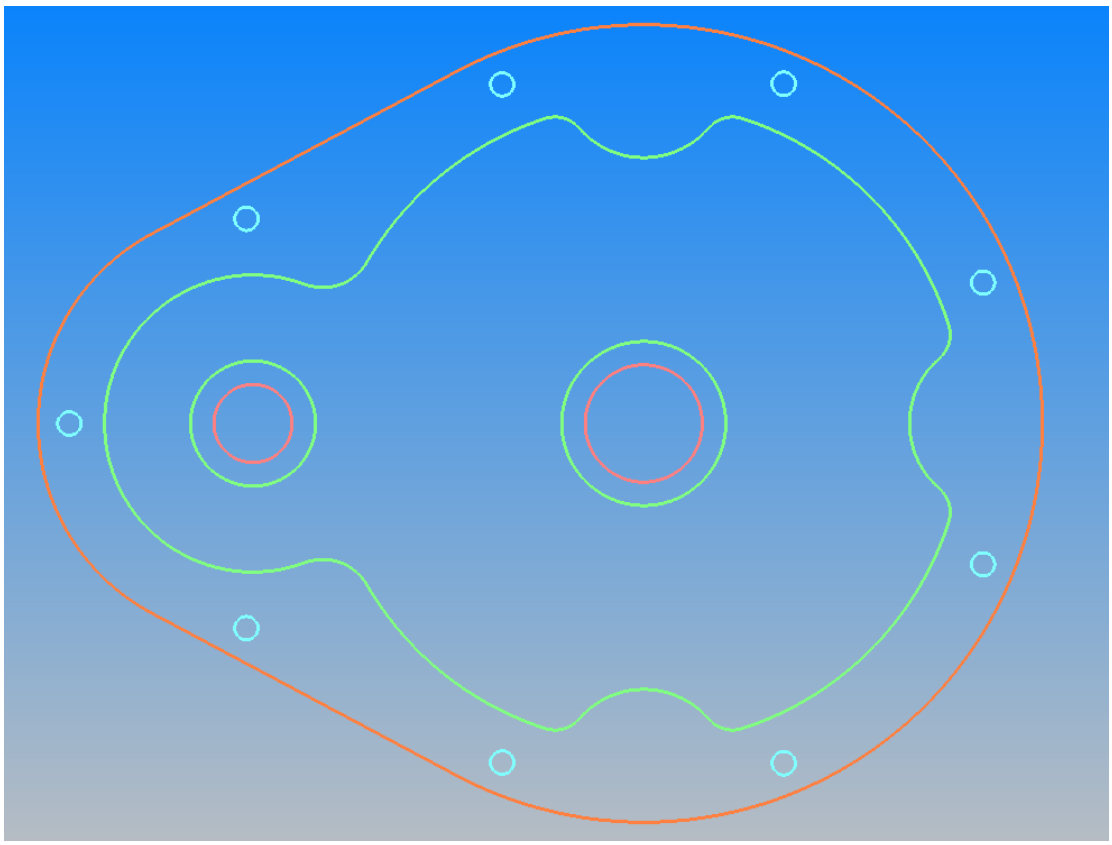


Figure 75 - Construction Geometry hidden.

Create profile 1

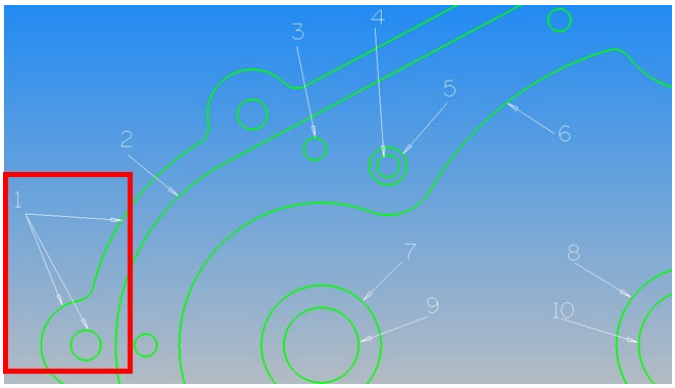


Figure 76 - Profile set 1

Create and activate the User Layer

The next profile is part of a different section of the drawing, so requires a new **User Layer**.

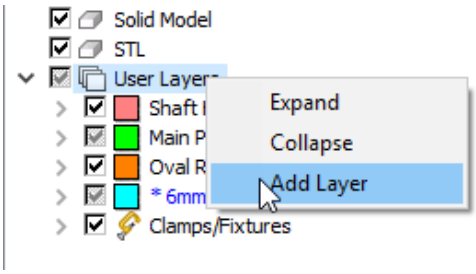


Figure 77 -<RClick> options for a new User Layer

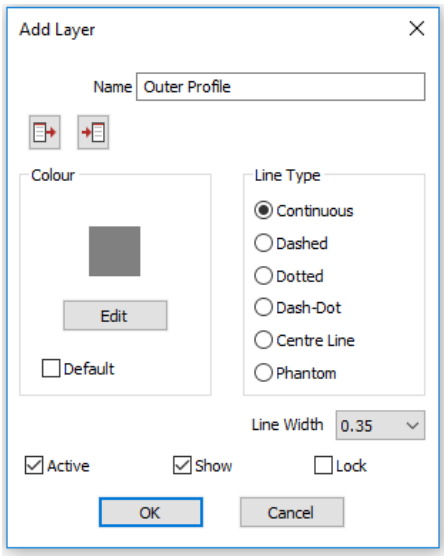


Figure 78 - User Layer Outer Profile

<LClick> [OK] to continue.

Profile 1 comprises of a profile the same as profile 2, 8mm outside with Ø8 holes and R12 lugs with 5mm fillets.

Offset profile 2.

Select **EDIT > Break Join etc. > Offset** 

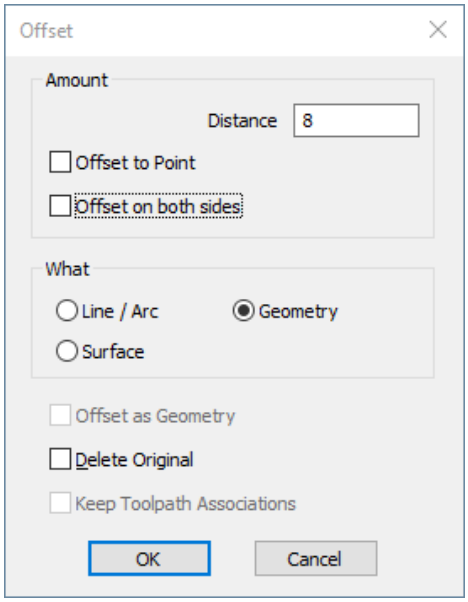


Figure 79 - Offset Dialogue

Set the options as shown then **[OK]**.
<LClick> profile 2 then **<LClick>** a point on the outside, the new profile will be drawn. **<RClick>** to finish the command.

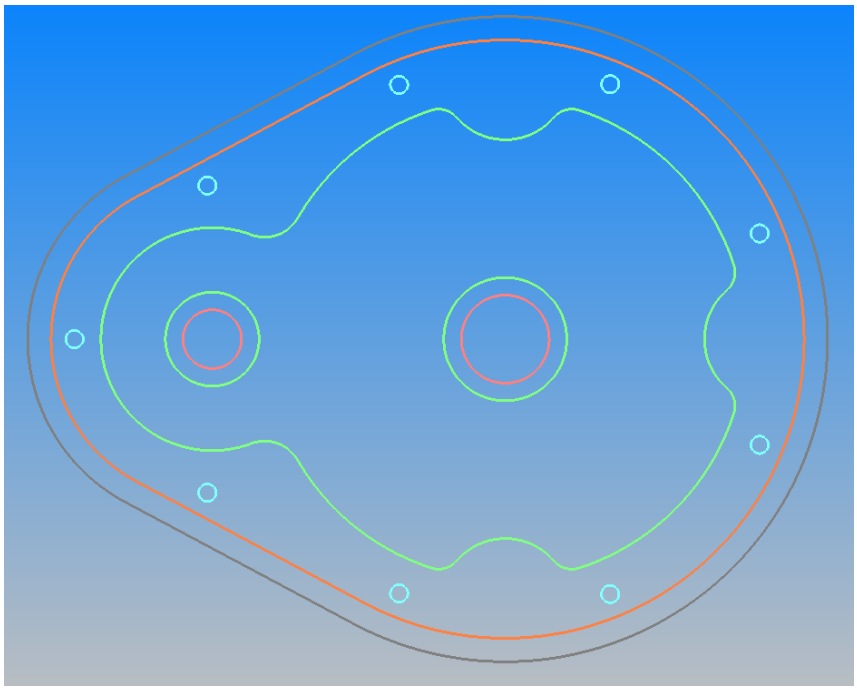


Figure 80 - The start of Profile 1

Hide unrequired geometries

To make the process of selection easier and to avoid errors, we will hide all of the geometries that we are not going to work on.

From the Project Manager, remove the tick from all the User Layers except **Outer Profile**.

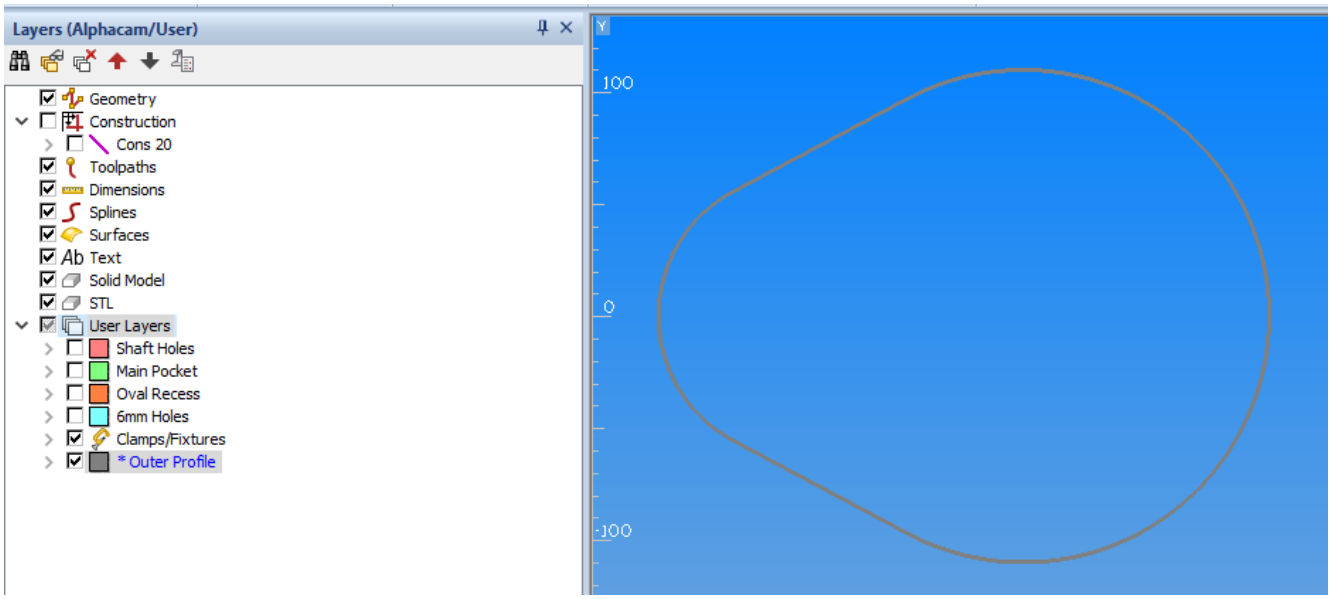


Figure 81 - Hiding Geometries by Unticking User Layers

Notes regarding the two offset profiles

It should be noted at this point that there is an option in the **EDIT > Break Join etc. > Offset**  which would make the creation of the two offset profiles easier.
The step by step option was used to guide you easily through the process.
There is nothing from stopping you using the **☑ Offset on both sides** option in the first instance.

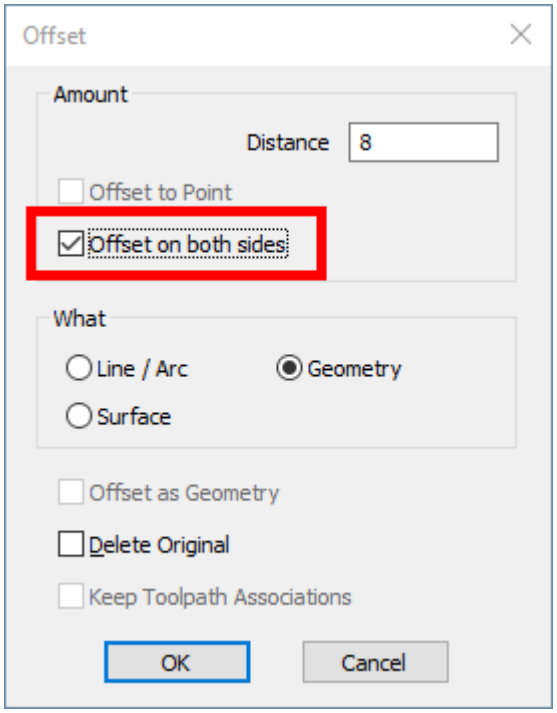


Figure 82 - Offset on both sides option

In doing this you would save yourself a few button clicks.

> You must also make sure that you select the correct offset line for the creation of the Equi-Spaced circles.

Create one Hole & Lug

Prior to using the next command ensure that Auto Snap is active (is the word **Auto** at the bottom right of the screen?). To activate Auto Snap press **<F2>**.

Select **GEOMETRY > Circle > Centre + Diameter** 

The command line will prompt you to enter the circle diameter.
Type **8 <Enter>** then position the cursor over the left most end of profile 1 as shown, (the snap selector will be displayed)

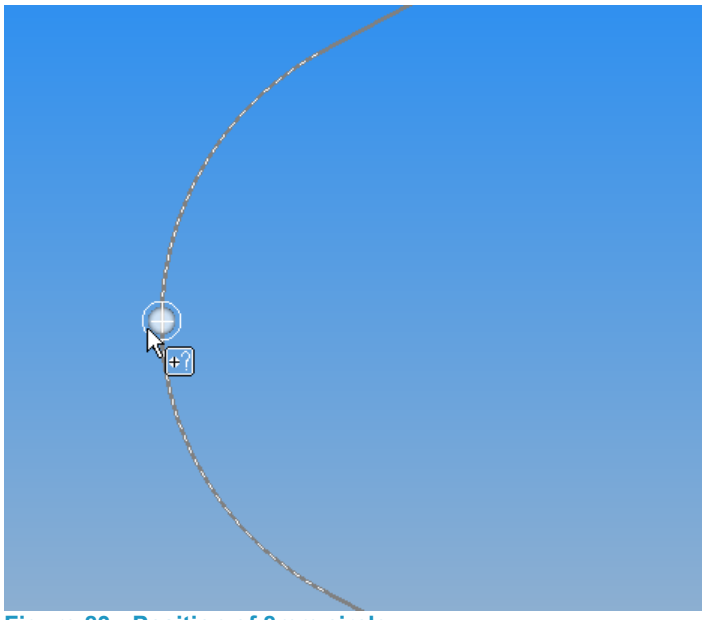


Figure 83 - Position of 8mm circle

<LClick> to select this position. The Ø8mm circle will be drawn. **<RClick>** to finish the command.

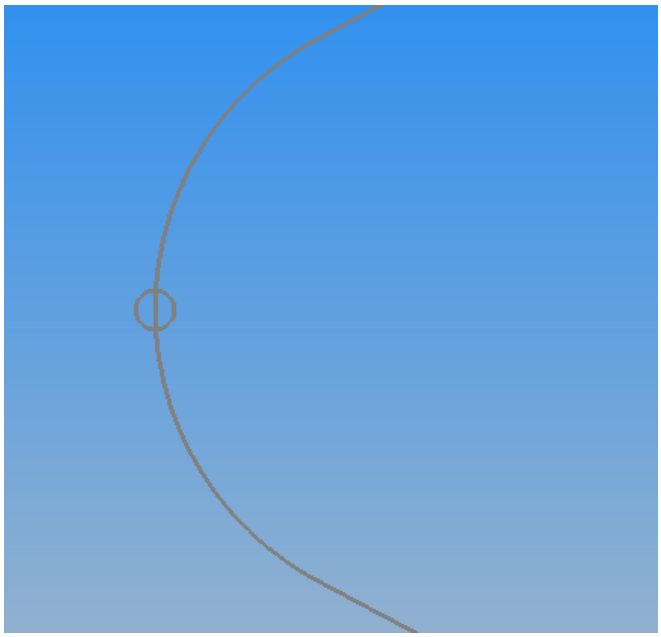


Figure 84 - 8mm Circle created

Select **GEOMETRY > Circle > Centre + Radius** 

The command line will prompt you to enter the circle radius.
Type **12 <Enter>,** position the cursor as previous, (the snap selector will be displayed) **<LClick>** to select this position. The R12mm circle will be drawn. **<RClick>** to finish the command.

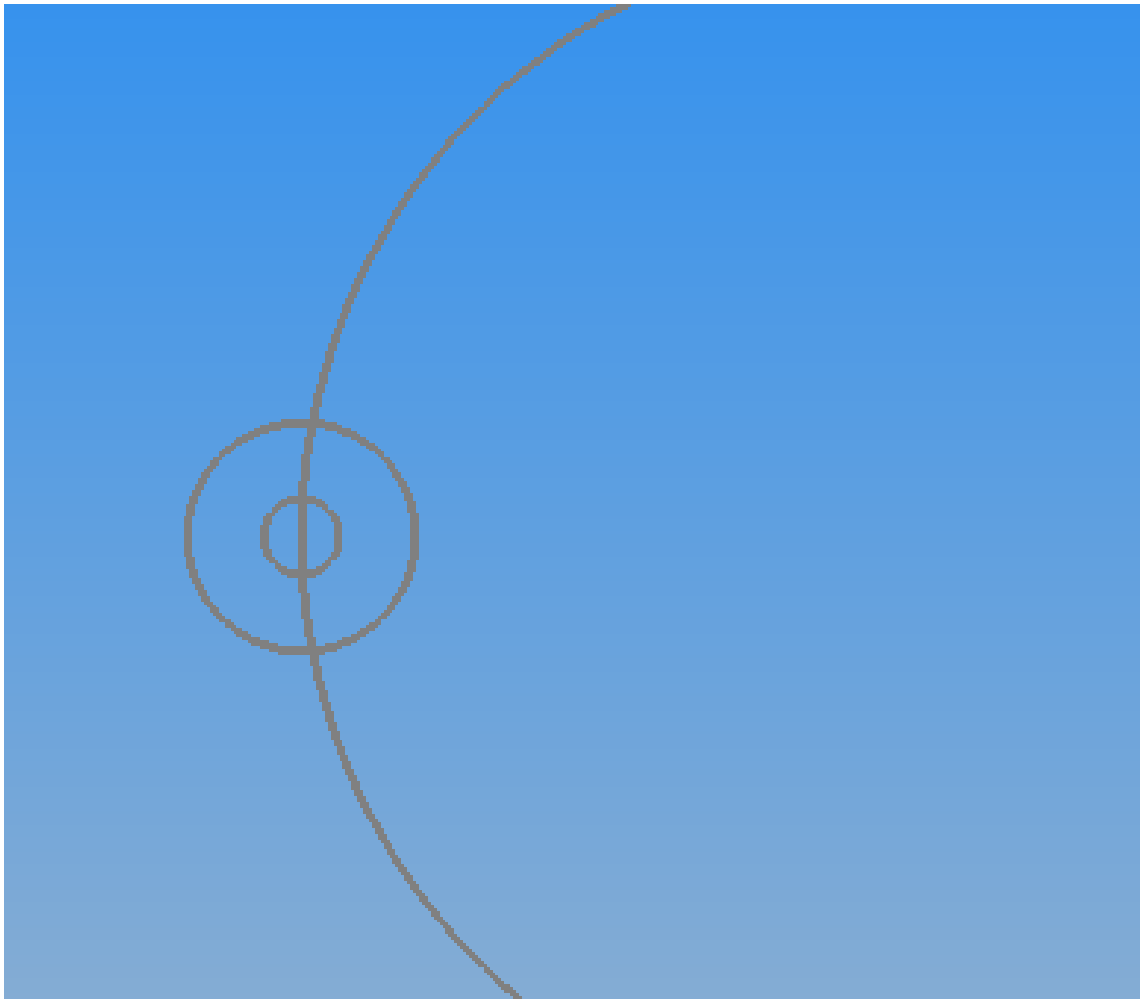


Figure 85 - R12 Circle created

Copy Hole and Lug

Select **EDIT > Move Copy etc. > Copy** 

<LClick> on the two circles (they should turn blue), then **<RClick>** or press **<Esc>**.

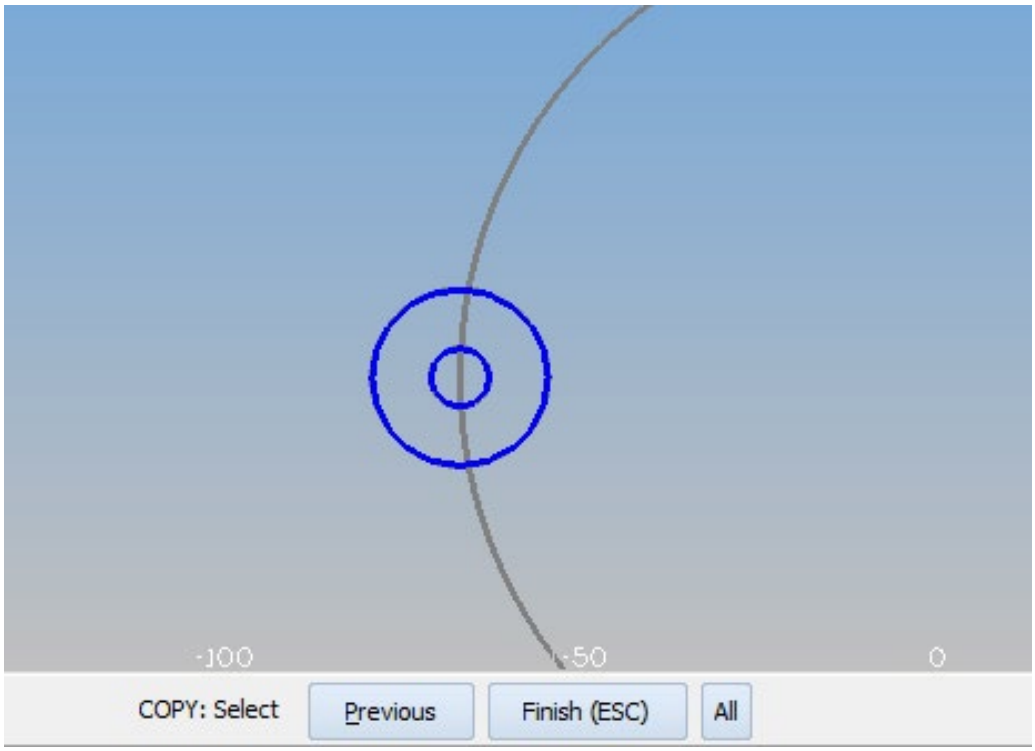


Figure 86 - Select the items to Copy

The system will prompt you to select the base point, using the auto snap functionality, **<LClick>** the centre point of the two selected circles.

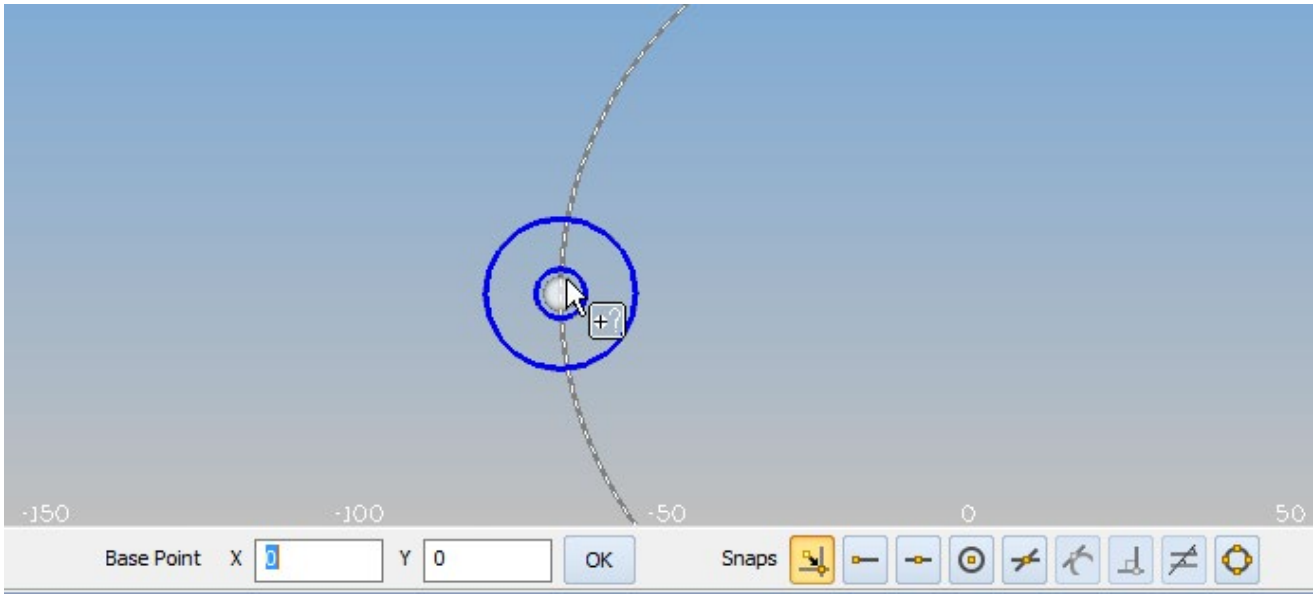


Figure 87 - Auto Snap to the centre of the two circles

The system will prompt you to **Drag to Position or Enter Coordinates**, type **-18.47 <Enter> 61.54**

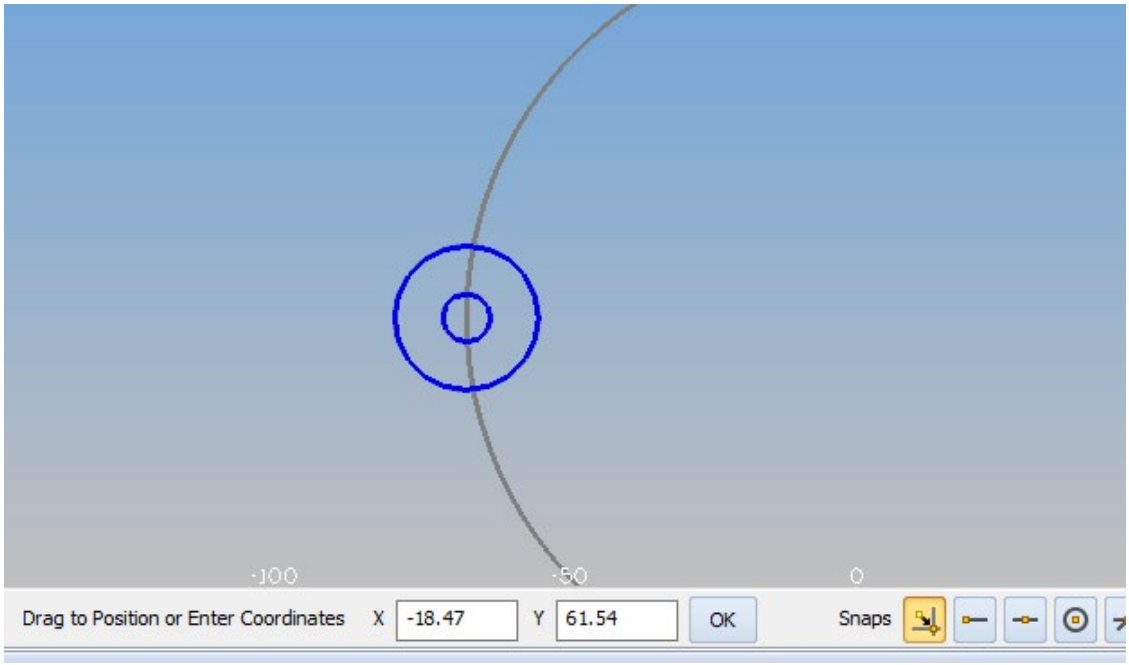


Figure 88 - Enter the first positions

<LClick> [OK] both circles will be copied to the new position.

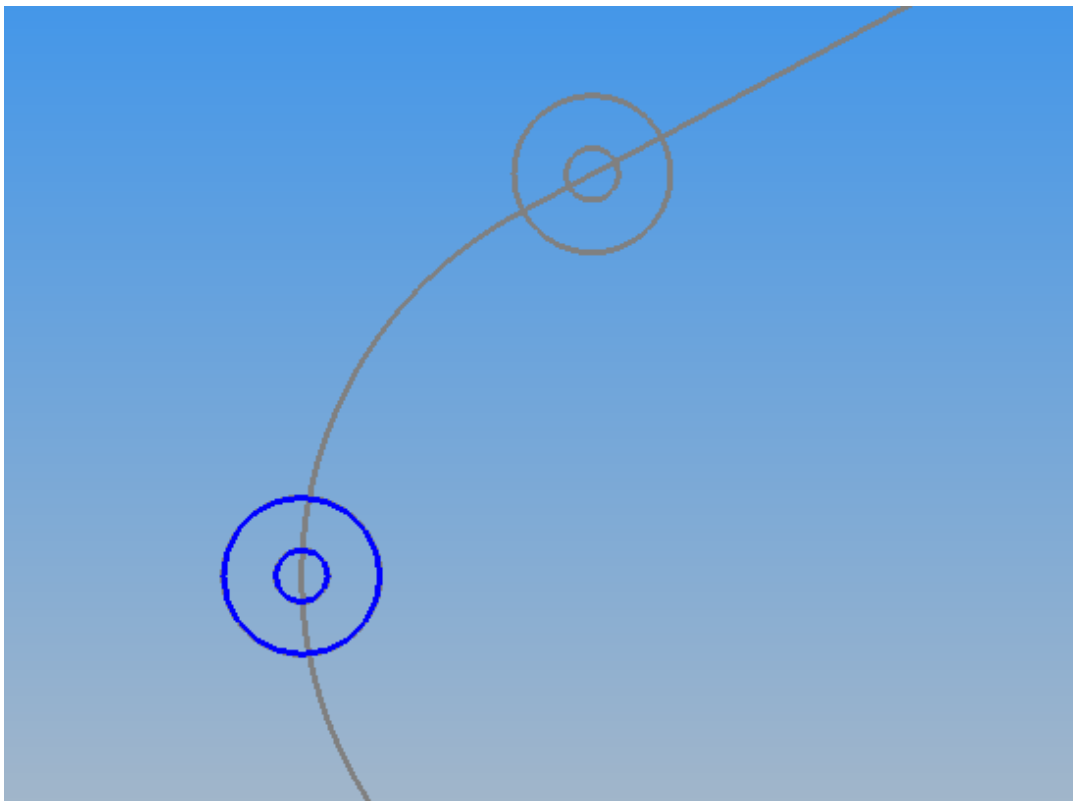


Figure 89 - First copied holes

The system will prompt you to **Drag to Position or Enter Coordinates**, type **68.86 <Enter> 105.5**.

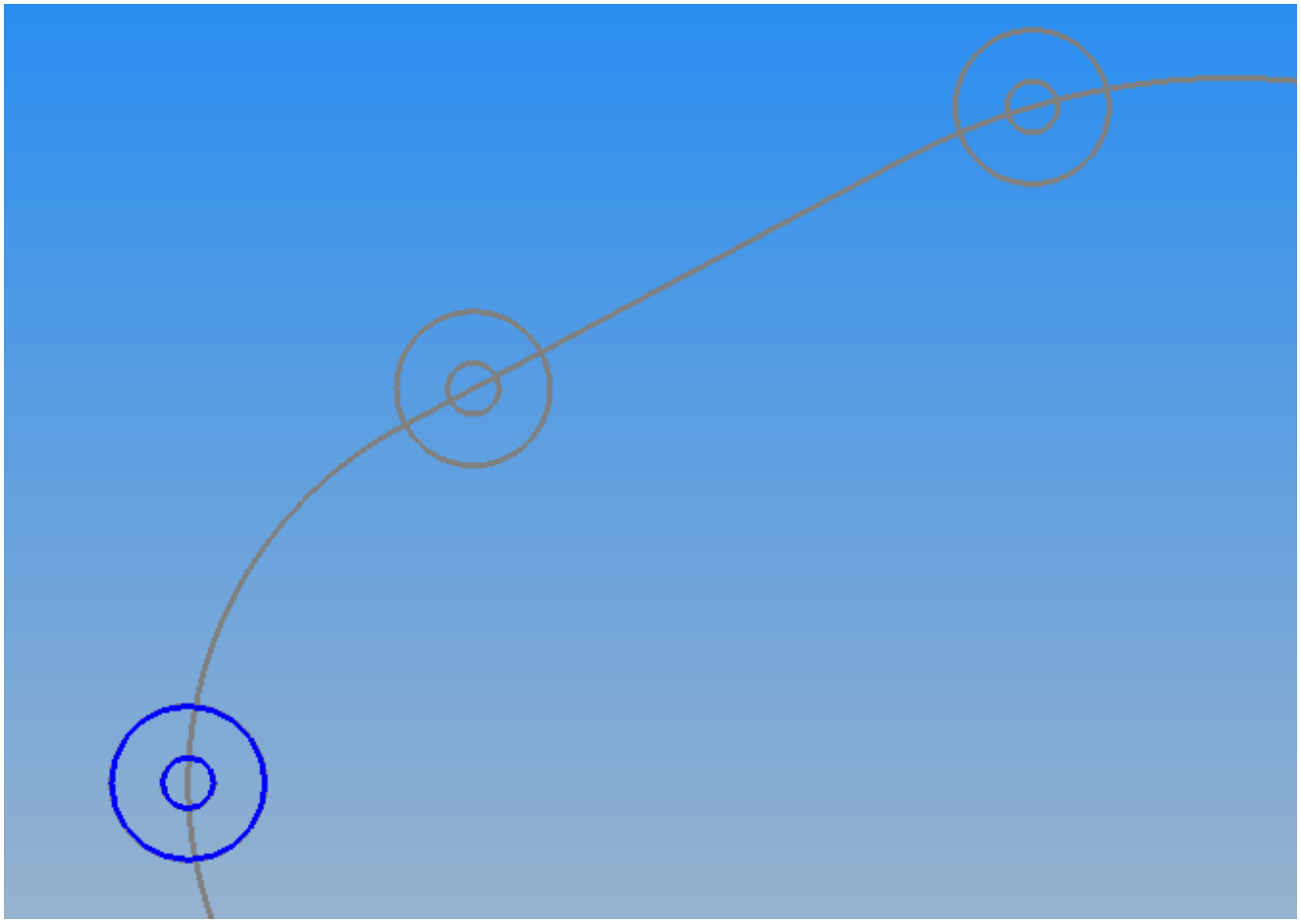


Figure 90 - Second copied holes

<LClick> [OK] both circles will be copied to the new position.

For the third set using the auto snap, **<LClick>** the right most quadrant position on the outer profile, as shown below.

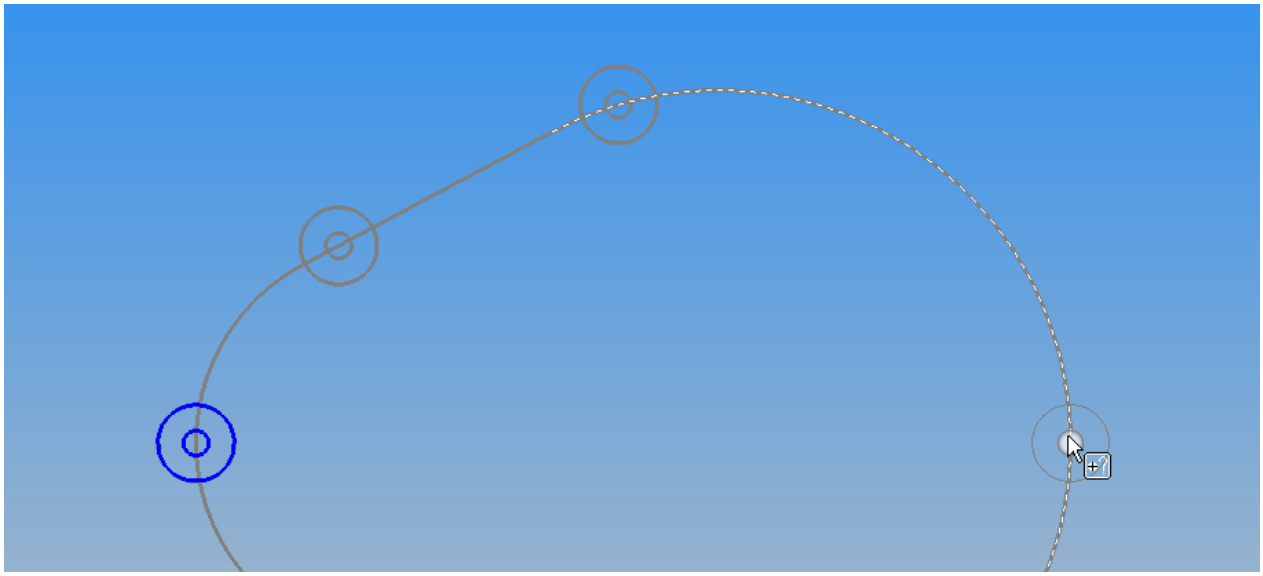


Figure 91 - Location of final Copied circles

<RClick> to finish the placement options. **<RClick>** again to finish the command completely.

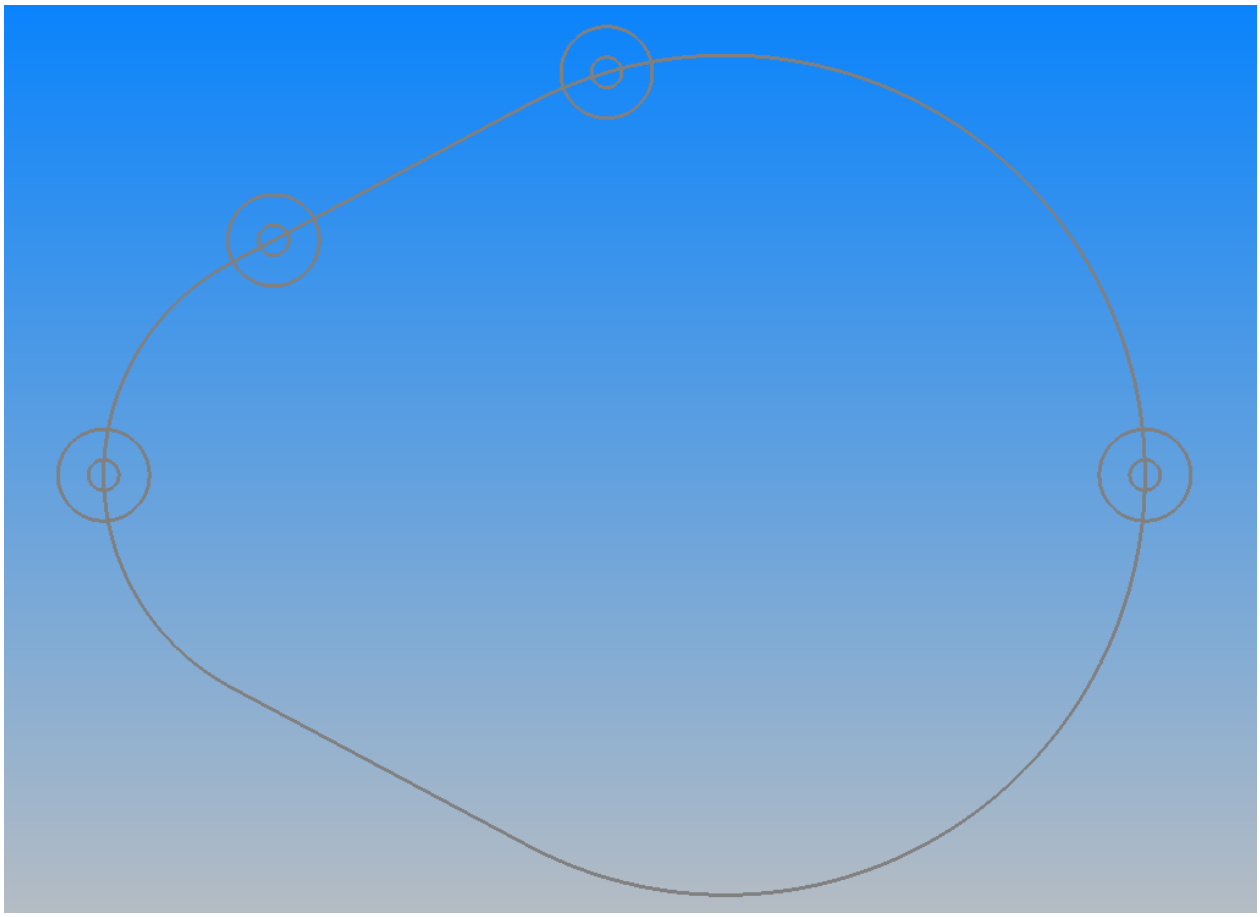


Figure 92 - Copied Circles completed

Rotate Hole and Lug to 35°

Select **EDIT > Move Copy Etc. > Rotate**

<LClick> on the hole and lug circles on the right side to select them.

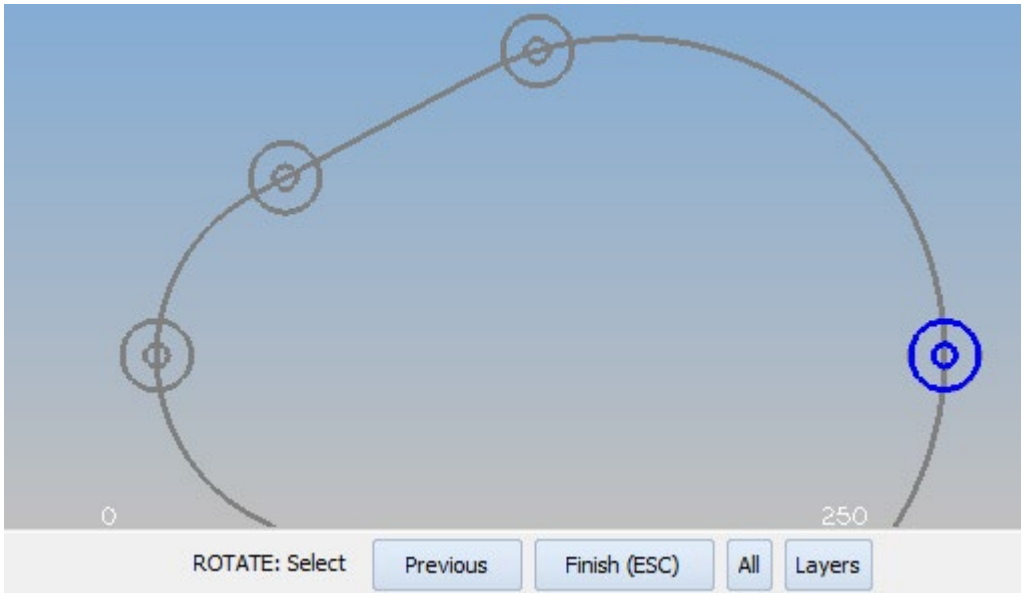


Figure 93 - Circles to rotate

<RClick> or Press **<Esc>** or **<LClick> [Finish (ESC)]**.
The system will prompt you to select the Base Point.

Figure 94 - Location point for rotation

Using the auto snap **<LClick>** the centre point of the Ø30 circle.

The system will prompt you to select or enter the **Rotation Angle (CW -ve).**

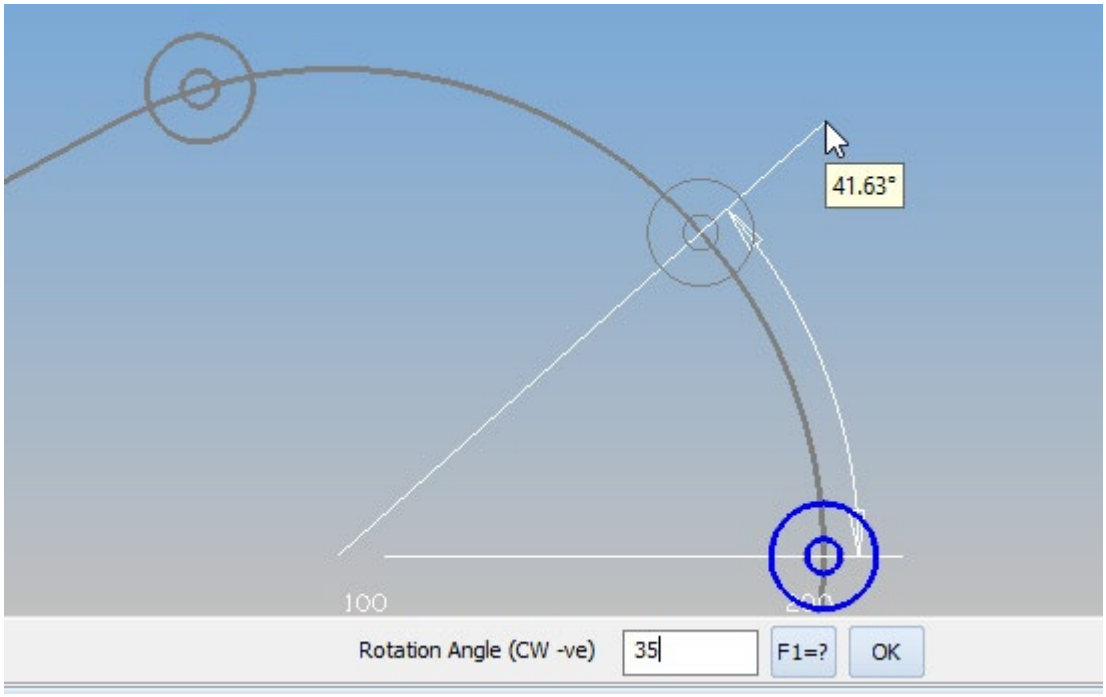


Figure 95 - Angle rotation options

Type **35** into the dialogue box, then **<LClick> [OK]**.
The system will prompt you **How Many Copies (Including Original)**.

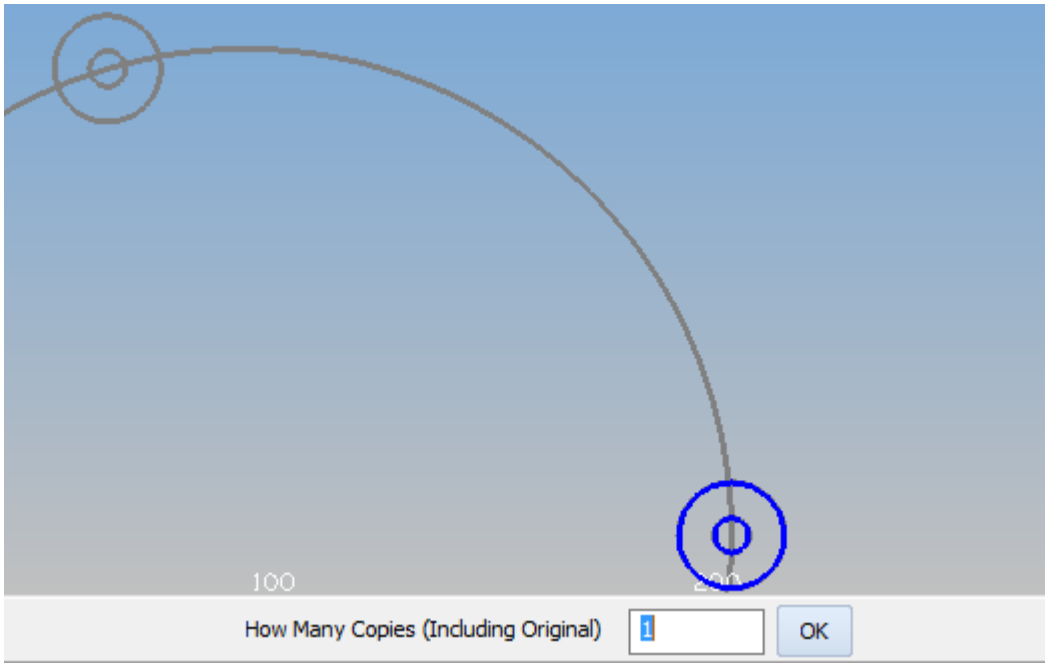


Figure 96 - Number of rotational copies

Type **1**, then **<LClick>[OK]**.
The hole and lug will be rotated to their new position. **<RClick>** to finish the command.

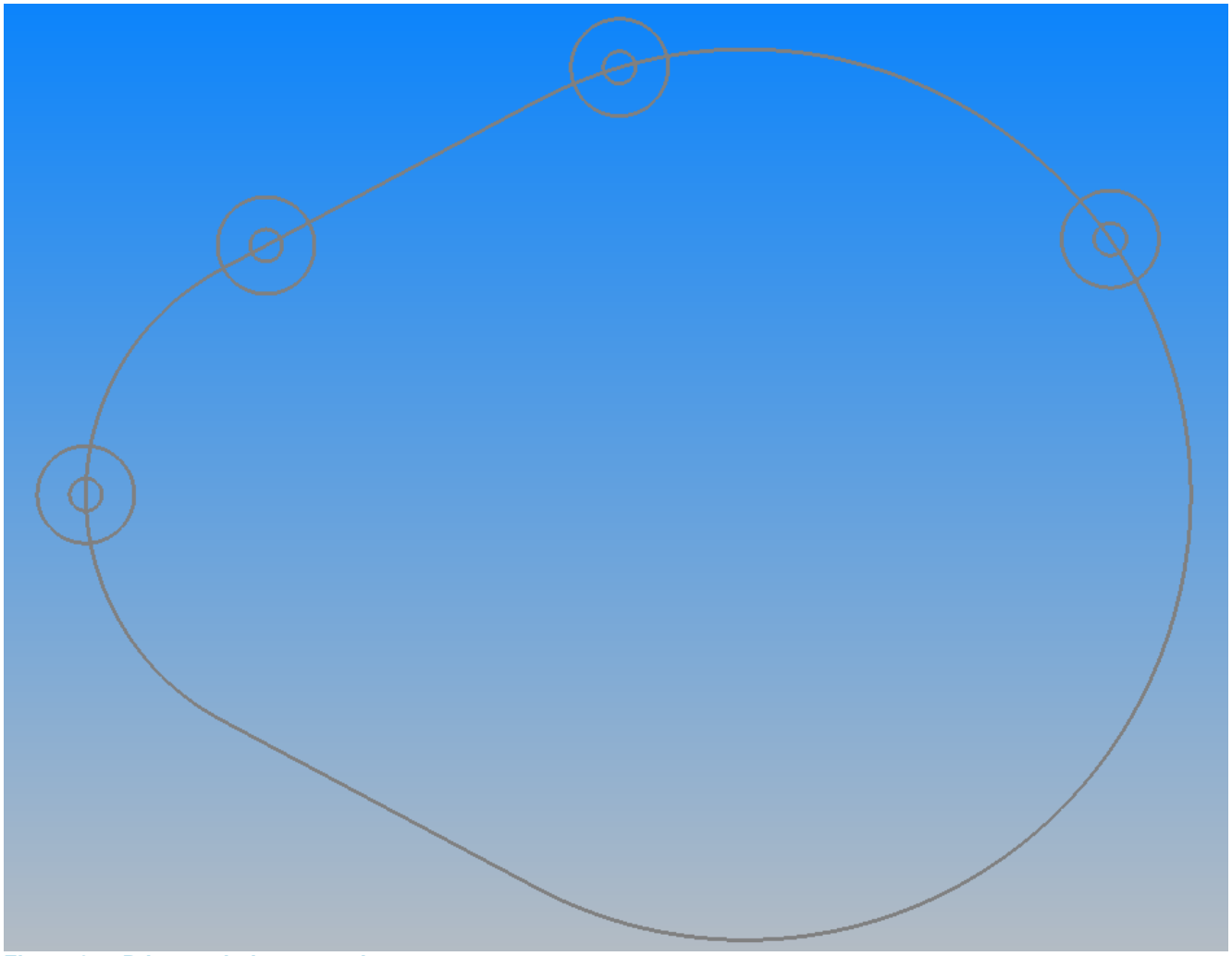


Figure 97 - Primary circles created

Mirror Holes & Lugs

Select **EDIT > Move Copy Etc. > Mirror**

When prompted, **<LClick>** the items shown below for mirroring.

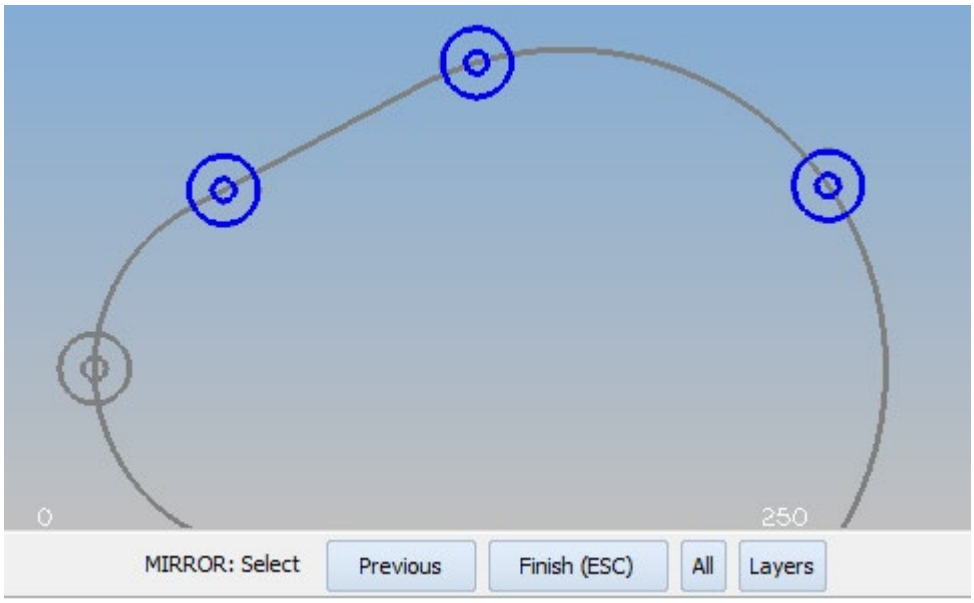


Figure 98 - Select items to Mirror

<RClick> or Press **<Esc>** or **<LClick> [Finish (Esc)]**.
The system will prompt you to select the First Point on Mirror line.

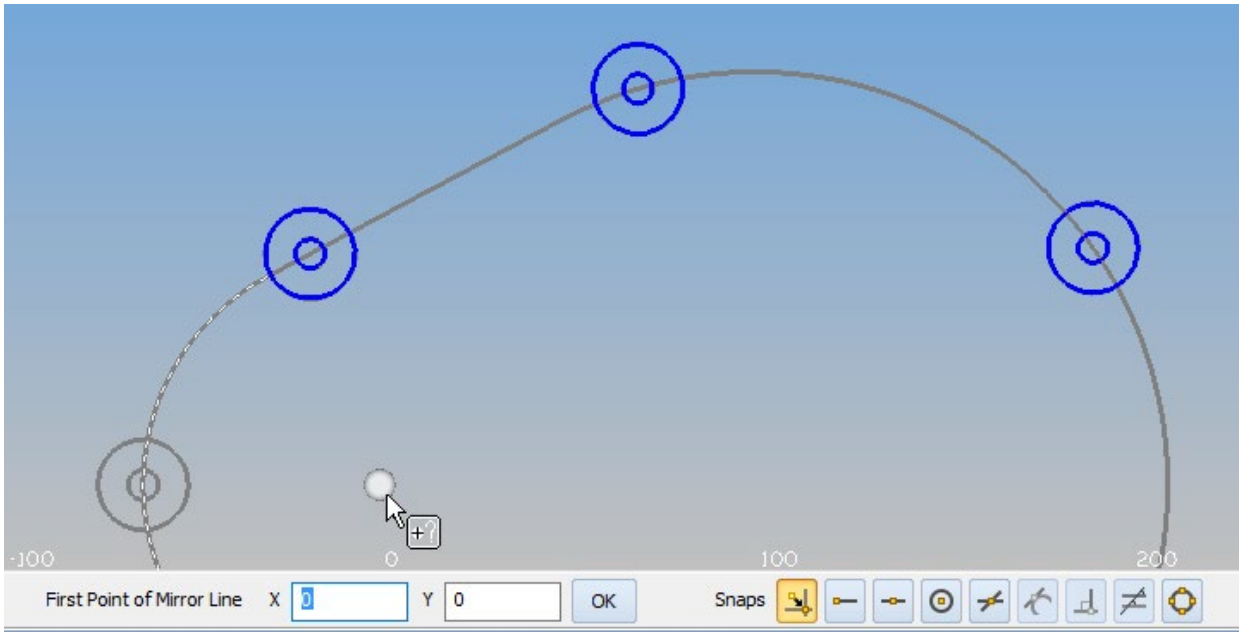


Figure 99 - First Mirror point

<LClick> the location as shown above.

The system will prompt you to select the Second Point on Mirror line.

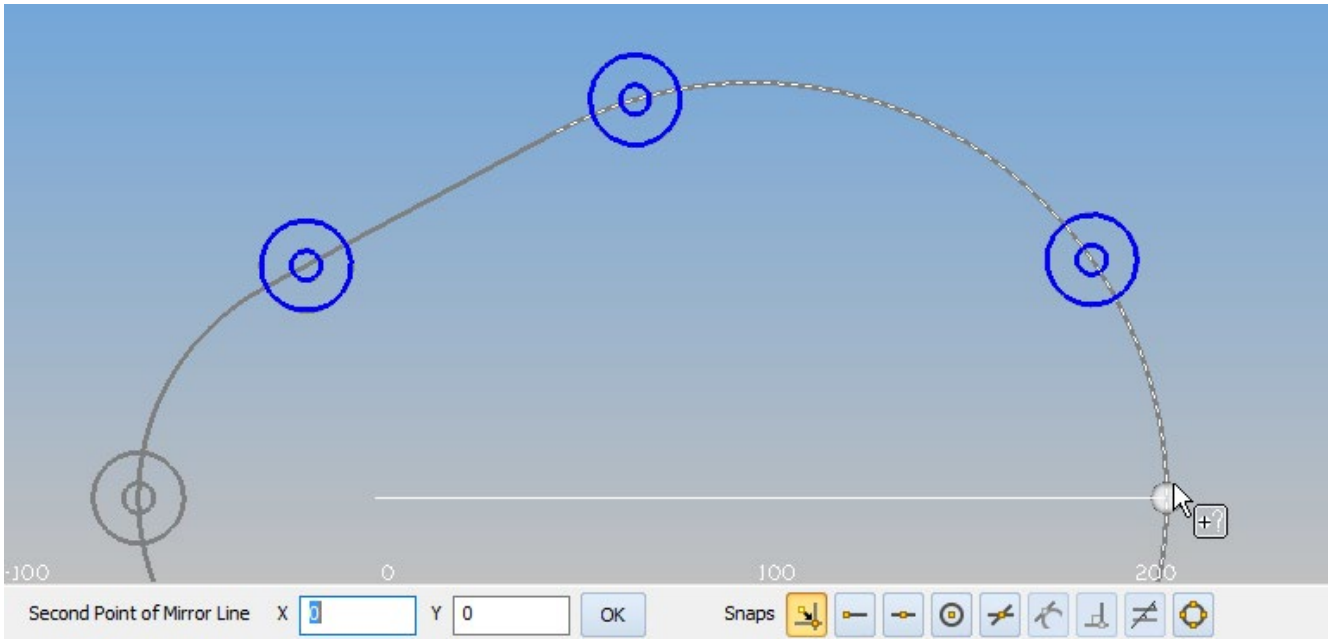


Figure 100 - Second point of Mirror

<LClick> the location as shown above.

> You can select any 2 points that describe the mirror axis. Mirror only requires Vector direction so auto snapping to X0 Y0 for one point and then entering the value of 1 in the X option box will give the same results.

The Mirror options dialogue is displayed as shown:

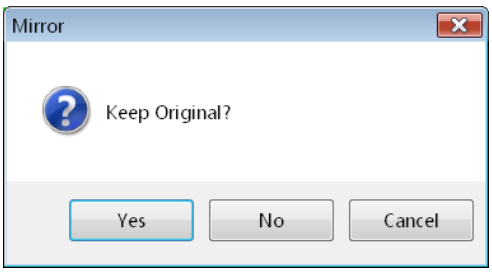


Figure 101 - Keep Original when Mirroring

Select the **[Yes]** option and the holes and lugs will be duplicated and mirrored.
<RClick> or **<Esc>** to finish the command.

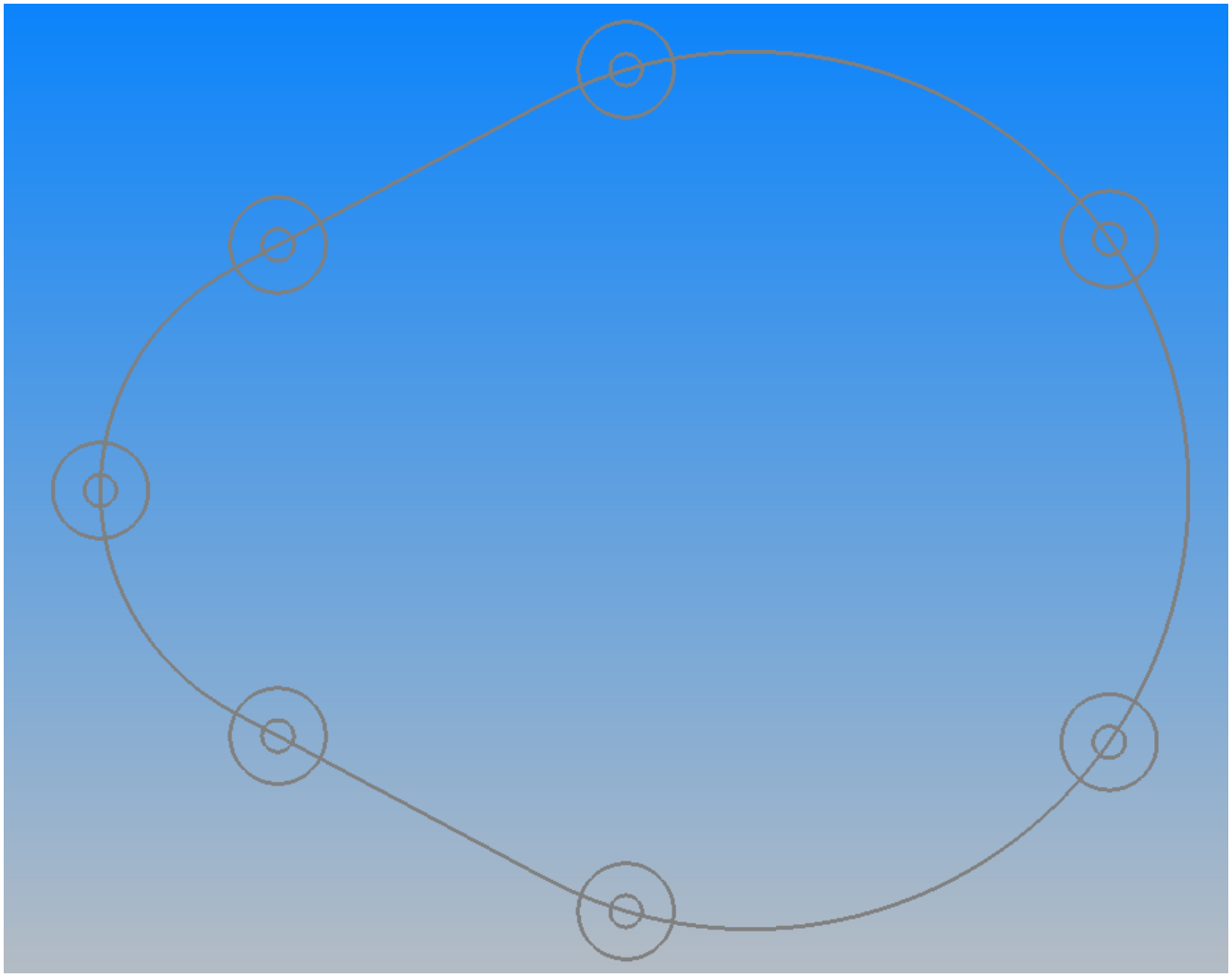


Figure 102 - Completed Mirror command

Trim and Join to create outer Profile

Method 1

Unite

Select **EDIT > Fabricate > Unite** 

For the items to Unite, select the outer profile and the lug circle,

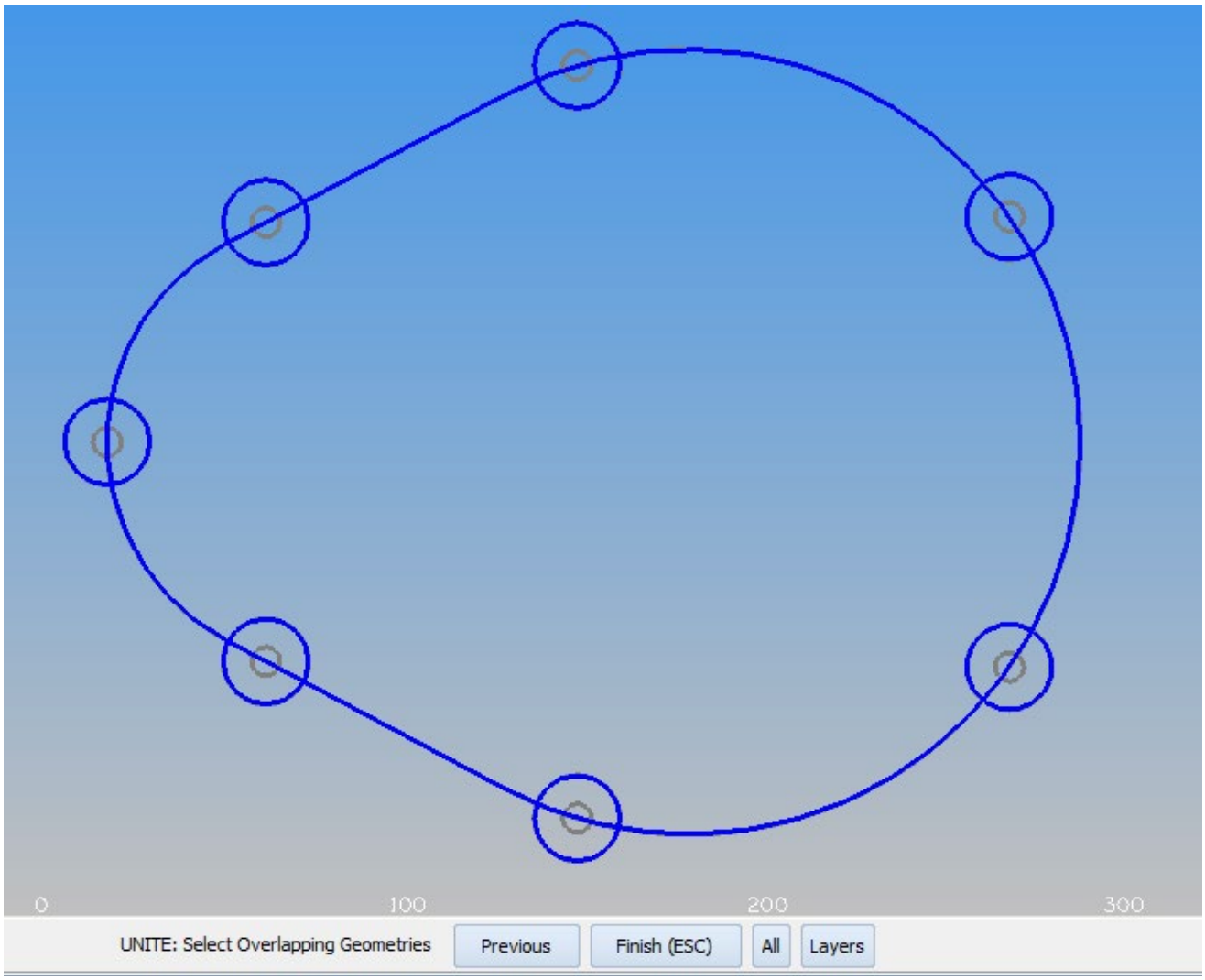


Figure 103 - Unite outer profile

<RClick> or Press **<Esc>** or **<LClick> [Finish (ESC)]**.
The lugs and outer profile will be trimmed and joined to form the closed boundary.

Method 2

Trim and Join

Trim Lugs & Outer Profile

Select **EDIT > Break Join etc. > Trim** 

For the cutting geometries select all the R12 circles and the outer profile.

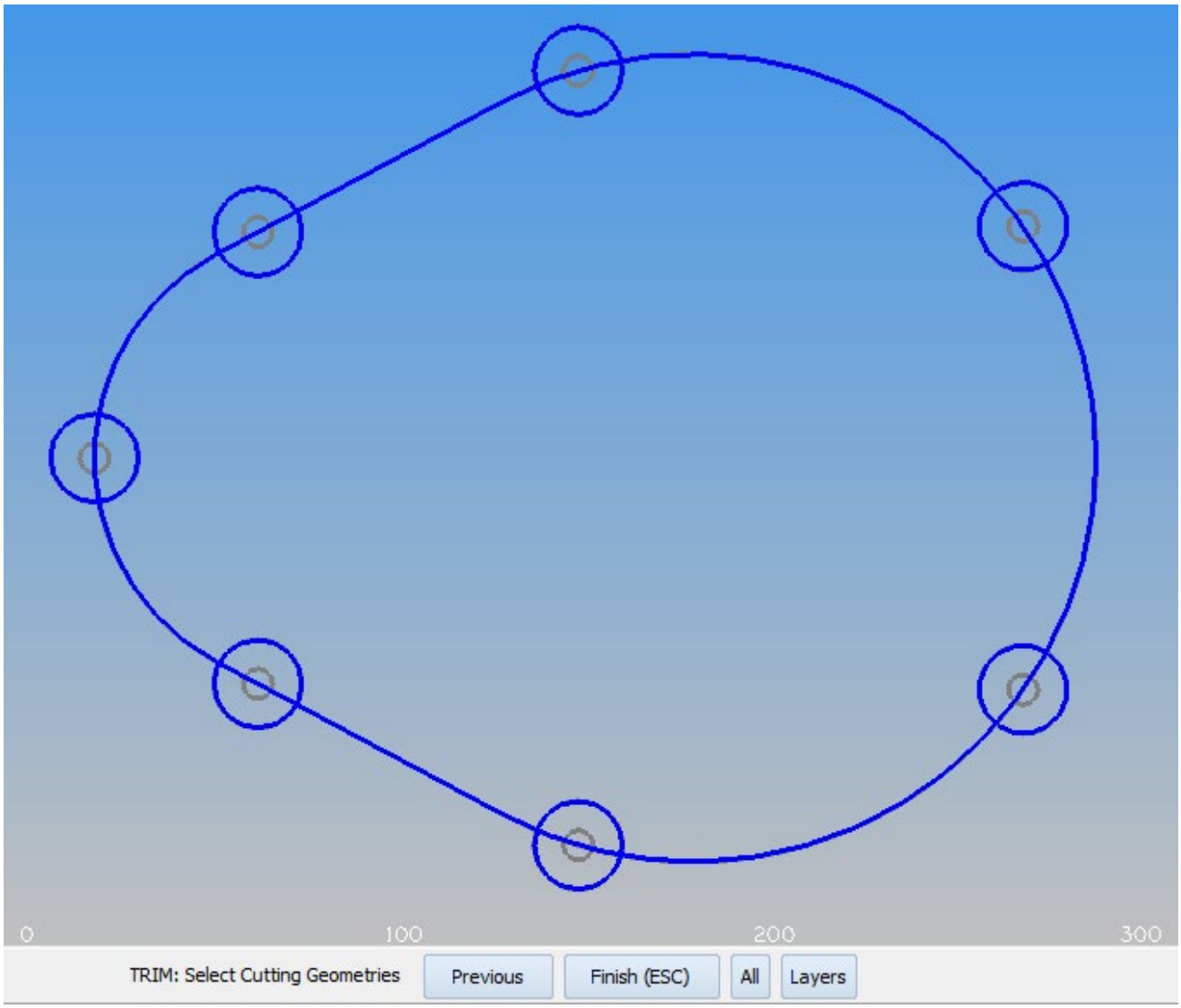


Figure 104 - Items to Trim

<RClick> or Press **<Esc>** or **<LClick> [Finish (Esc)]** to continue.

For the geometries to be Trimmed select all the inner parts of the R12 circles, then select the portions of the outer profile inside the lugs.

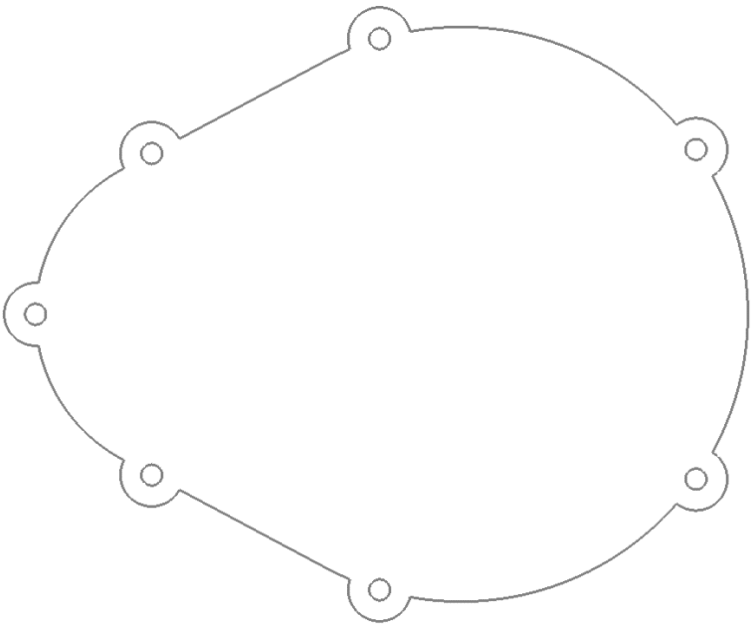


Figure 105 - Unwanted Geometries Trimmed away

Remember that Refresh or **<Ctrl> + <R>** or  can be used at any time.

When the trimming is complete **<RClick>** or Press **<Esc>** to complete the process.

Join outer profile Method 1

Select **EDIT > Break Join Etc. > Join** 

Select the **[Previous]** button on the prompt line, this will select what was blue in the previous command, **<RClick>** or Press **<Esc>** to complete the command.

Select **<Ctrl> + <G>** or **View > Display Option > Ghost Tools**  to display the ghost tools, ensure that there is only one on the outer profile.

Join outer profile Method 2

Select **EDIT > Break Join Etc. > Join** 

Select the **[All]** button on the prompt line, this will select all the visible geometries, **<RClick>** or **<Esc>** to complete the command.

Select **<Ctrl> + <G>** or **View > Display Option > Ghost Tools**  to display the ghost tools, ensure that there is only one on the outer profile.

Selecting other geometries that are not connected with the geometries being joined has no effect.

Fillet 5mm

All the sharp corners in the outer profile can be filleted in one go.

Select **EDIT > Break Join etc. > Fillet**

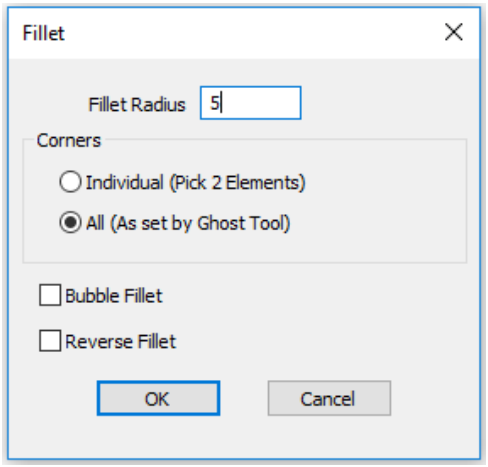


Figure 106 - Filleting Options

Set the options as shown then **[OK]**.
Select the outer profile then **<RClick>** or Press **<Esc>** and the fillet radii will be created around the outer profiles' sharp corners.

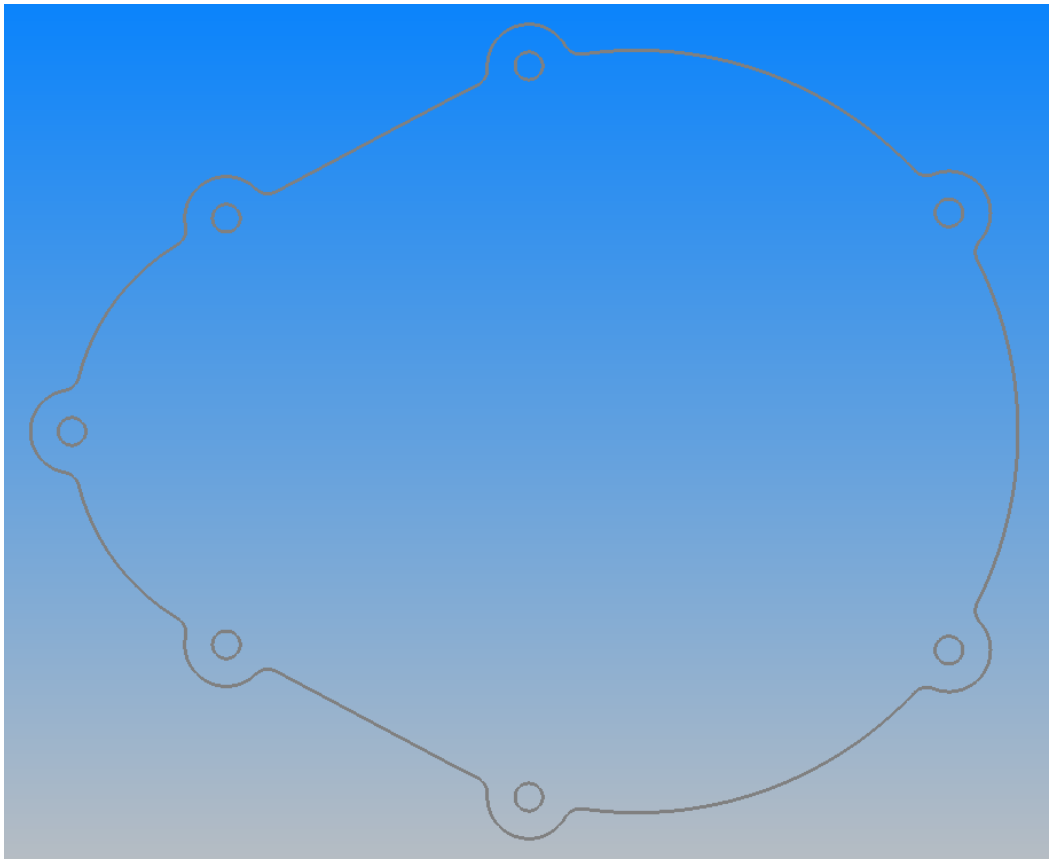


Figure 107 - 5mm Fillet radii applied

8mm Holes Layer

To finally tidy up the drawing layers so far, we will create a **User Layer** for the 8mm circles.

Create and activate the User Layer

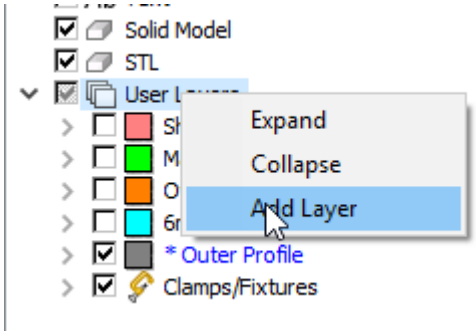


Figure 108 -<RClick> options for a new User Layer

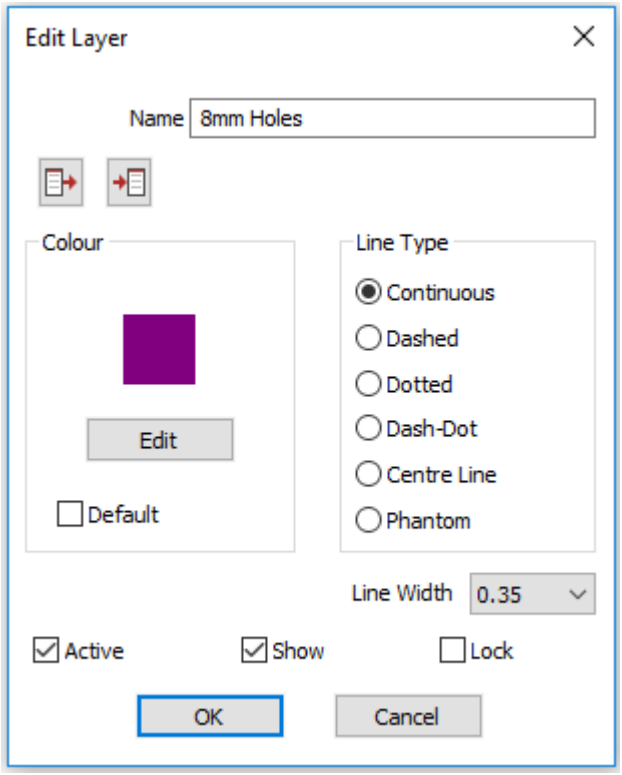


Figure 109 - User Layer 8mm Holes

<LClick> [OK] to continue.

Alter the current Layer settings for the 8mm circles.

While it is possible to use the Drag and Drop method we have seen earlier, there are a number of geometries that need adjusting so using the following method is easier.

Using **EDIT > Change** 

Set the options as follows.

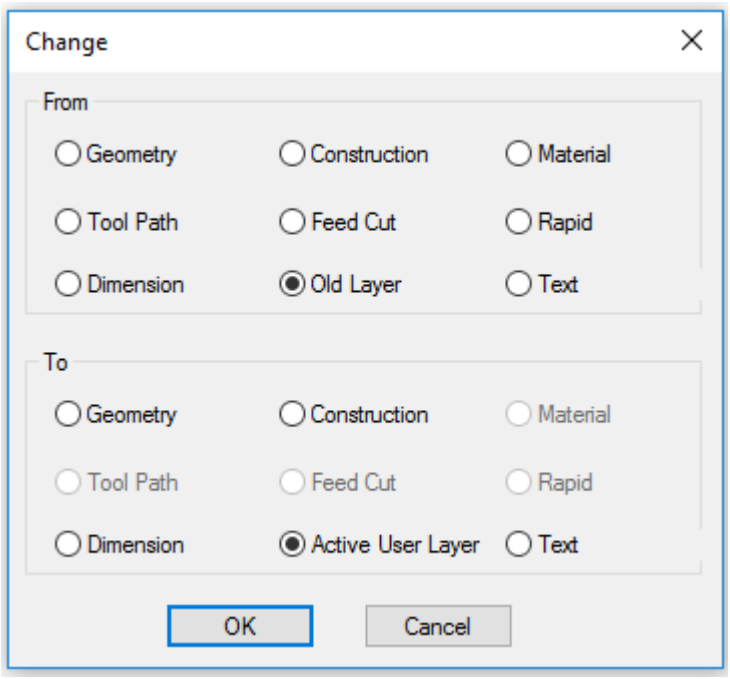


Figure 110 - Changing layer status

These settings maintain all sub layer settings within a **User Layer** structure.

Construction will remain on Construction, Geometry on Geometry etc, but all of these elements will be relocated from the present location within the Layer tree structure, to the current active User Layer.

<LClick> [OK] to continue.

As we have hidden all the other User Layers, when prompted for the items to change, use the **[All]** option at the bottom of the drawing area,

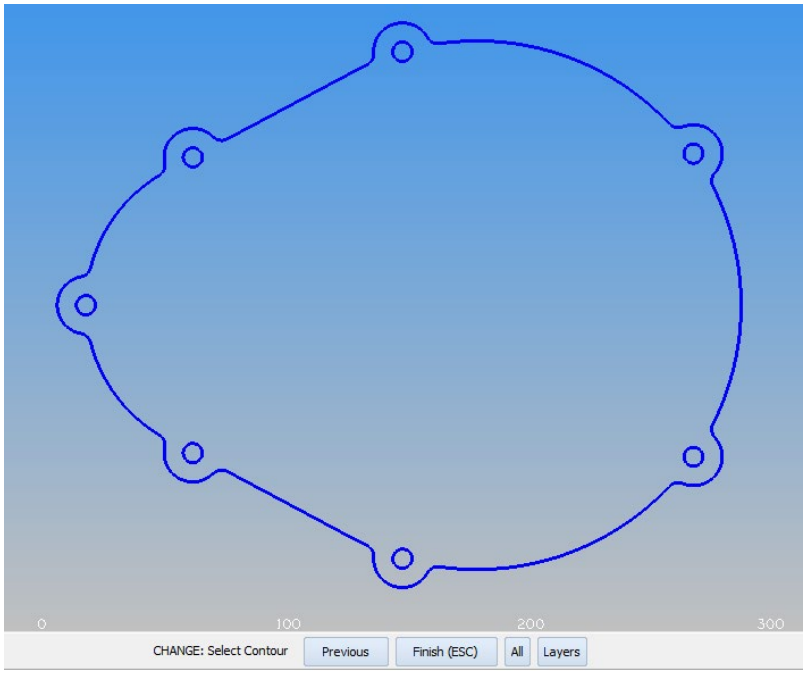


Figure 111 - Use All to make selection easier

Then **<LClick>** the **Outer Profile**, to deselect it.

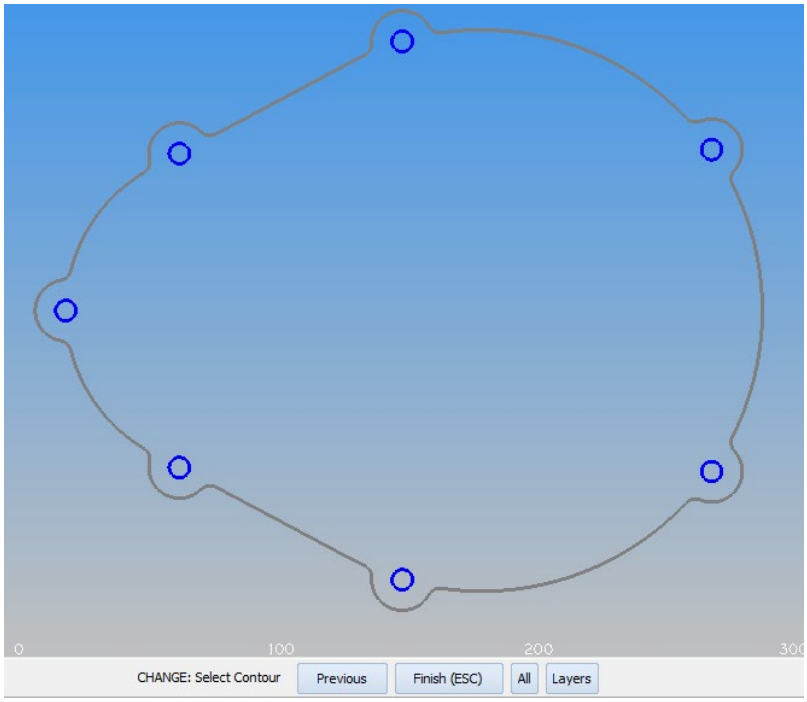


Figure 112 - Outer Profile deselected

<LClick> [Finish (Esc)] to complete to change process.

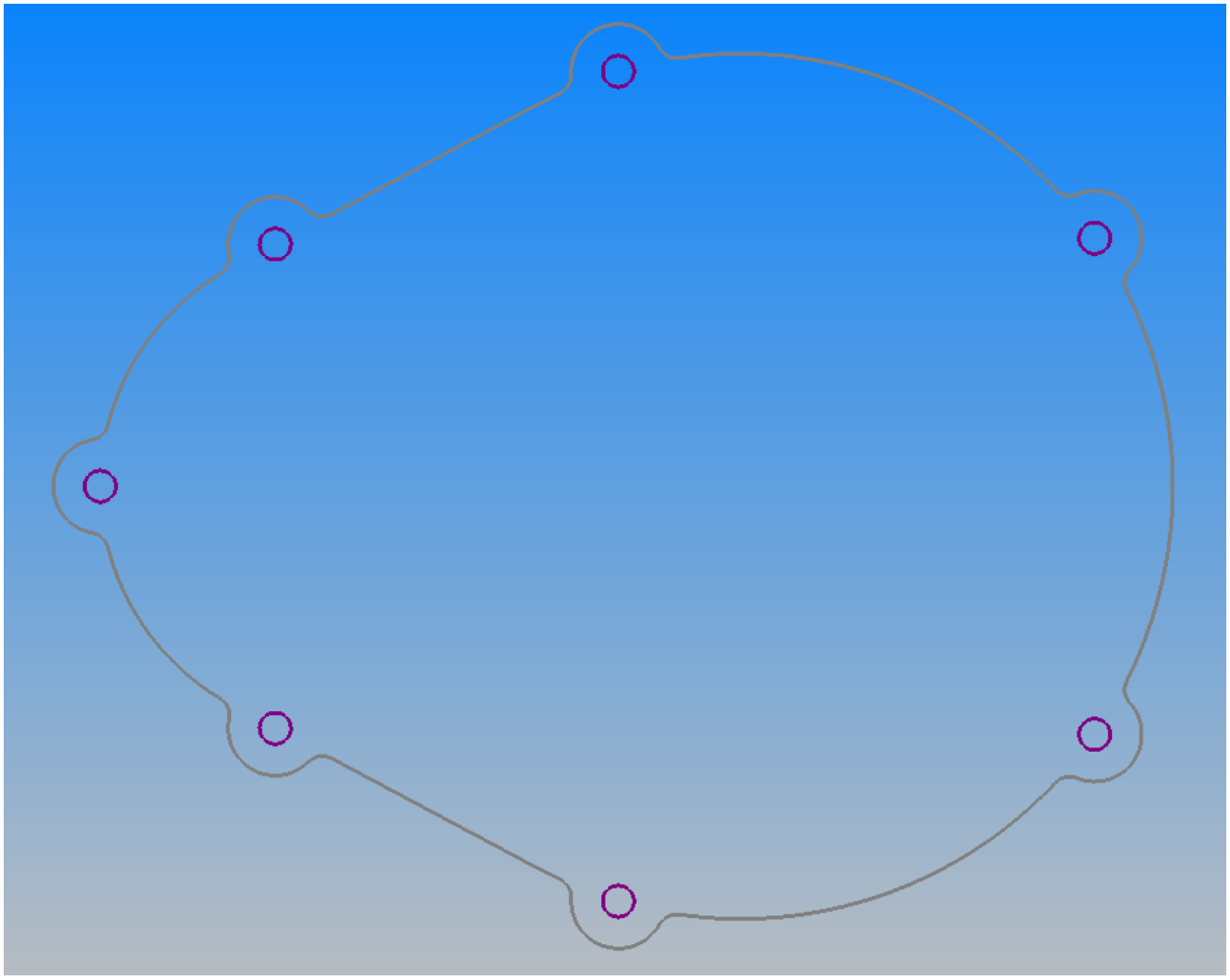


Figure 113 - Completed Change command

Draw Rapids

Select **VIEW > Display Options > Draw Geometry Rapids**

The **Draw Rapids** command shows the connections between the geometry start points in the order they were created.

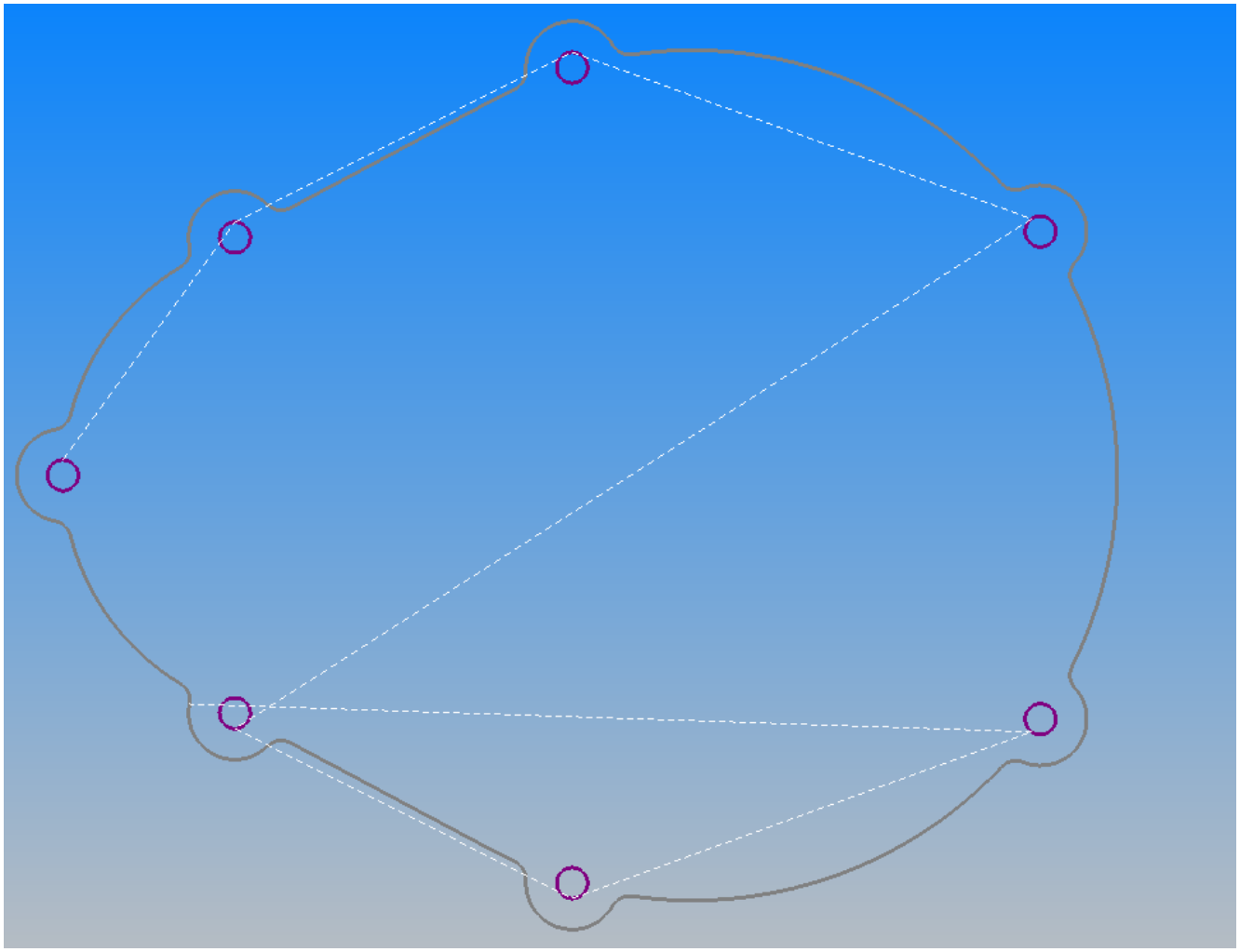


Figure 114 - Geometry Rapids showing the link between each geometry Start Point

You can see that the order of the outer profile holes passes back across the part to the bottom set of holes; this is because the top holes were mirrored to create the bottom ones.

This is the order in which the holes would be drilled.

Whilst it is possible to reorder a series of tool paths, it is best practice to set the order of the geometries especially if the holes are to be machined more than once. i.e. Centre Drill, Drill, Tap and Chamfer.

Order Geometries.

Select **EDIT > Start Order. > Order**

This command allows you to reorder geometries or tool paths.

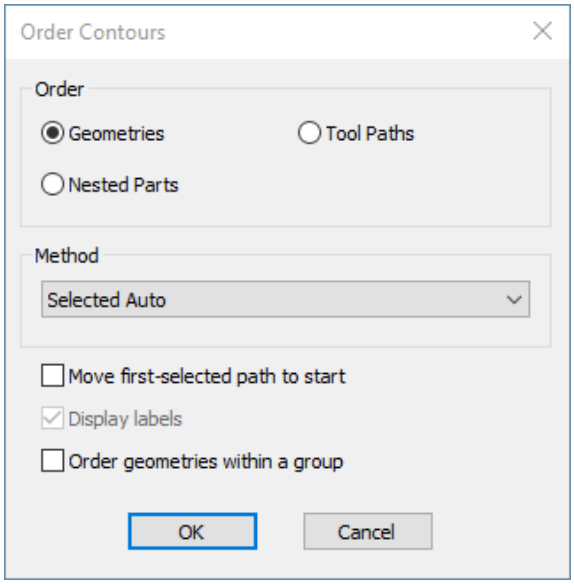


Figure 115 - Ordering Dialogue

Set the options as shown then select **[OK]**.
The system prompts you to select the geometries **<LClick>** on each of the 8mm circles, when they are all blue **<RClick>** or Press **<Esc>**.

Figure 116 - Items to re-order

The order in which the geometries are selected does not affect this method, however, using the Manual option it would.

The system requires you to select the geometry that will be the first in the pattern **<LClick>** on the left most Ø8 mm circle.

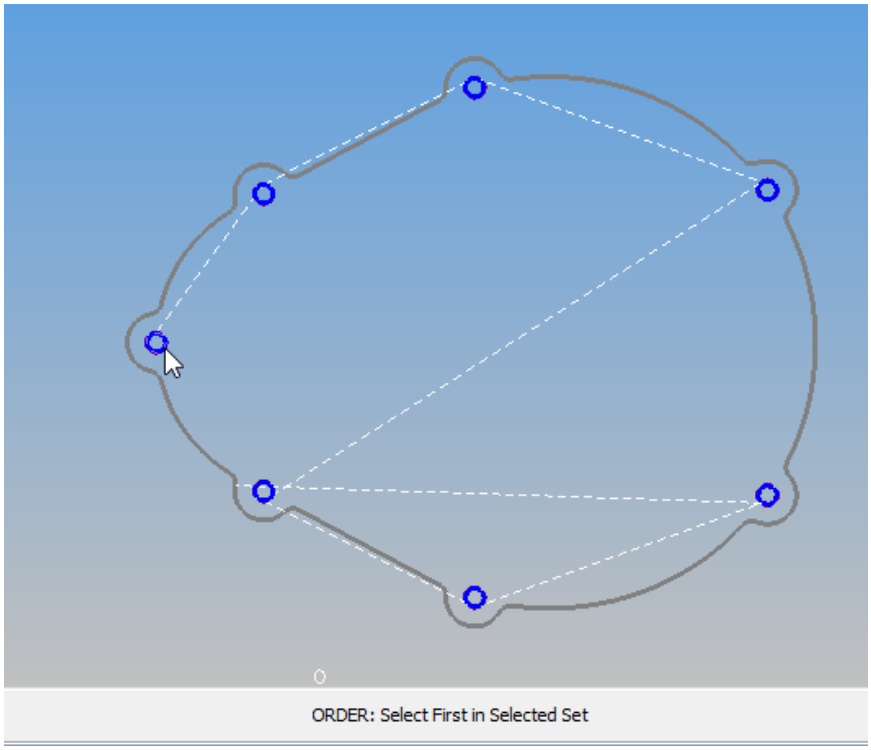


Figure 117 - Select the First in the set

The geometries will be reordered.

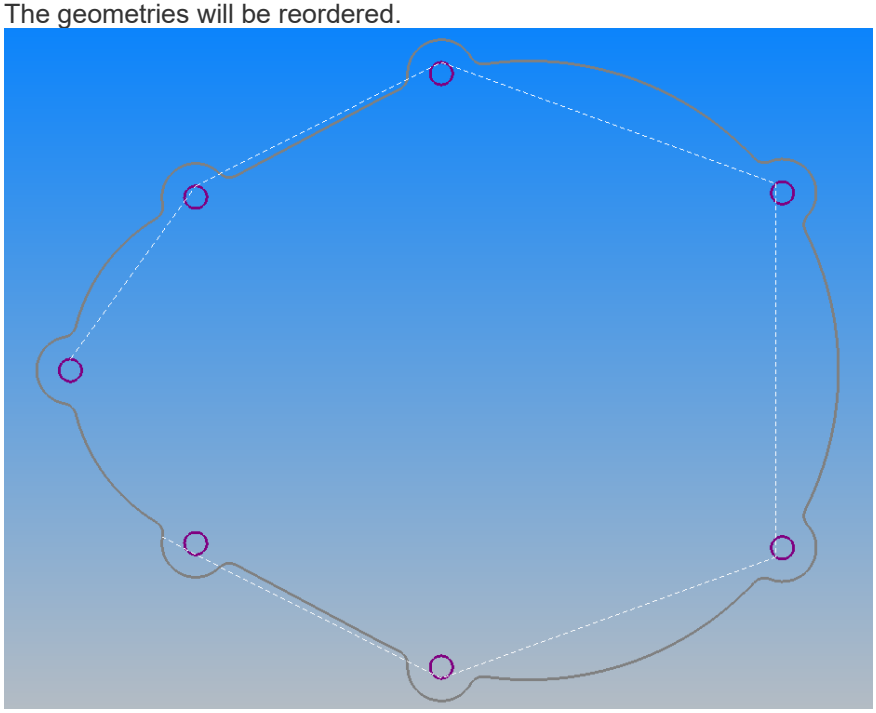


Figure 118 - Circles are re-ordered

Create profiles 4 & 5

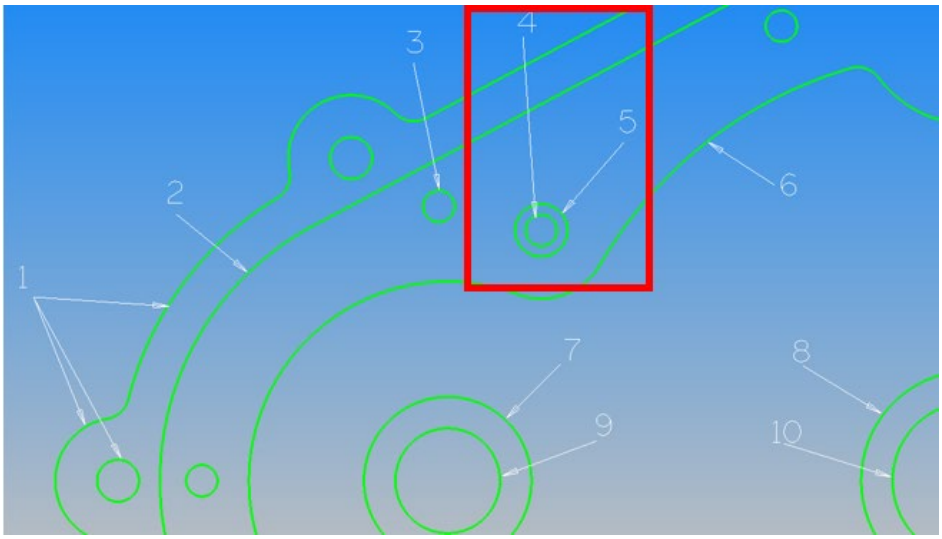


Figure 119 - Profile 4 & 5

These next four geometries are part of the 6mm circle set.
From the Project Manager page, turn on the **Main Pocket** user layer and the **6mm Holes**.
<RClick> on the **6mm Holes** User Layer and make it **Active**.

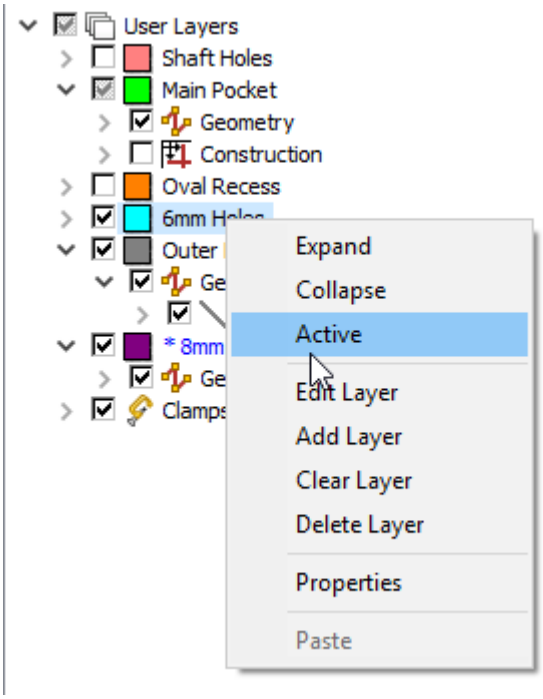


Figure 120 - Activate the 6mm circles layer

Ø6mm and Ø10mm holes at fillet centres

Select **GEOMETRY > Circle > Centre and Diameter** .

Enter 6 for the diameter then using auto snap or **<F8>** create the circles at the R13 Fillet centre.
Enter 6 for the diameter then using auto snap or **<F8>** create the circles at the opposite R13 Fillet centre.

Enter 10 for the diameter then using auto snap or **<F8>** create the circles at the R13 Fillet centre.
Enter 10 for the diameter then using auto snap or **<F8>** create the circles at the opposite R13 Fillet centre.

> To keep adding circles of the same size, left clicking in the drawing area has the same action as pressing Enter to accept the current radius or diameter value, or the same as **<LClick>** on the **[OK]** option.

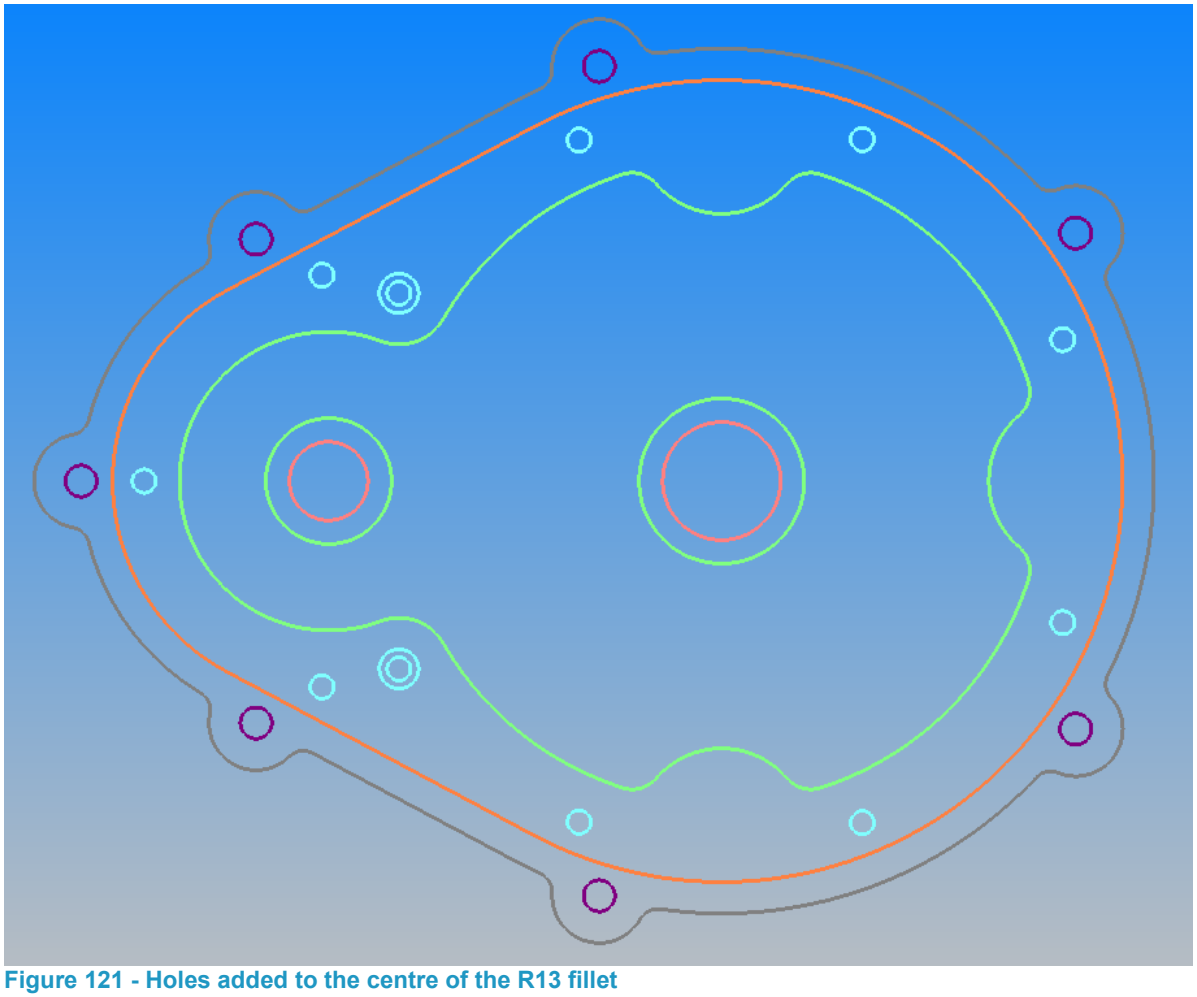


Figure 121 - Holes added to the centre of the R13 fillet

<RClick> on the **6mm Holes** user layer and select the Active option once more to de-activate the layer.

Profile 11, Text

Create the Base Line

For the Base line create an arc using **ARC > Start + Second + End Points.**

> As the Text is printed from left to right, the direction of the base line needs to be also drawn left to right for the text to be positioned above the base line or right to left if the text is to be positioned below the base line

Select **GEOMETRY > Construction** to turn the construction option on.

Select **GEOMETRY > Arc > Start + Second + End Points**
Select the points as shown below.

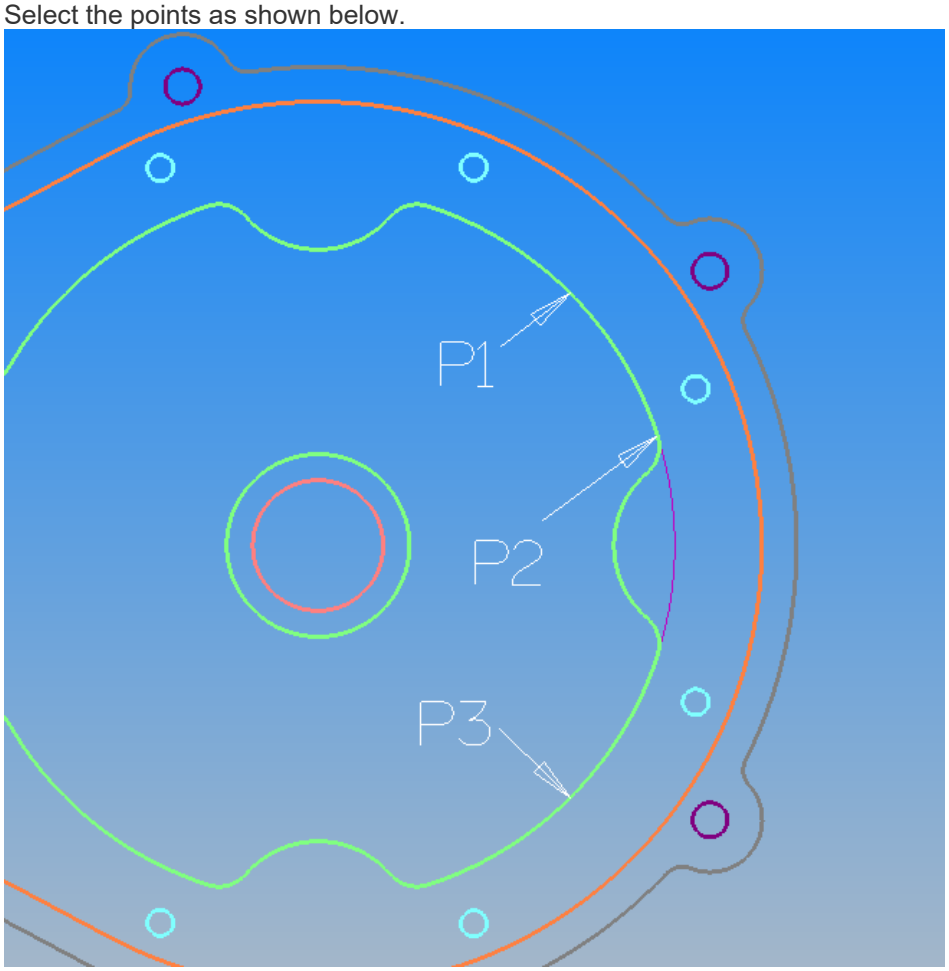


Figure 122 - Choice options for the 3 point Arc

Ensuring that P1 and P3 are the corresponding positions above and below the centre line of the part, so that the midpoint of the created arc is midway across the right hand indent radius feature close to P2.

<RClick> to complete the arc command.

Create the Text

Select **GEOMETRY > Text** Ab
Set the options as shown in the Text dialogue.

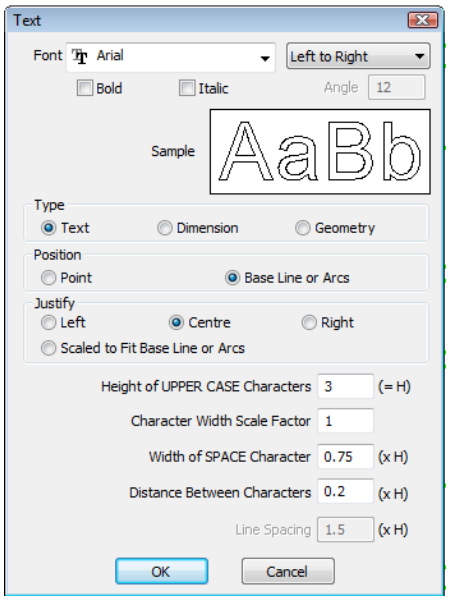


Figure 123 - Text creation Dialogue options

<LClick> [OK] to continue and then **<LClick>** the previously created arc.

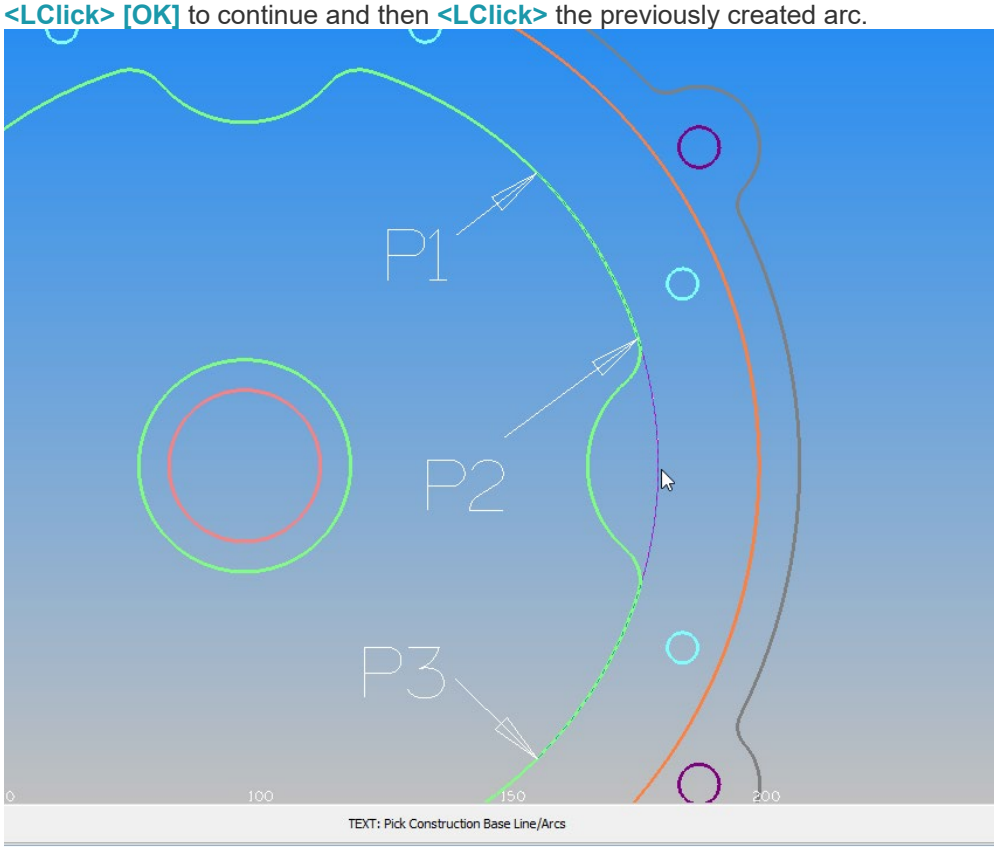


Figure 124 - Select the arc to place the Text on

Enter the text to be engraved “**Part No. 27854**”
Turn off the Construction layer to hide the reference arc.

Figure 125 - Text positioned correctly

Just for clarification, if the construction Arc had been drawn in the opposite direction, selecting the lower point first and working in a CCW direction, the text would be placed as below.

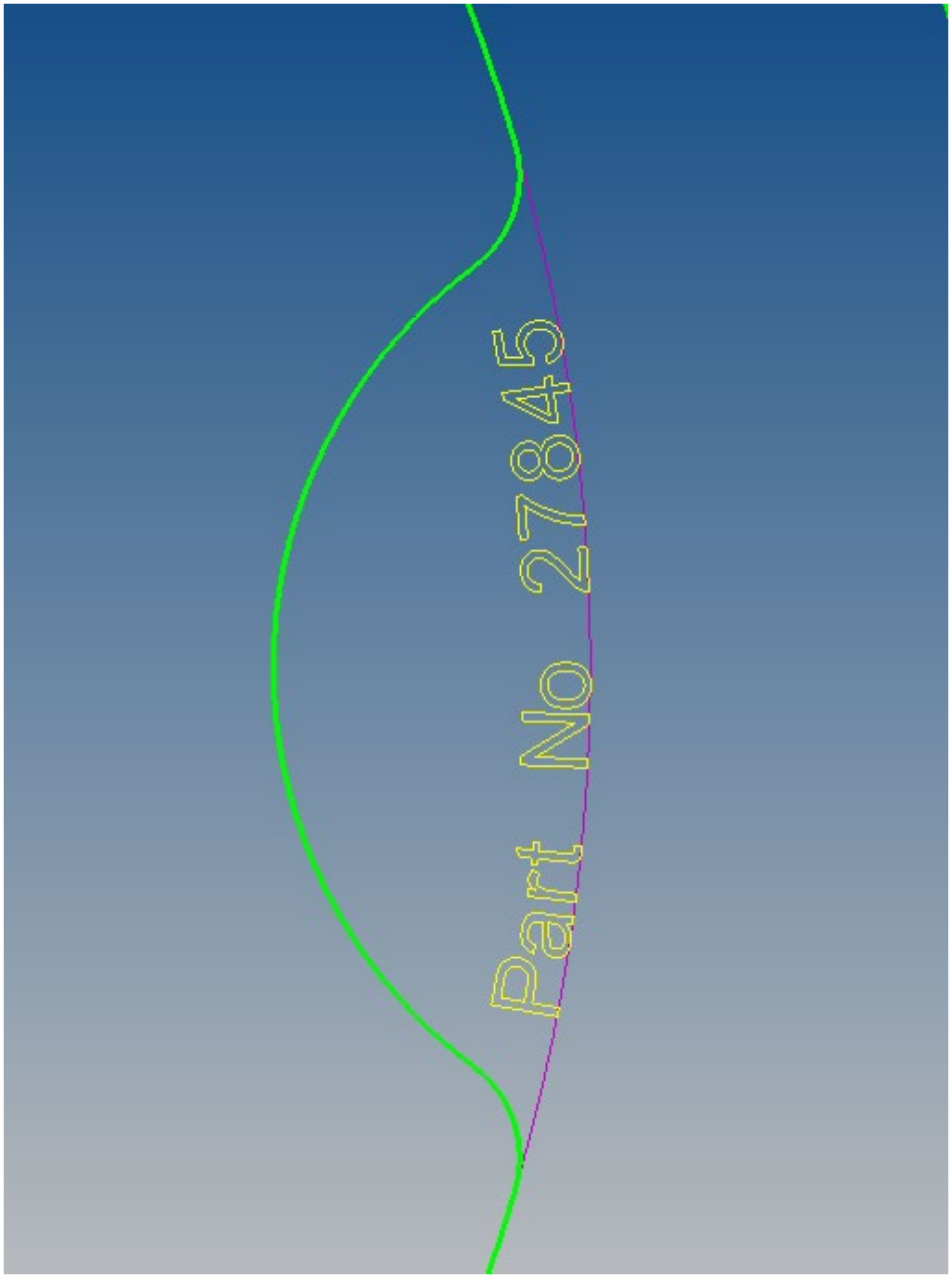


Figure 126 - Text positioned incorrectly

Take care when creating base line items for Text applications.

Position the Geometry for Machining

Create the material

Please note that the use of Materials by either of the following methods is not available in the **Essential** level of the software.

Use the **3D > Auto Set Material**  Macro.
(Unavailable in the Essential level of the software.)

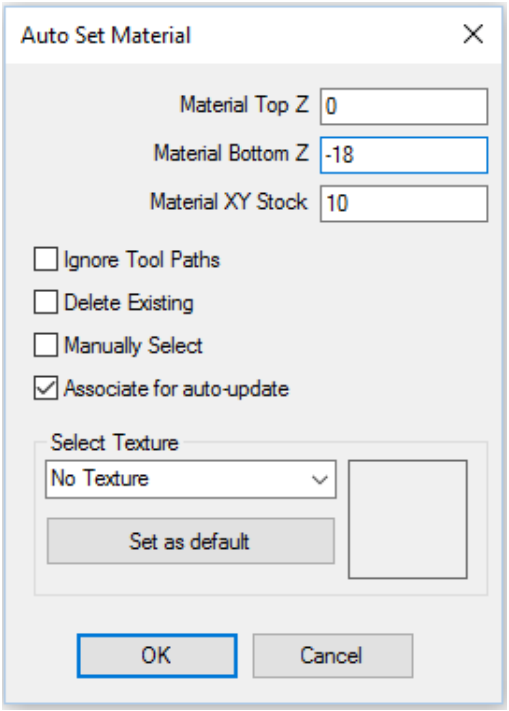


Figure 127 - Auto Set Material options Dialogue

This can be switched on via the **Add-Ins/Macros > Add Ins** 

In the **Add-Ins manager** dialogue on the left hand listing, tick the **☑ Create Default Materia**l option to make this command active in the 3D tab.

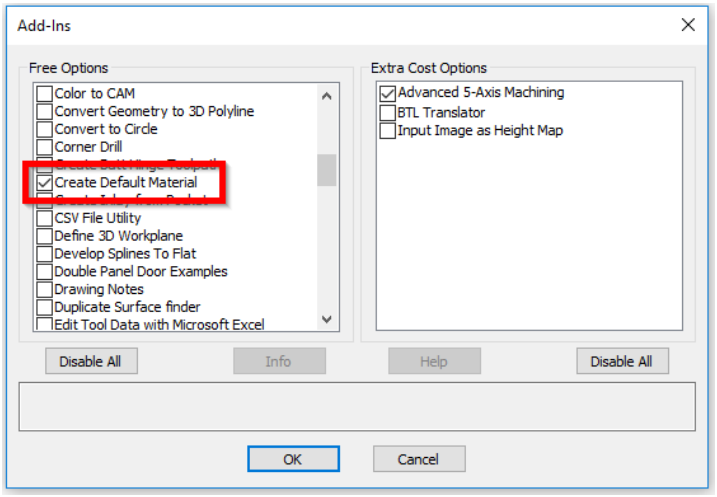


Figure 128 - Add-In manager dialogue

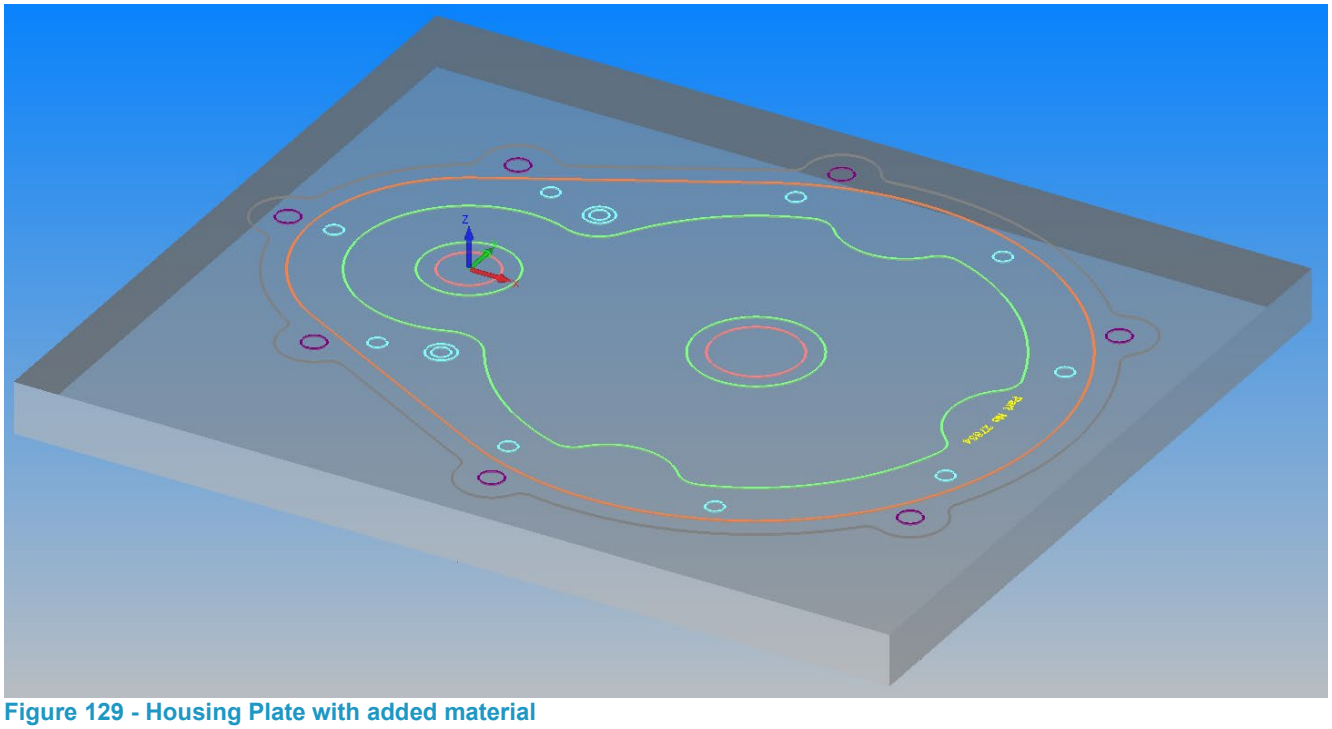


Figure 129 - Housing Plate with added material

One thing to note is that the XY Stock option is applied to all four sides of the Auto Set Material. If you needed differing allowances, then the following method or similar would need to be used.

Manually create the material geometry and define the material

Select **GEOMETRY > Special Geometries > Enclosing Rectangle** 

Select the outer profile then **<RClick>** and the enclosing rectangle is drawn.

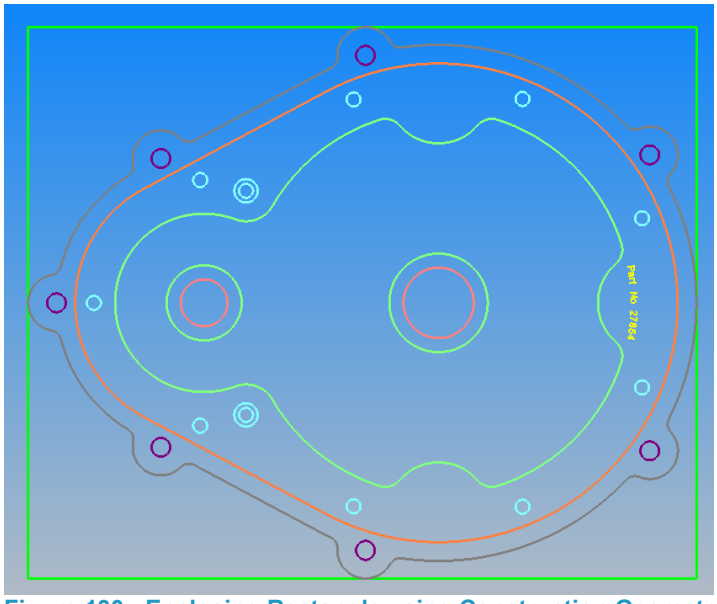


Figure 130 - Enclosing Rectangle using Construction Geometry

Select **EDIT > Break Join etc. > Offset** 

Set the Offset Value to 6, the option to ⊙ **Geometry**, and check ☑ **Delete Original**
then **<LClick> [OK]**.
Select the enclosing rectangle then a point on the outside, the new rectangle will be drawn.

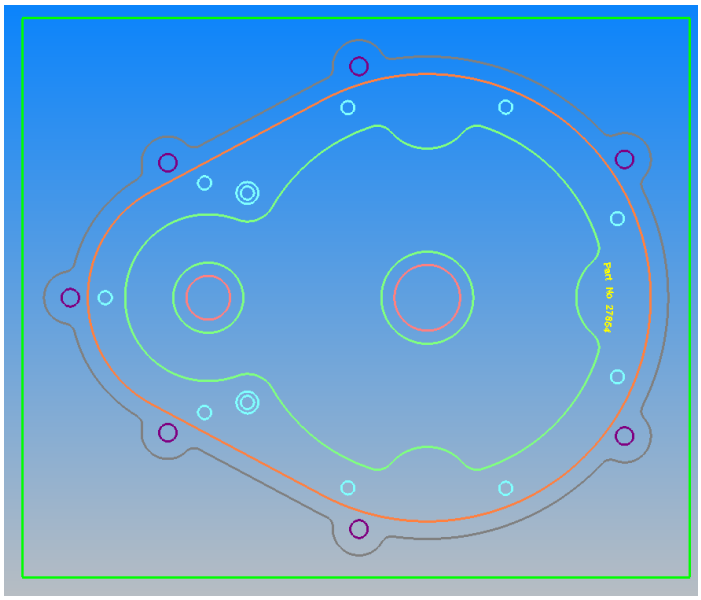


Figure 131 - Rectangle adjusted to suit actual material form

Set the Material

Select **3D > Set Materials**
Select the rectangle and the dialogue will be displayed.

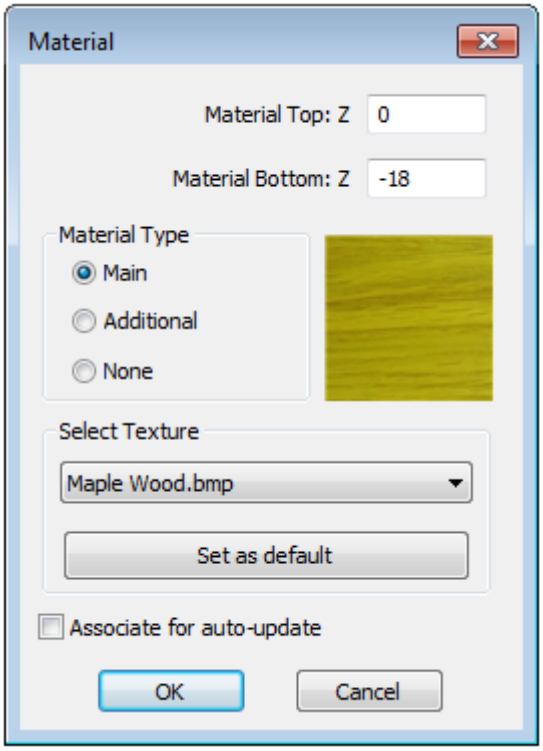


Figure 132 - Set Materials Dialogue options

Set the options for the Material Top Z to 0 and the Material Bottom Z to -18.
Select **[OK]**.

With either of these options it is possible to assign a texture image to the material to simulate the appearance of an actual material.

The default location for these images is **"…. LICOMDAT/Textures"**

Any standard image file can be used, BMP, JPG, PNG etc., the size of the image will dictate how the item will appear in ALPHACAM.

Move the job to a suitable position

Select **EDIT > Move Copy etc. > Move**

Select the **[All]** option on the prompt line then **<RClick>**.
For the Base point using the auto snap select the lower left corner of the rectangle.
For the new position type **0,0**

> If you have any items hidden, you MUST make them visible when you move the part to the machining location or they will be left in the original positions.

Set the ISO view

This will enable you to see the geometry and material.

Select **VIEW > VIEWS > Iso**

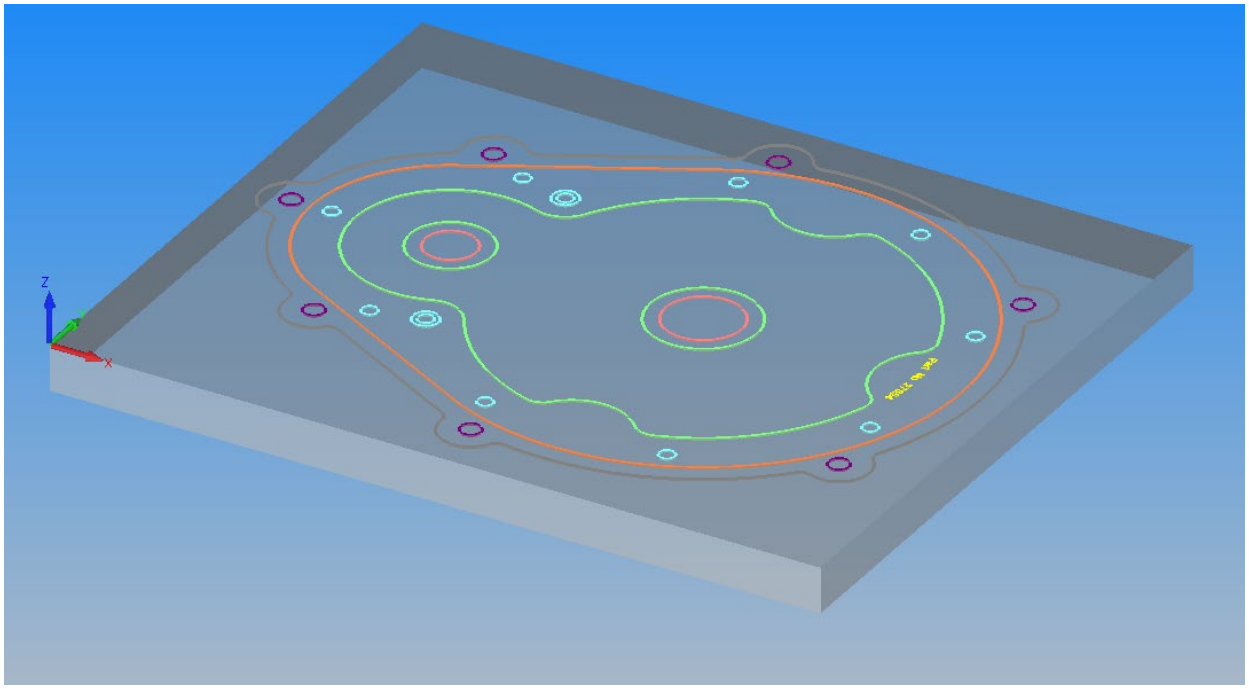


Figure 133 - Part moved to correct location for machining

Prior to saving the job ensure that the geometry profiles only have one ghost tool.

Select **<Ctrl> + <G>** or **View > Display Options > Ghost Tools** to display the ghost tools.

Save the Job

Select **FILE > Save**
The save dialogue is displayed, Select the required location folder then enter **Housing Plate** as the job name.

Setting Geometry Z Levels

If you wish to machine the component using **Auto Z** machining (**Standard ALPHACAM** and higher) then you will need to set the geometry Z Levels.

This allows for a method of machining that takes the depth information directly from the drawing file rather than the user entering values in the machining dialogue options.

Select **3D > Set Geometry Z Levels** 
The system prompts you Select Geometries, **<LClick>** on Geometries **1, the outer 8mm holes, plus geometries 9, & 10**

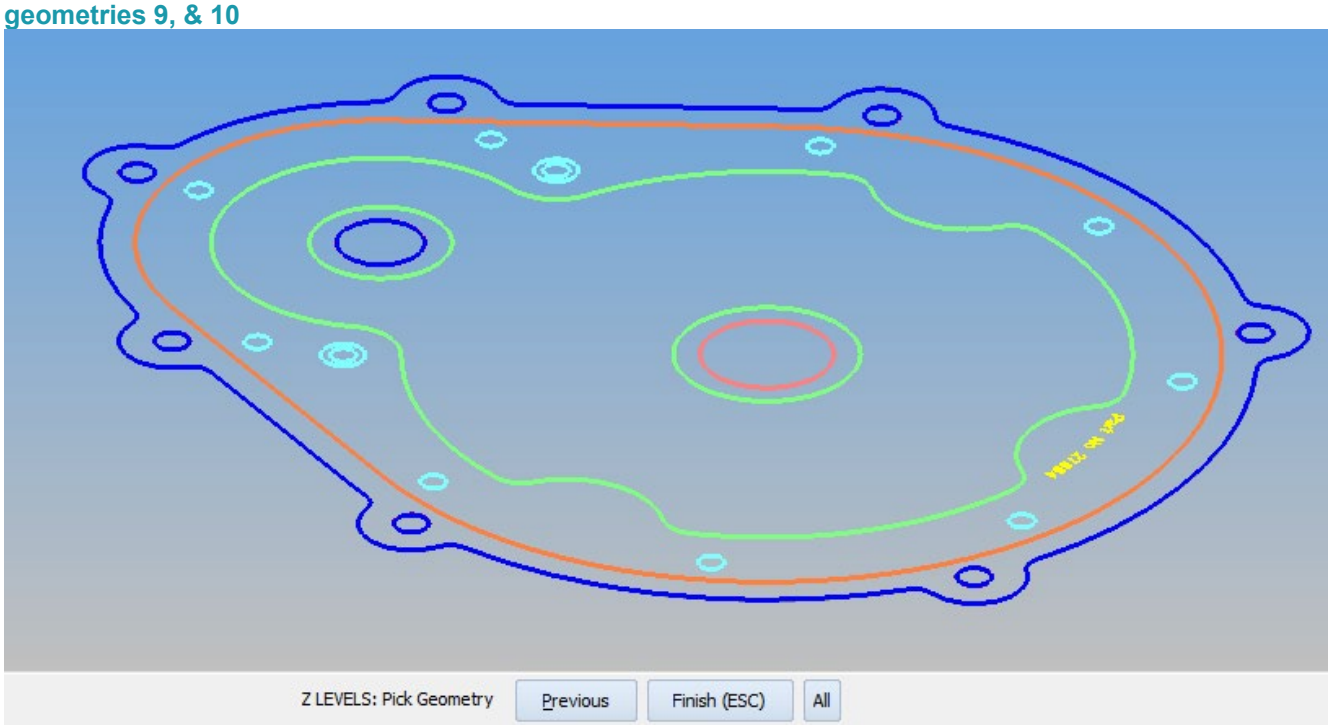


Figure 134 - Items to select for first Z Level application

Then **<RClick>**.
In the Dialogue Set the **Top Z = 0** and the **Bottom Z = -18** then **<LClick> [OK]**.

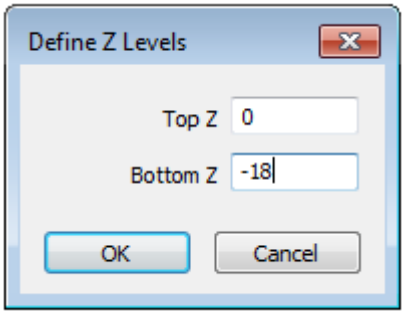


Figure 135 - Set Geometry Z Levels Dialogue

Continue with the remaining options as detailed below;

<LClick> on Geometry **4 (the Ø6 circles on the R13 centres)** then **<RClick>**.
In the Dialogue Set the **Top Z = -3** and the **Bottom Z = -12** then **<LClick> [OK]**.

<LClick> on Geometry **2 (the inner oval)** then **<RClick>**.
In the Dialogue Set the **Top Z = 0** and the **Bottom Z = -3** then **<LClick> [OK]**.

<LClick> on Geometries **3** (the Ø6 Equi-Spaced circles) then **<RClick>**.
In the Dialogue Set the **Top Z = -3** and the **Bottom Z = -10** then **<LClick> [OK]**.

<LClick> on Geometry **5** (Ø10 circles) then **<RClick>**.
In the Dialogue Set the **Top Z = -3** and the **Bottom Z = -3** then **<LClick> [OK]**.

<LClick> on Geometries **6 (the shaped main pocket) then <RClick>**.
In the Dialogue Set the **Top Z = -3** and the **Bottom Z = -12** then **<LClick> [OK]**.

<LClick> on Geometries **7 & 8** (the offset circles of the central set of four) then **<RClick>**.
In the Dialogue Set the **Top Z = 0** and the **Bottom Z = -12** then **<LClick> [OK]**.

<RClick> to finish the command.

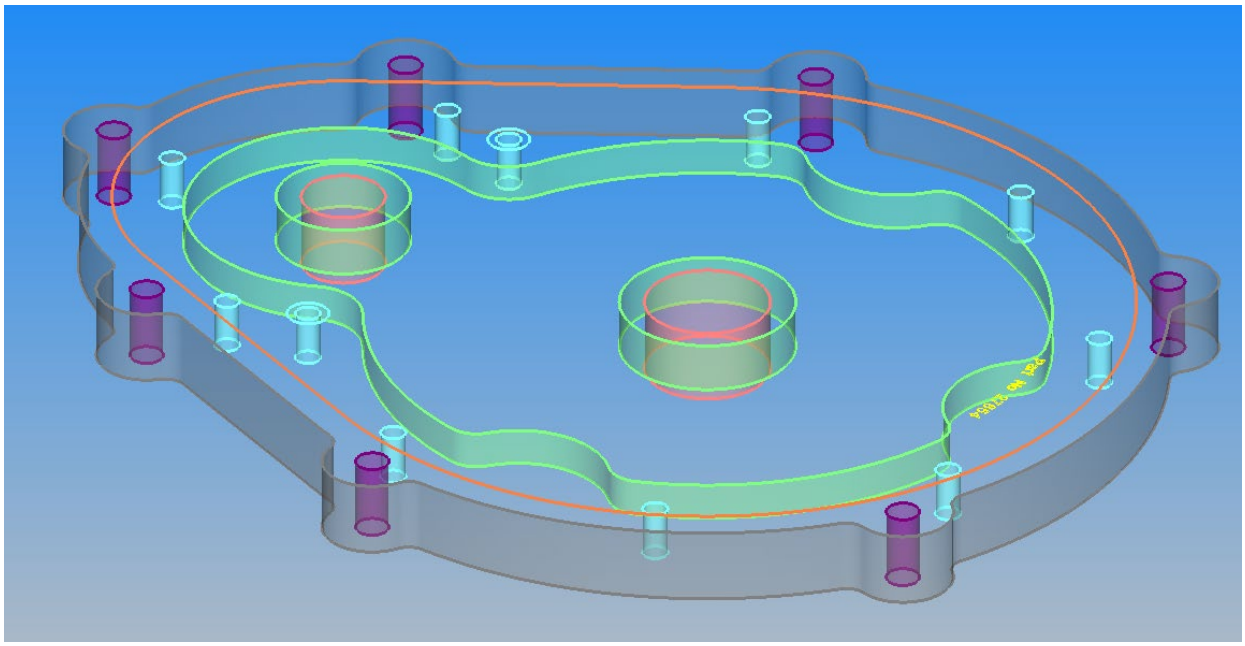


Figure 136 - Finished part with Geometry Z Levels applied

Note that you cannot assign Geometry Z Levels to the text on this drawing and the resulting machining method does not support Auto Z machining any way.

Version amendments

V	Amendment Description	A	Software Version	Amended Date
13	Minor text formatting alterations.	1	2020.1	02/10/2019
13	Template altered to Hexagon branding	0	2020.0	15/03/2019

HEXAGON
MANUFACTURING INTELLIGENCE

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