



HEXAGON
MANUFACTURING INTELLIGENCE

ALPHACAM 2020.1 FEATURE EXTRACTION



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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.

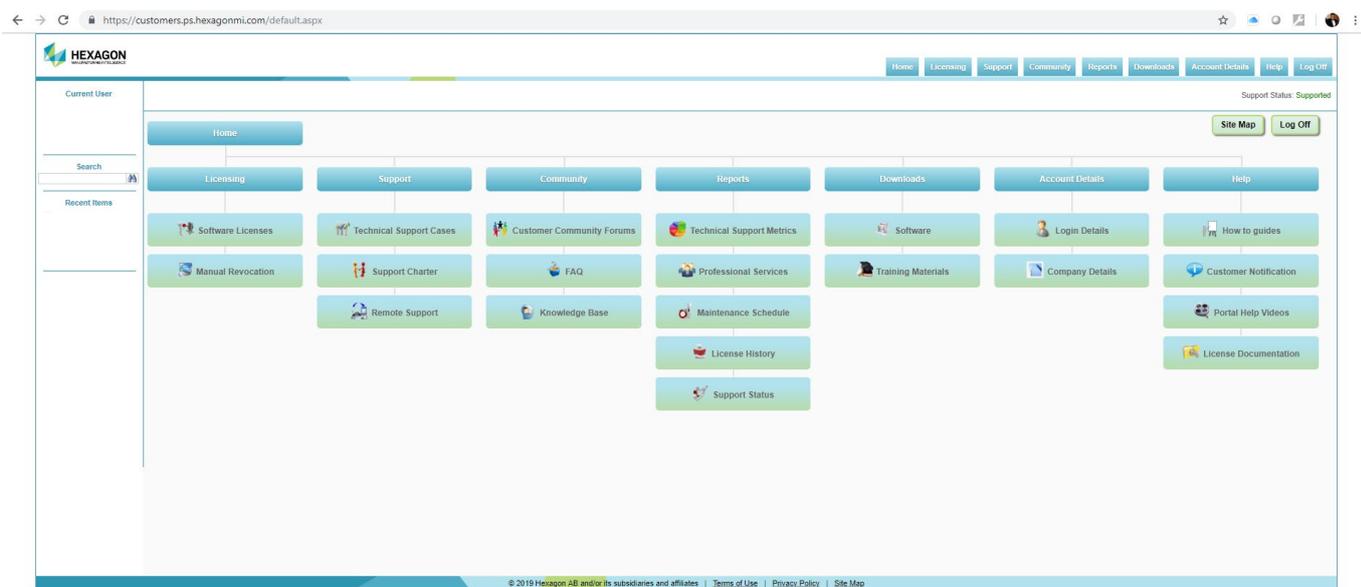
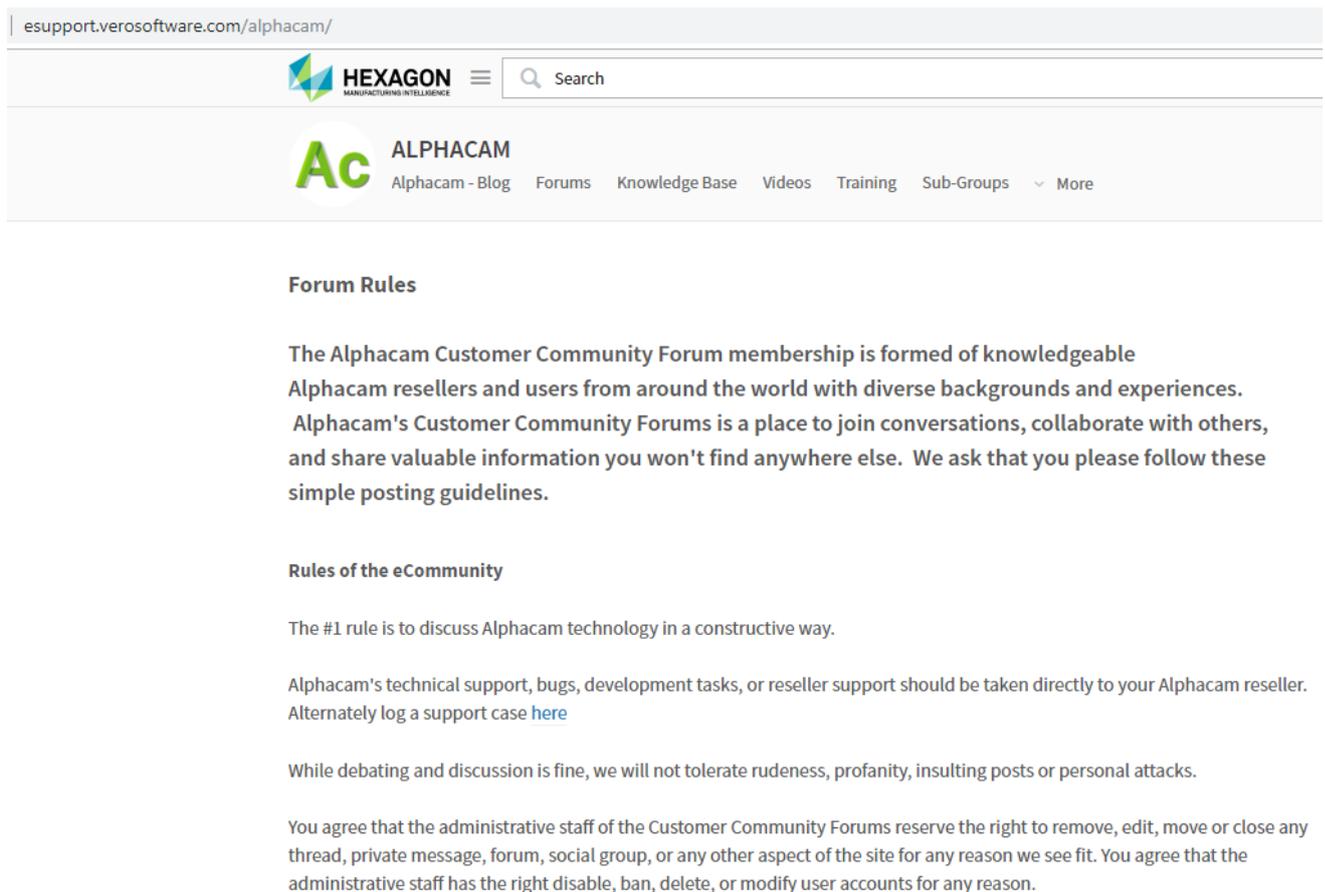


Figure 1 - Customer Support Portal

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](#).



The screenshot shows the ALPHACAM esupport page. At the top, there is a navigation bar with the Hexagon logo, a search bar, and a menu. Below the navigation bar, there is a section for ALPHACAM with a large 'Ac' logo and a list of links: 'Alphacam - Blog', 'Forums', 'Knowledge Base', 'Videos', 'Training', 'Sub-Groups', and 'More'. The main content area is titled 'Forum Rules' and contains the following text:

Forum Rules

The Alphacam Customer Community Forum membership is formed of knowledgeable Alphacam resellers and users from around the world with diverse backgrounds and experiences. Alphacam's Customer Community Forums is a place to join conversations, collaborate with others, and share valuable information you won't find anywhere else. We ask that you please follow these simple posting guidelines.

Rules of the eCommunity

The #1 rule is to discuss Alphacam technology in a constructive way.

Alphacam's technical support, bugs, development tasks, or reseller support should be taken directly to your Alphacam reseller. Alternately log a support case [here](#)

While debating and discussion is fine, we will not tolerate rudeness, profanity, insulting posts or personal attacks.

You agree that the administrative staff of the Customer Community Forums reserve the right to remove, edit, move or close any thread, private message, forum, social group, or any other aspect of the site for any reason we see fit. You agree that the administrative staff has the right to disable, ban, delete, or modify user accounts for any reason.

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.

Current Model Import Versions

Geometric Data in one of the following formats can be imported into ALPHACAM.

E = Essential **S** = Standard **A** = Advanced **U** = Ultimate

TRANSLATOR AND VERSION	ROUTER	STONE	LATHE	MILL	WIRE	PROFILING
DXF/DWG (AutoCAD 2019)	E S A U	E S A U	E S A U	E S A U	S A	A U
IGES (not version specific)	E S A U	E S A U	E S A U	E S A U	S A	A U
Rhino 5.0	S A U	S A U	S A U	S A U	S A	A U
STL (not version specific)	S A U	S A U	S A U	S A U	S A	A U
SketchUp 2019	S A U	S A U	S A U	S A U	S A	A U
Adobe PDF	E S A U	E S A U	U	E S A U	S A	A U
Adobe Illustrator	E S A U	E S A U	U	E S A U	S A	A U
PostScript	E S A U	E S A U	U	E S A U	S A	A U
STEP AP203/AP214/AP242 (preferred Schemas)	A U	A U	A U	A U	A	U
Part Modeler (current)	A U	A U	A U	A U	A	U
VISI Version 20	A U	A U	A U	A U	A	U
Autodesk Inventor 2019 (current, IPT & IAM files) 	A U	A U	A U	A U	A	U
ACIS R1 – 2017 1.0	A U	A U	A U	A U	A	U
Parasolid 9.0 – 29.0.137	A U	A U	A U	A U	A	U
SOLIDWORKS 2019 (SLDPRT files only)	A U	A U	A U	A U	A	U
Solid Edge ST9 (PAR & PSM files)	A U	A U	A U	A U	A	U
Spaceclaim 2018	A U	A U	A U	A U	A	U
JT Open (8.x and 9.x)	A U	A U	A U	A U	A	U
EXTRA COST						
Catia V4 4.1.9 – 4.2.4	A U	A U	A U	A U	A	U
CATIA V5 V5R8 – V5-6R2017	A U	A U	A U	A U	A	U
CATIA V6 Up to V6 R2017x Note: CATIA V6 users should export their database objects as CATIA V5 CATParts or CATProducts.	A U	A U	A U	A U	A	U
Creo Parametrics 4.0 (formerly Pro/E Wildfire)	A U	A U	A U	A U	A	U
NX11 (formerly Unigraphics NX)	A U	A U	A U	A U	A	U

 In order to import Autodesk Inventor files, **Autodesk Inventor®** or **Inventor® View™** must also be installed. **Inventor® View™** can be downloaded from the

Feature Extraction Configuration

HOME > Configure > Feature Extraction 

This dialogue is split into 6 tabbed sections.

General

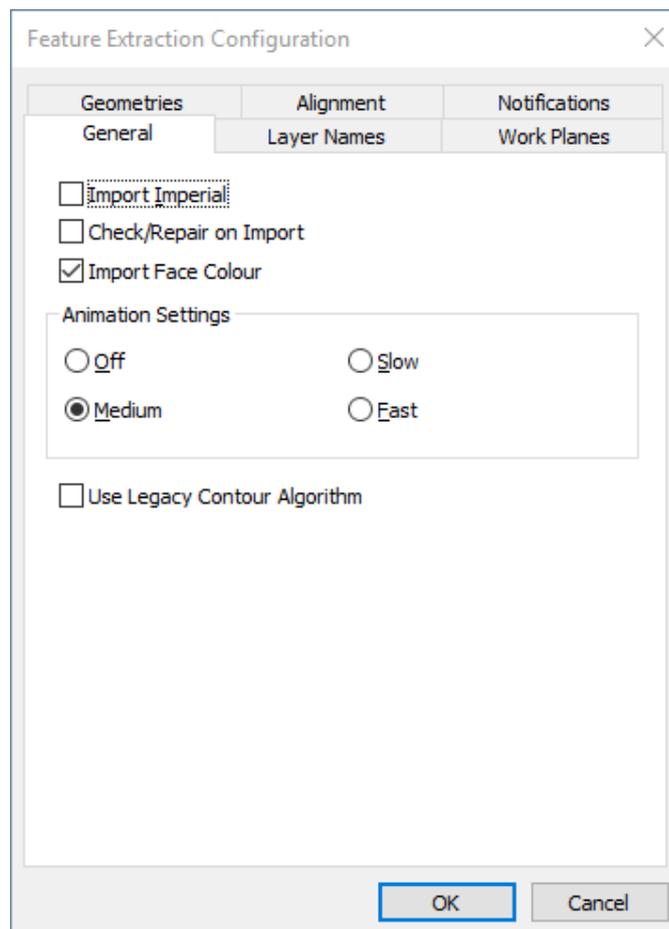


Figure 3 - Configuration General Tab

Import Imperial

If the models you import are always required to be scaled up to be imperial, checking this option will automatically scale all models on import by 25.4.

Check/Repair on Import

This causes every model imported to be checked for inconsistencies and an attempt at repair made. This is only necessary if you import very poor models. This option can be run on individual models from the Solid Model Utilities menu.

Import Face Colour

If selected, this option retains the original colours applied to the model.

Animation Settings

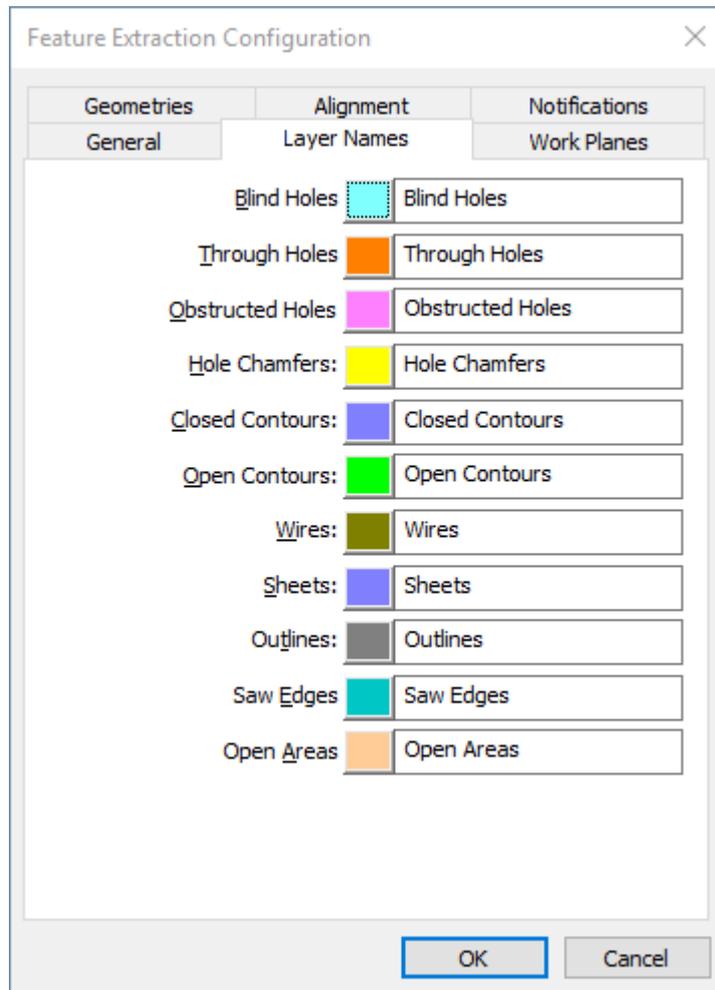
Allows for control of the on screen movement of parts during Feature Extraction without resorting to the main settings options.

Use Legacy Contour Algorithm

Forces the extraction to use the previous revision of the process in case of any perceived errors.



Layer Names



This section allows you to set the Layer names for the different features extracted.

Figure 4 - Configuration Layer Names Tab

Work Planes

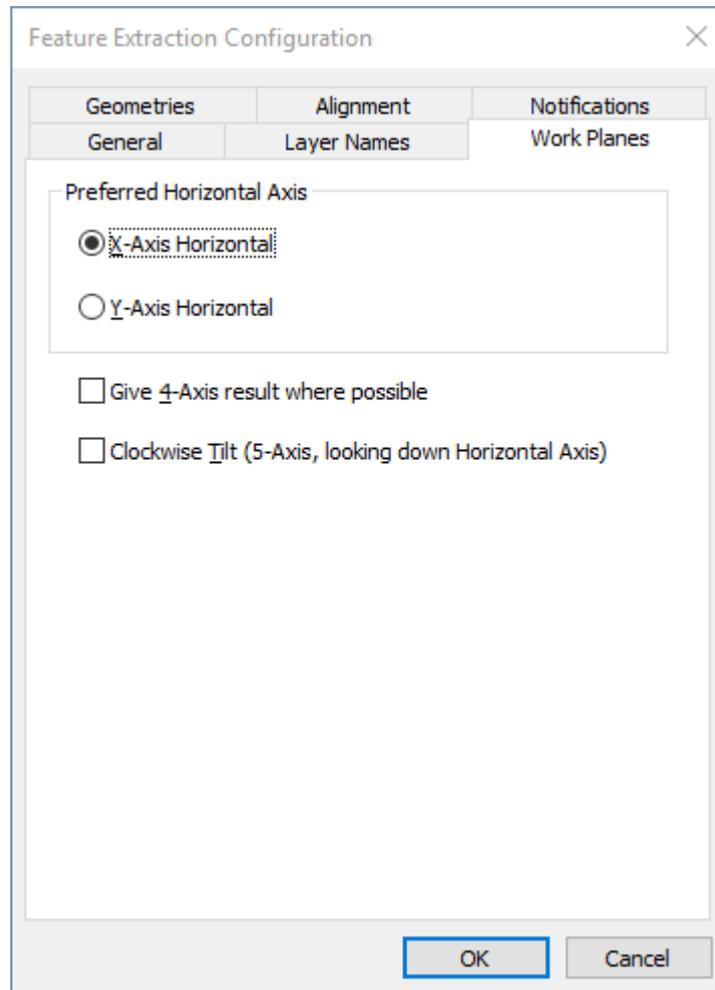


Figure 5 - Configuration Work Planes Tab

This section allows you to set the orientation of any work planes created from the model. You can set which axis is to be kept horizontal and the direction of tilt for 5-Axis machines.

The options used are dependent upon your machine tool configuration.

Geometries

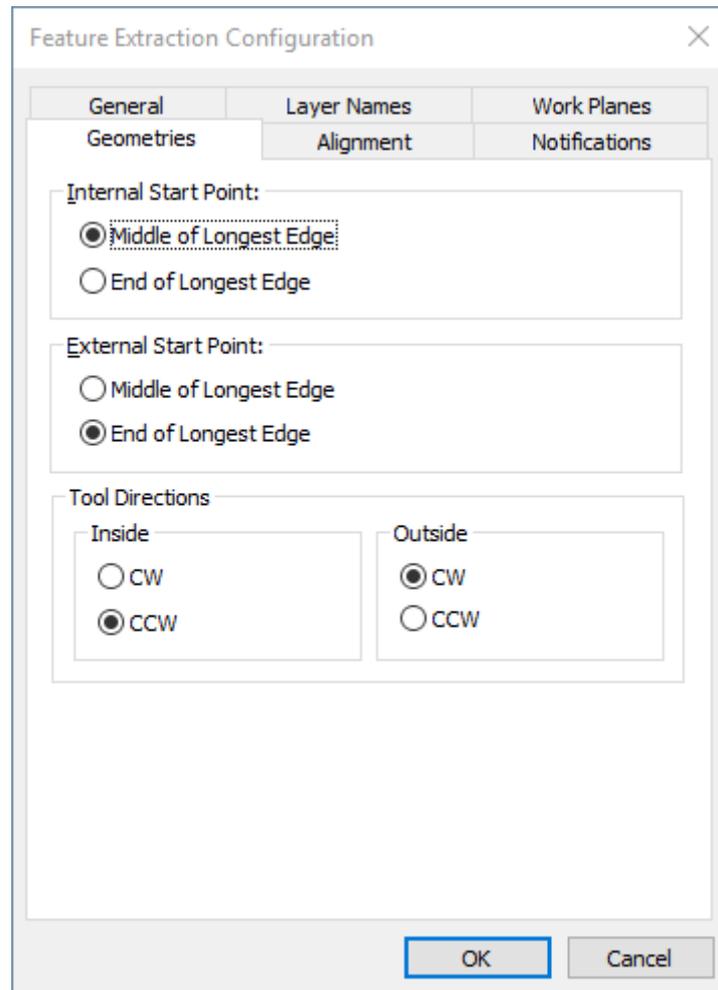
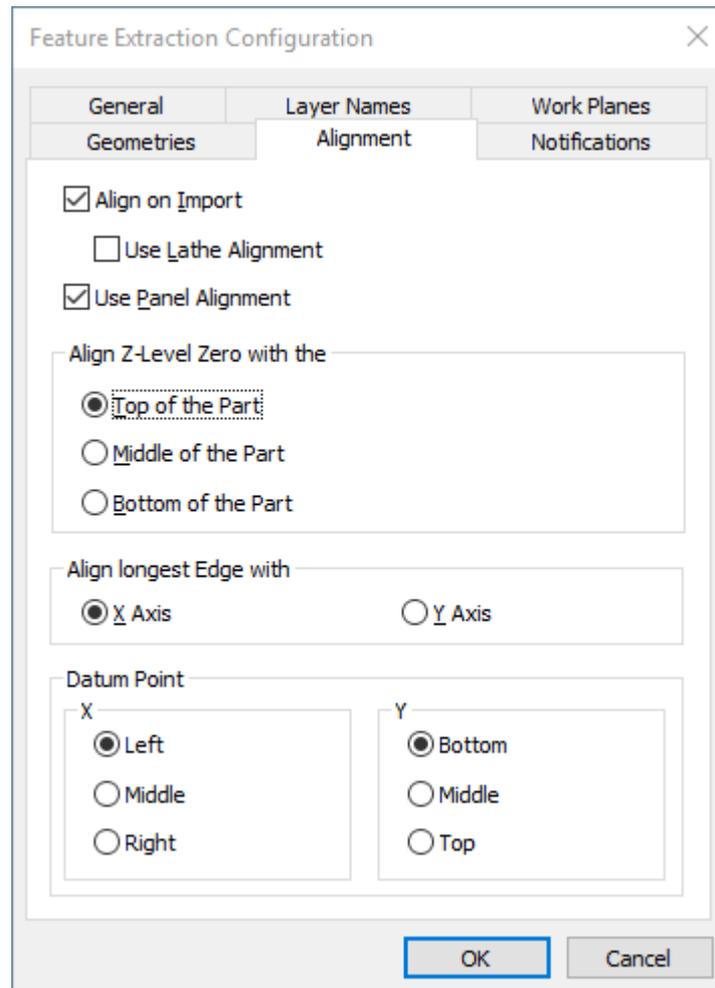


Figure 6 - Configuration Geometries Tab

This allows you to set where the system is to position the start point on extracted geometries ready for adding machining.

You are also able to set your preferred machining directions for Inside (Pockets) and Outside (Profile) geometries.

Alignment



This settings page allows you to set the automatic positioning and orientation of the part with respect to the XY&Z axis for suitable manufacturing.

Figure 7 - Configuration Alignment Tab

- Align the part automatically when it is imported. This will also permit alignment to suit the thickness of a panel using the **Use Panel Alignment** option or a major cylinder axis using the **Use Lathe Alignment** option.
- Align Z zero with the Top, Middle, or Bottom of the model.
- Align the longest axis along the X or Y global axis.
- Align X zero with the Left Middle or Right of the model (looking in the X/Y view).
- Align Y zero with the Top, Middle, or Bottom of the model (looking in the X/Y view).

Notifications

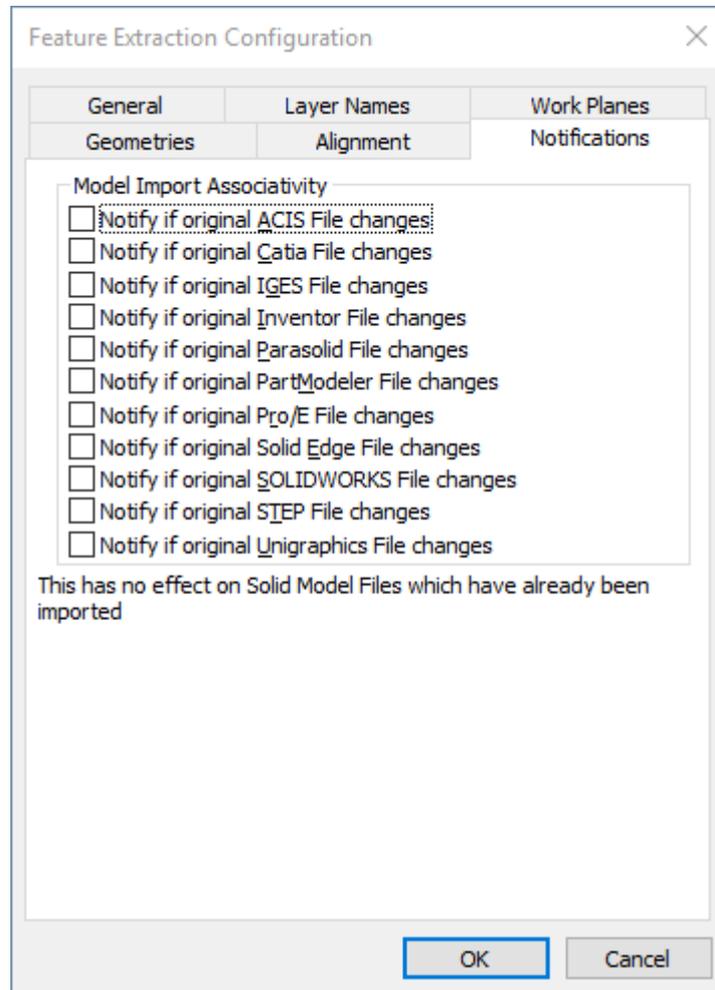


Figure 8 - Configuration Notification Tab

This section allows you to set the associativity between the model in the ALPHACAM drawing and the original solid file.

The associativity is only applicable for models imported after the associativity has been activated. It has no effect on previously imported models.

When these option(s) have been activated you will be notified when opening the drawing if the models modified dates are different to the original model.

Feature Extraction Commands

The Feature Extraction commands are split into 3 sections

Solid Model Utilities

COMMAND	ICON	DESCRIPTION
Auto-Align Part		Makes ALPHACAM attempt to auto-align the Solid Model's bounding rectangle to the X0, Y0, Z0 coordinate.
Align Part		Allows you to Align a Solid Model manually.
Auto Align Lathe Part		Makes ALPHACAM attempt to auto-align the solid models' major axis along the Z axis of the global datum.
Align Lathe Part		Allows you to Align the Solid Model manually along the Z axis.
Cap Holes		Allows you to add additional stock to holes when using 3D Machining.
Remove Contours defined as Holes		Makes ALPHACAM scan all contours in the drawing to see if any of the contour Z-Levels match the Z-Levels of holes that are found.
Offset Faces		Allows you to offset face(s) of a Solid Model in ALPHACAM as an STL.
Create Bounding Work Volume		Allows you to easily create a Work Volume that will fit around a Solid Model.
Reset Geometry Z-Levels		Allows you to reset the Z-Levels of any geometry that has been extracted from your Solid Model.
Set View down Normal		Allows you to change the view of the drawing area to look down upon a Cylinder, Cone, or Plane.
Face/Edge/Point Details		Causes ALPHACAM to display information about the Faces, Edges, and Points that your Mouse Pointer is hovering over. Disables AutoSnap if in use.
Check/Repair Part		Allows you to check a Solid Model to ensure that it will be able to be used in ALPHACAM.
Set Face Colour		Allows you to designate a colour to a Face. This colour will only be shown in Shading.
Paint Faces		Allows you to set the colour of selected Solid Model faces.
Feature Extraction Configure		Accesses the settings menu in the same manner as HOME > Configure > Feature Extraction

Solid Model Extraction

COMMAND	ICON	DESCRIPTION
Automatic Extraction		Automatically extracts features from any Solid Models that are visible in the current drawing.
Drillable Hole Extraction		Extracts any Holes that can be considered Drillable by ALPHACAM from a Solid Model by manual selection.
Hole Chamfer Extraction		Extracts Holes from a Solid Model that use a conical pattern as opposed to a standard straight hole pattern.
Contour from Picked Edges		Extracts a 2D contour from selected edges.
Contour around Picked Faces		Extracts a contour from selected faces. (creates Geometry on work plane)
Contour through the Current Work Plane		Extracts a contour through the entire model based on the current active work plane.
3D Edge Extraction		Extracts a 3D Polyline from a Solid Model.
Z-Level Section		Extracts contours at various values in the Z direction.
Profiles from extended Edges		Extracts profiles from a Solid Model by selecting 3 or more adjacent Faces.
Edge for Sawing		Extracts a geometry with top and bottom Z values and correct work plane if the face is at any angle.
Surfaces from Faces		Extracts Surfaces from any faces that exist in a Solid Model.
Edges from Face Outlines		Creates a 3D Polyline that represents the outline of the selected face(s) of your Solid Model.
Extract Solid Body Outlines		Extracts the body outline of all visible Solid Models.
Spun Outline Extraction		Extracts an individual geometry that is shaped as the profile of the Solid Model.
Projected Face Outlines to Work Plane		Extracts an outline of selected faces and have that outline projected to the active Work Plane.

Associated Commands

Work Plane Commands

All the standard Select Work Plane commands can be used given their respective geometry and view criteria. The following 3 options are particularly applicable when working with solids.

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. These commands allow you to establish an active work plane onto which, 2D geometry features can be extracted or projected, or in which toolpaths can be created.

Best Fit Plane Through Points.

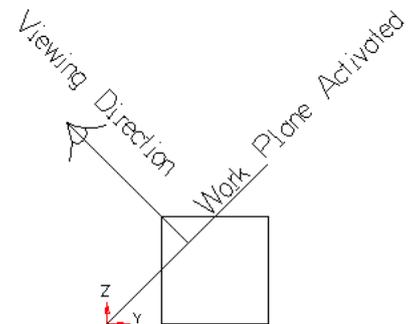
This will activate a work plane that fits evenly through the points in a 3D spline or polyline the spline or polyline must contain at least 6 points.



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

Normal to View

This will activate a plane which passes through the global datum that is normal to the current view; a plane is created through the model as you are looking at it on screen when the command is selected.



Create 3D Work Plane

This command allows for the creation of a dedicated work independent of the solid model. When actioned, a dialogue box allows for the direct input of Tilt and Twist angles to suit the required work plane attitude to the chosen solid model.

View Commands

Set View down Normal

This will create a view looking perpendicular to selected Solid Model Face.
Accessed from the Solid Model Utilities tab.

Set View Normal to Work Plane

This sets the view to a 3D view looking normal to and through the current work plane.
Accessed from the Views tab.

Manipulation Commands

3D > 3D Part Rotation

This allows solids and other geometry types to be rotated about an axis.
Or
Rotated to a flat plane from 3 points selected on the model.

Feature Extraction

The process

1. Import the cad model(s) either using the **HOME > Input Cad** option or drag and drop the file into ALPHACAM.
2. **Align** the model so it is in the correct orientation for manufacture.
3. **Position** the model so it is in a position suitable for machining.
4. Use the **Feature Extraction** options to extract the required geometries or use **Standard Geometry** creation commands to create geometry that cannot be extracted or where the feature is not available for extraction.
5. Organise geometries onto **User Layers** to suit ease of application of machining.
6. Check and set **Tool Directions** and **Start Points**.
7. Apply machining strategies either from the machining menu or by Machining styles.

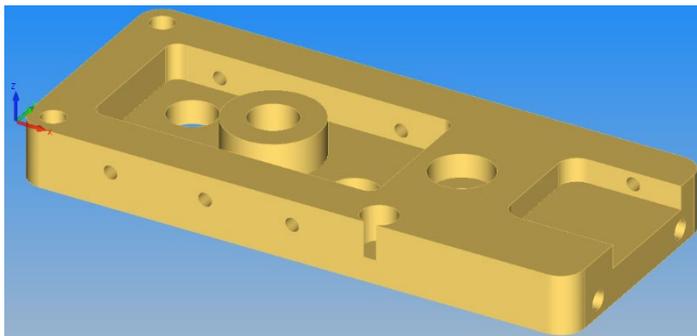


Figure 9 - Solid Model

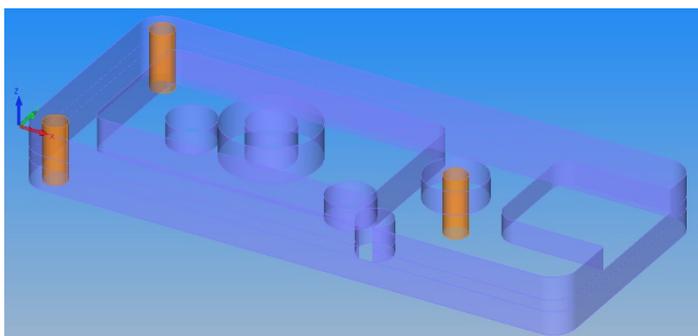


Figure 11 - Features Extracted

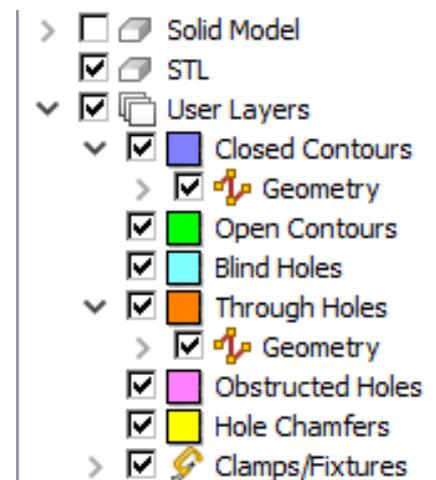


Figure 10 - Resulting User Layer structure

The Ideal scenario is to import the model and run automatic extraction to acquire the required geometries. These geometries are then machined using Auto Z Machining.

However, as we all know the ideal scenario does not happen all the time due to the condition of the model and the features that are required for machining.

When using **Feature Extraction**, it is important to remember that all normal geometry creation commands are still applicable.

Automatic Extraction

This command will automatically extract 2D features from solid models. 2D contours and holes are extracted from the model with an upper and lower geometry to represent the Z values.

Tool Directions are automatically set, enabling **Auto Z** machining and/or **Machining Styles** to be easily applied.

The command is split into three distinct areas accessed by the tabs along the top.

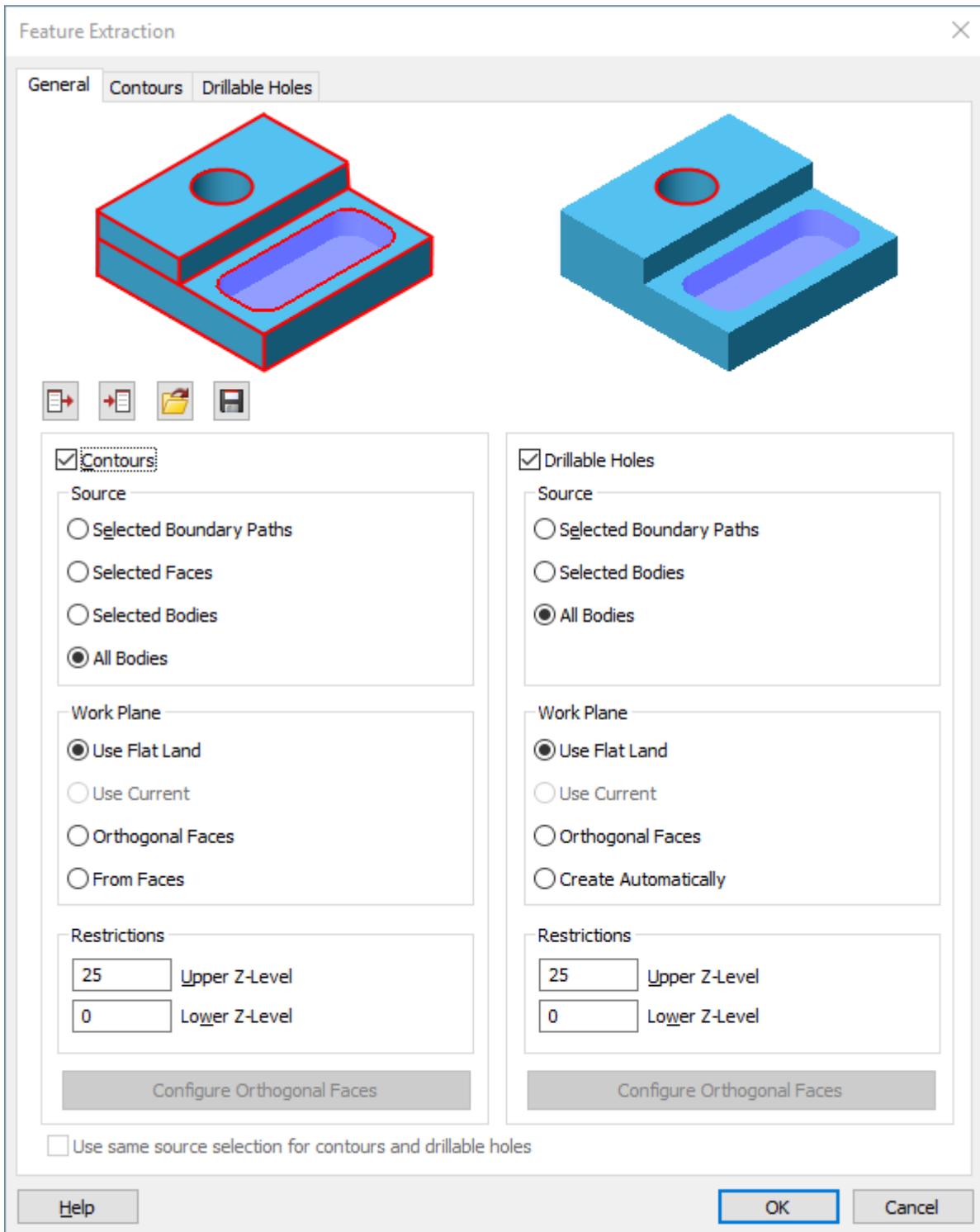


Figure 12 - Automatic Extraction dialogue

General tab

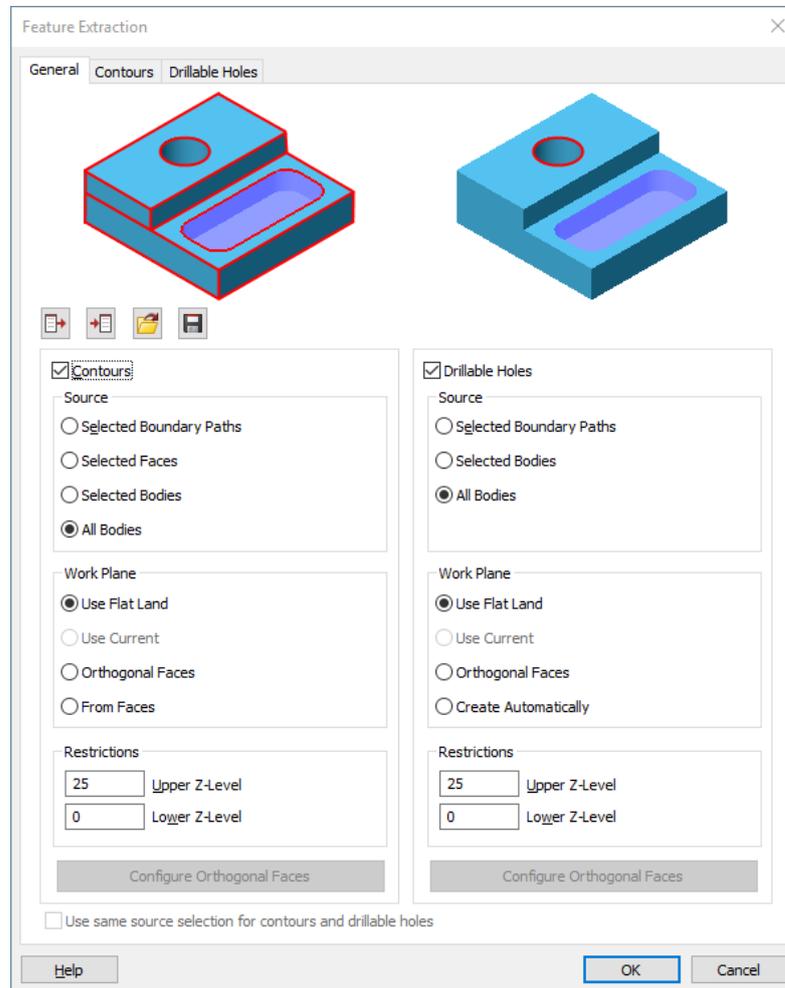


Figure 13 - Automatic Extraction General tab

- Contours and/or
- Drillable holes

If ticked, the corresponding **Contours** or **Drillable Holes** tabs become available.

Source

Selected Boundary Paths

Extracts all areas contained within or crossing the selected 2D boundary. Ghost Tool side is ignored.

Selected Faces

Extracts all areas between the Z levels described by the selection of two or more faces.

Selected Bodies

Extracts elements from the chosen body or bodies

All Bodies

Extracts all elements from all visible bodies.

Work Plane	⊙ Use Flat Land	Extracts elements to Flat Land if there are no alternatives.
	⊙ Use Current	Extracts to the current work active plane.
	⊙ Orthogonal Faces	Extracts to one or more of the Cardinal faces of a part based on a selection process.
	⊙ Create Automatically	Creates Work Planes automatically when extracting drillable holes at any attitude. BUT does not extract non-hole geometries.
	⊙ From Faces	Creates Work Planes automatically when extracting contour features at any attitude. BUT does not extract hole geometries.
Restrictions	Limits the Upper and Lower Z values for the extraction to take place within.	



Note that when using the ⊙ **Create Automatically** feature option and the ⊙ **From Faces** option, should any extracted contours or holes lay on the same angled face, a single work plane is generated to contain both extracted items.

Configure Orthogonal Faces

This option allows for the configuration of which of the six cardinal faces are to be analysed for Feature Extraction.

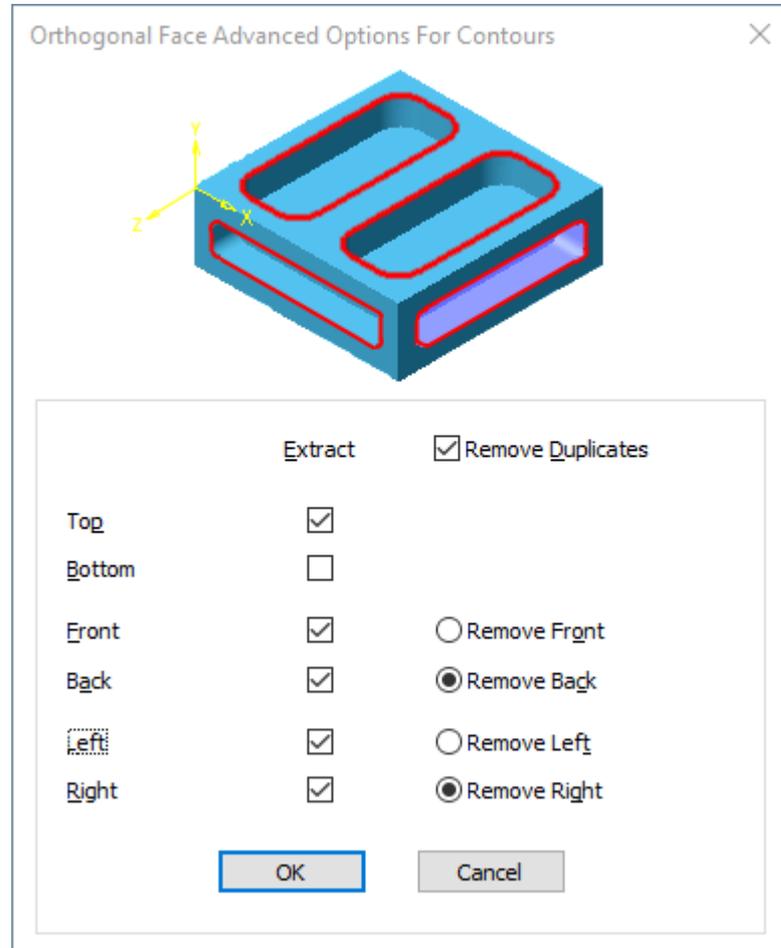


Figure 14 - Orthogonal face configuration dialogue options

Using the **Remove Duplicates** option prevents repetitive geometries from being created during the process.



Note that even after using this form of extraction processing, it may still be necessary to use the **Delete** command to remove unwanted geometries.

Configure Faces

This option allows for the configuration of which faces are to be analysed for Feature Extraction of contours.

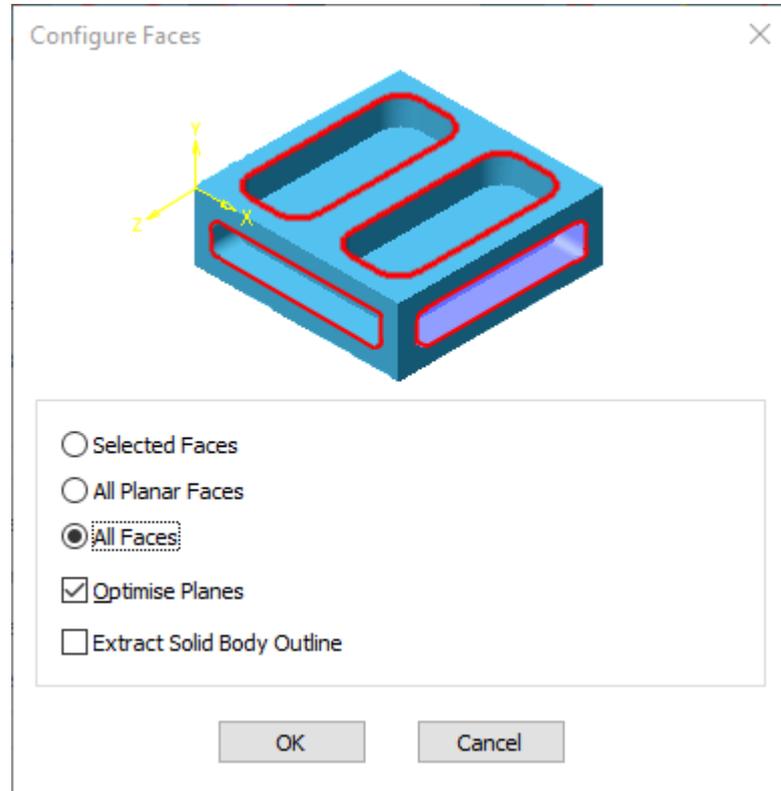


Figure 15 - Dialogue options for Face extraction of Contours

- Ⓒ Selected Faces
Allows individual selection of the model faces to be analysed.
- Ⓒ All Planar Faces
Extracts all features from any flat faces.
- Ⓒ All Faces
Extracts all features from all model faces, even if they are unsuitable for manufacture depending on the model alignment.
- ☑ Optimise Planes
If active, this option creates a single work plane for all features lying in the same planar angle and assigns the correct Z Geometry Level to them.
If not active, features at differing levels will have their own work planes created even though they are in the same planar angle.
- ☑ Extract Solid Body Outline
Extracts the solid model edges from the part in the global Z direction, Flat Land.

Drillable Holes

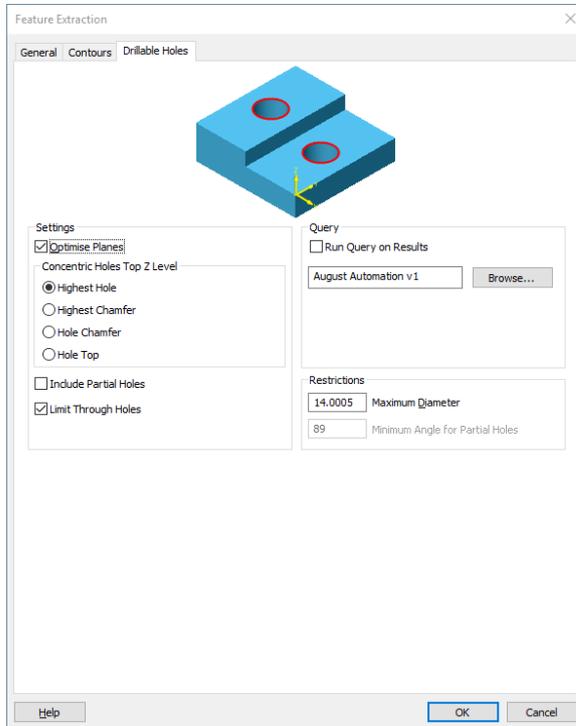


Figure 16 - Drillable Holes configuration tab

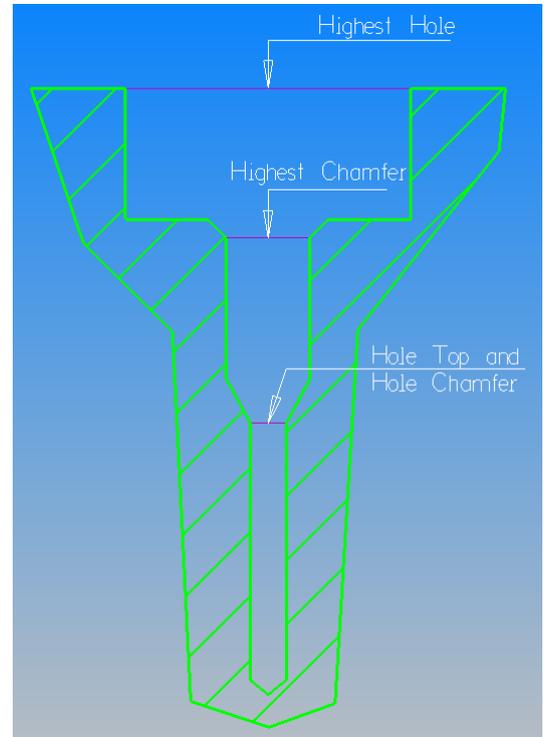


Figure 17 - Hole level diagram

Settings

Optimise Planes

Places all parallel holes that are found on one Work Plane as opposed to multiple Work Planes.

Concentric Holes Top Z Level

Finds concentric holes and sets the Z top of the extracted items to the selected option.

Include Partial Holes

If checked, all found partial holes (open arcs) will be considered as drillable holes based on the angle option set

Limit Through holes

Generates a single geometry for any holes that are found which go through the Model if **NOT** ticked.

Sets the **Maximum Diameter** for drillable holes.

Restrictions



Maximum Diameter also dictates which extracted Layer the geometry will be assigned to. This value and below, the geometry is assigned to one of the three Drillable Holes layers. Above this value, the geometries are assigned to one of the Contours layers dependent on the option settings.

Sets the **Minimum Angle for Partial Holes** to be included.

Query

Allows for the running of a query to further sort the extracted geometries.

Contours

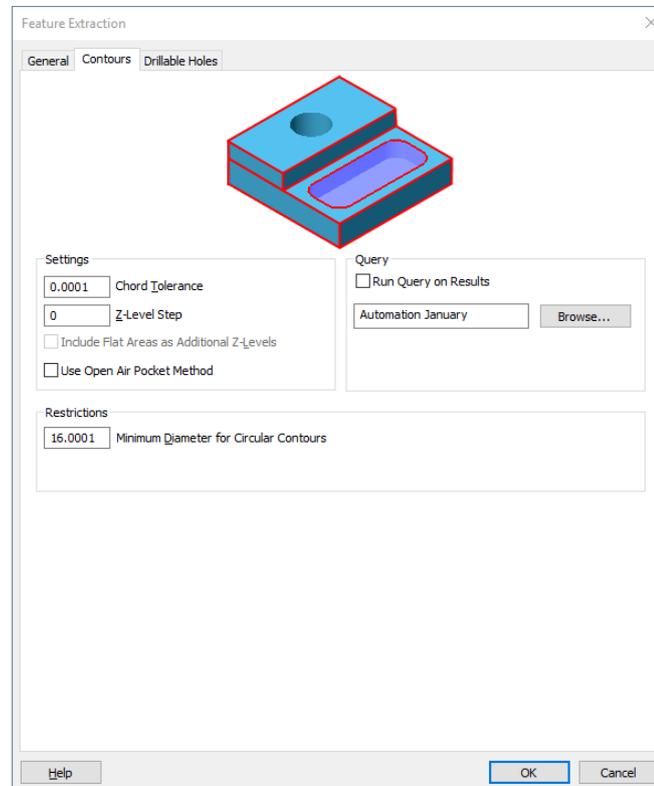


Figure 18 - Contours configuration tab

Chord Tolerance

Allows you to specify the tolerance that ALPHACAM will use to convert Spline edges to paths.

Allows you to specify the steps that ALPHACAM will take past the Z 0 coordinate to look for extractable items.

Z-Level Step



Note that 0 will produce a single extracted geometry set; a numeric figure will split the part into slices of that value down the Z axis.

Settings

Include Flat Areas as Additional Z-Levels

Includes any flat areas found on Islands in the extraction process as additional Z-Levels.

Use Open Air Pocket Method

This option will cause the contour finder to attempt to return pocketed areas and outline geometries rather than simply closed contours.

Restrictions

Sets the minimum diameter for circles to be defined as contours.

Query

Allows for the running of a query to further sort the extracted geometries.

Depending on the **From** option that you have selected, the next steps may differ slightly:

- If you chose to use **Boundary Paths** as your From option, you will be required to select the geometry or geometries that you wish to use as your boundary(ies) using your Mouse Pointer.
- When using a Boundary Path, you should note that any Contours or Holes that partially cross the boundary will be included in the Feature Extraction process.
- If you chose to use **Selected Faces**, you will be required to select the Face(s) of the Solid Model that you would like to extract from using your Mouse Pointer.
- If you chose to use **Selected Bodies**, you will be required to select the Body(ies) that you would like to use to extract from.
- If you chose to use **All Bodies**, ALPHACAM will extract features from every single Solid Model that exists and is visible in the current drawing.

Query

The query option is a method by which a script is run that tests each geometry that is extracted against one or more conditions:

E.G.

IF <is Circle> and **<is less than or equal to>**, **<value>** move to Layer **<name>**

This then allows you to further automate the process of Feature Extraction to suit any Machining Styles with Layer assignments or AutoStyles you have created by moving items from the default Feature Extraction layers to your own layer set up.



These Query scripts need to be written with care so that the process runs in the correct order and geometry items are worked upon correctly.

#1 Feature Extraction Example 1, Face by Face

Import the model

Select **HOME > Input CAD**  and the following dialogue is displayed.

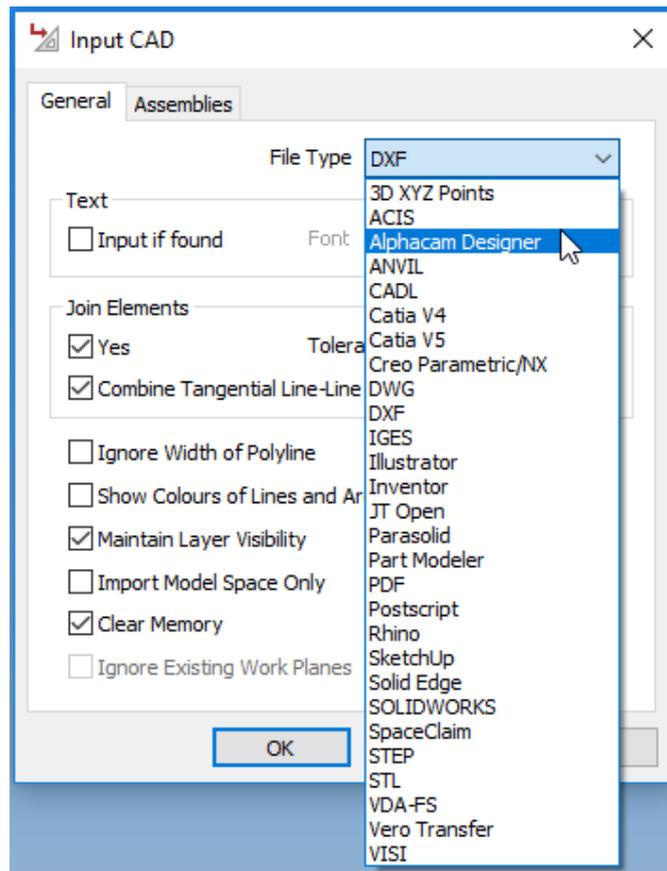


Figure 19 - Input Cad dialogue set for ALPHACAM Designer

Set the file type to **Alphacam Designer**, **Clear Memory**, and then select **[OK]**
The file selection dialogue is displayed.

Navigate to the “.....\ALP TRG 114 Feature Extraction 2020\Examples\Alphacam Designer Examples” folder and select the “#1 Feature Extraction Example 1.vdf” then select **[Open]**.

The model is imported and displayed on screen.

Drag & Drop

Note that if you can access the required folder of the model via Windows Explorer, ALPHACAM supports Drag & Drop of files to make the process simpler.
The Input dialogue will automatically be set to the correct file type instead of you manually using the drop down option as above.

If the model is displayed as wire frame, then **VIEW > Shading**  is turned off.

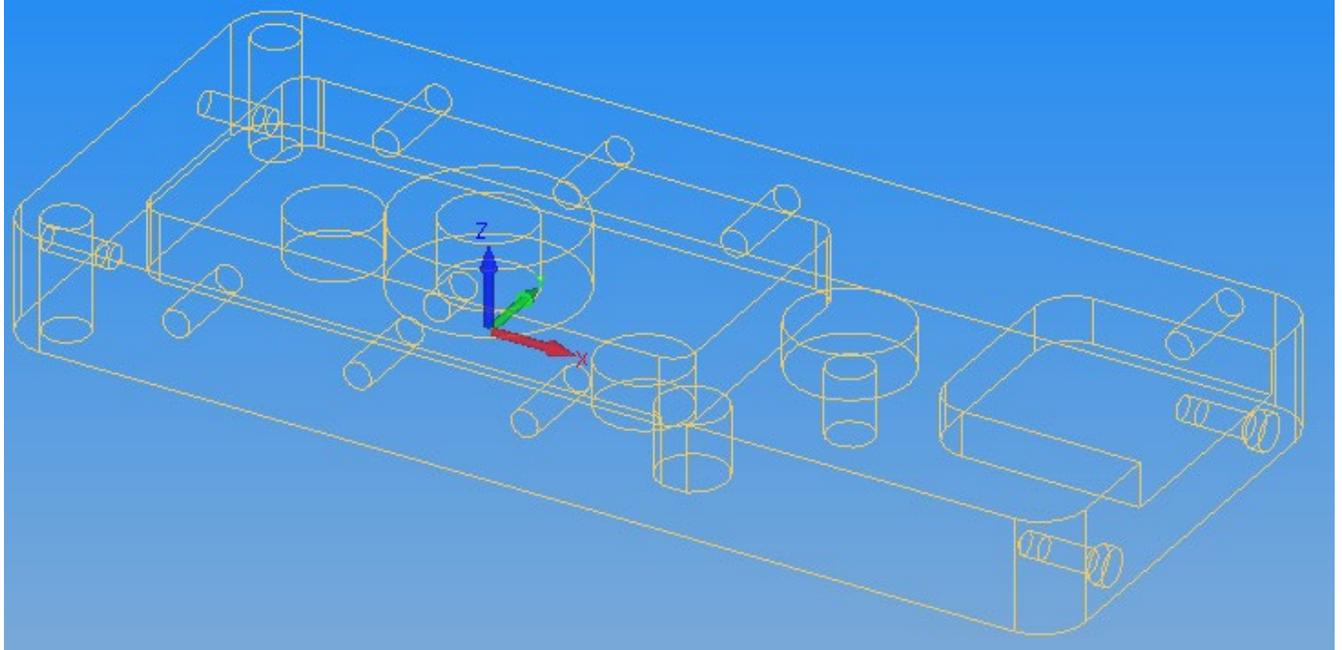


Figure 20 - Model Shown Unshaded

If the model is displayed as a solid, then **VIEW > Shading**  is on

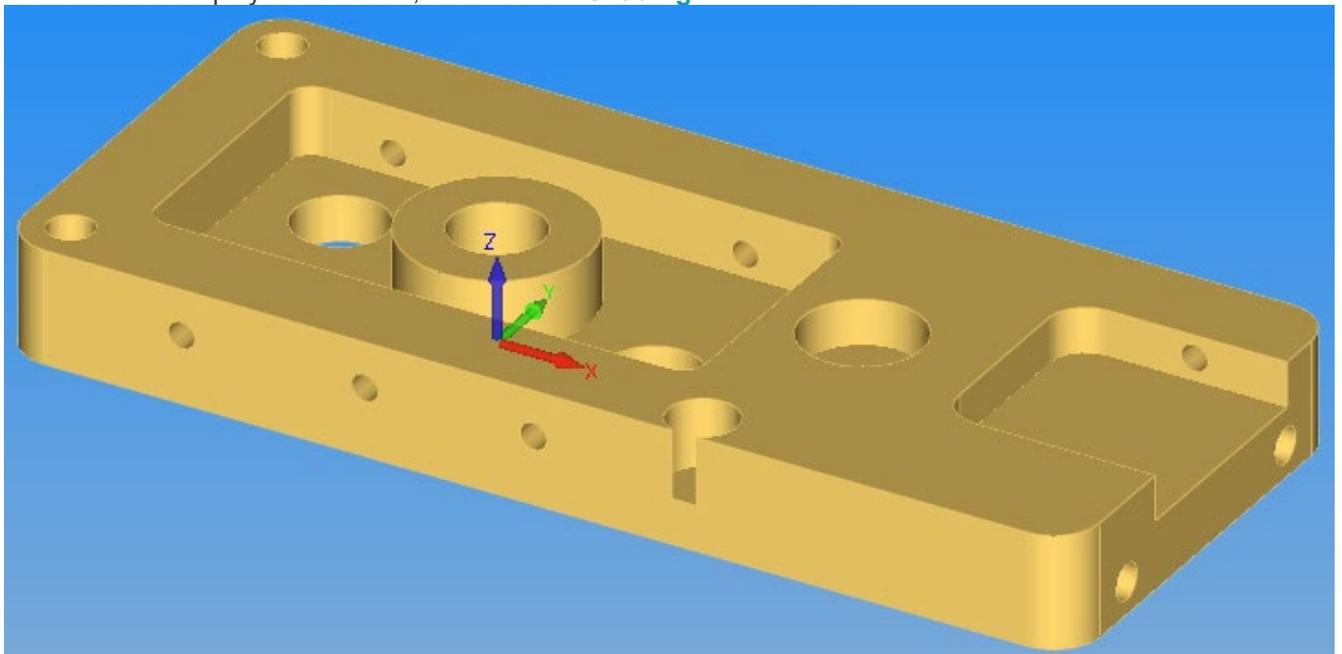


Figure 21 - Model Shown Shaded

The position of the model will depend upon the settings in the feature extraction configuration options. Remember if **Align on Import** is set not active, the part will be imported as designed.

Align the Model

For this example, we will set the global datum to the top of the part with the XY datum at the lower left corner.

Select **SOLID MODEL UTILITIES > Auto-Align Part** 

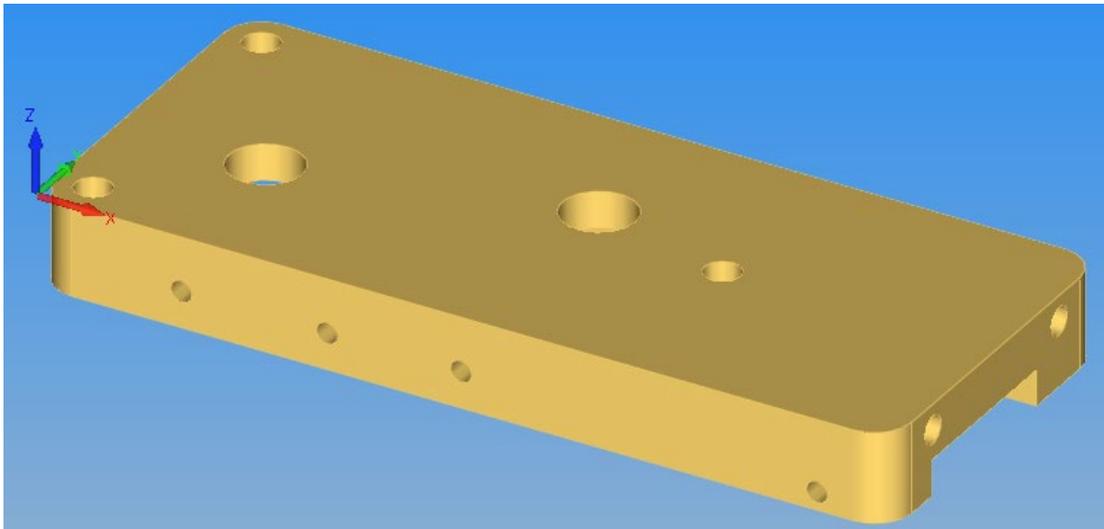


Figure 22 - Part with bottom face Auto-Aligned

If the part is repositioned upside down, select **SOLID MODEL UTILITIES > Auto-Align Part**  again and the part will be flipped over.

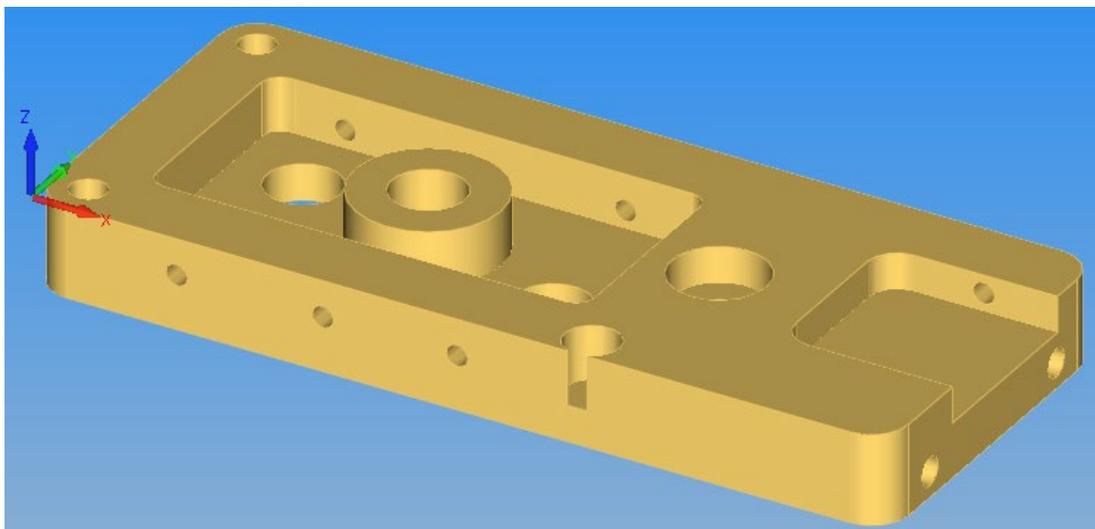


Figure 23 - Part with top face Auto-Aligned



It is worth noting here that **Auto-Align Part** will only flip the part over 180° along the X axis (in this case).



If the correct face is never displayed face up on the model you are working with, then you will need to use the manual **Align Part** option detailed later in this manual.

Extract the Features, Top Face only.

To extract the features from the model,

Select **SOLID MODEL EXTRACT > Automatic Extraction** 
The options dialogues are displayed.

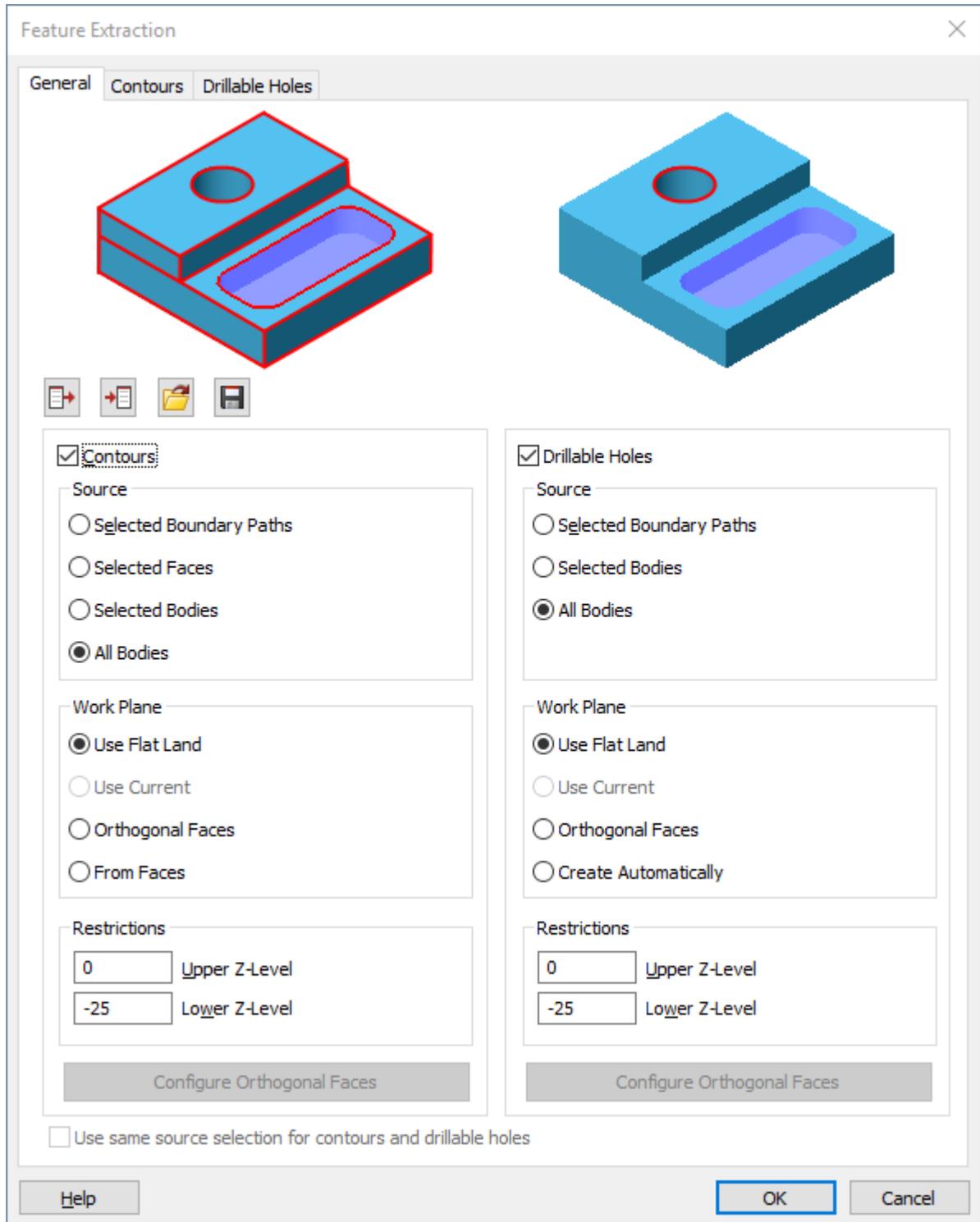


Figure 24 - Automatic Extraction General tab

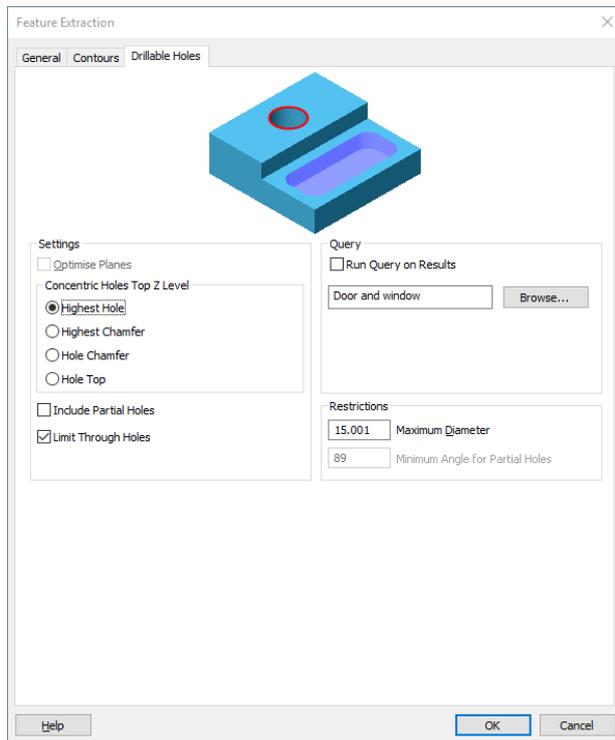


Figure 25 - Automatic Extraction Drillable Holes tab

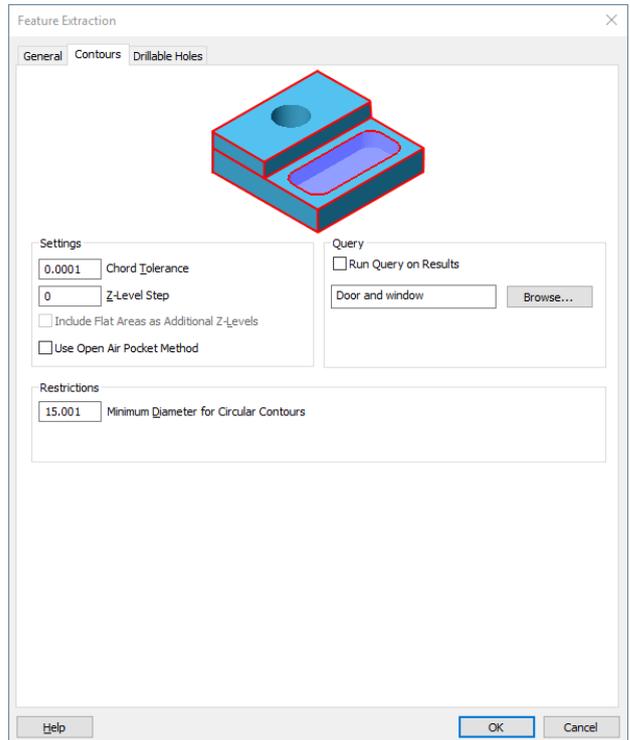


Figure 26 - Automatic Extraction Contours tab

Set the options as shown on each tab, then **<LClick> [OK]**.

The progress bar is displayed across the middle of the screen and closes automatically when the feature extraction is complete.

The 2D features will be extracted, holes will be grouped together by size in one of the Holes layers depending on their type, closed contours are stored in the **Closed Contours** layer, and any open contours will be stored in the **Open Contours** layer.

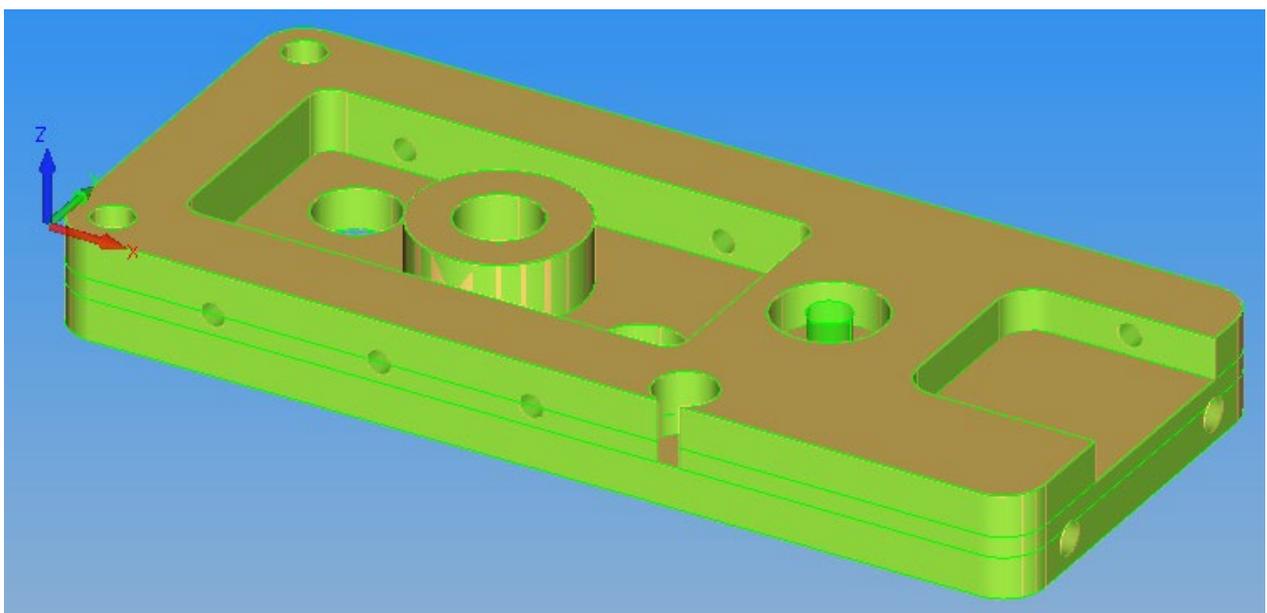
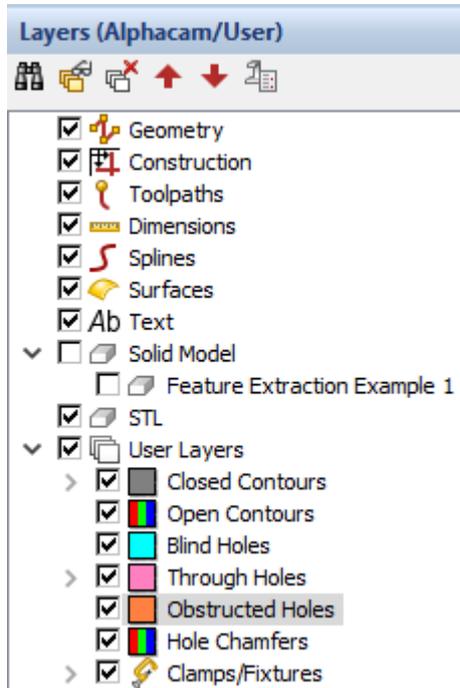


Figure 27 - Features extracted and highlighted for ease of identification



If you turn off the solid display you will be able to see the geometries extracted more clearly.

To turn off the solid uncheck the **Solid Model** in the **Solid Model** Layer in the Layers Project Manager page.

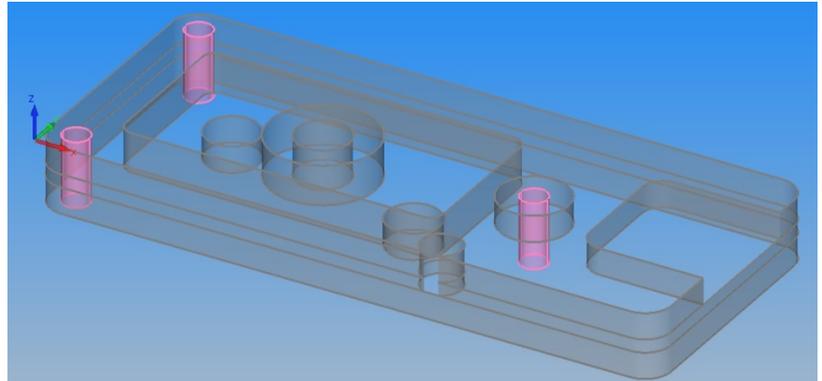


Figure 28 - Layer assignment for extracted items

You will see that the extracted geometries have their top and bottom Z values automatically assigned. From the colours, you can see the geometry profiles that have been extracted. The process has extracted the edges that created closed profiles.

You can also see that the holes on the side faces have not been extracted.

If you change the Automatic extraction parameters, it will change the geometries that will be extracted. Sometimes it may be necessary to run the automatic extraction more than once with different parameters.



Note that the colours of the Layers have been altered manually by editing the options and turning off the **Default** tick box, then choosing a different colour. This is just to illustrate the differing items within this section of the manual.

Visibility of extracted geometries

Two options you may consider are making the extracted sections as visible in the previous solid image where they are shown in the dominant green colour.

This is set using the **VIEW > Display**

Options  and making the choices from

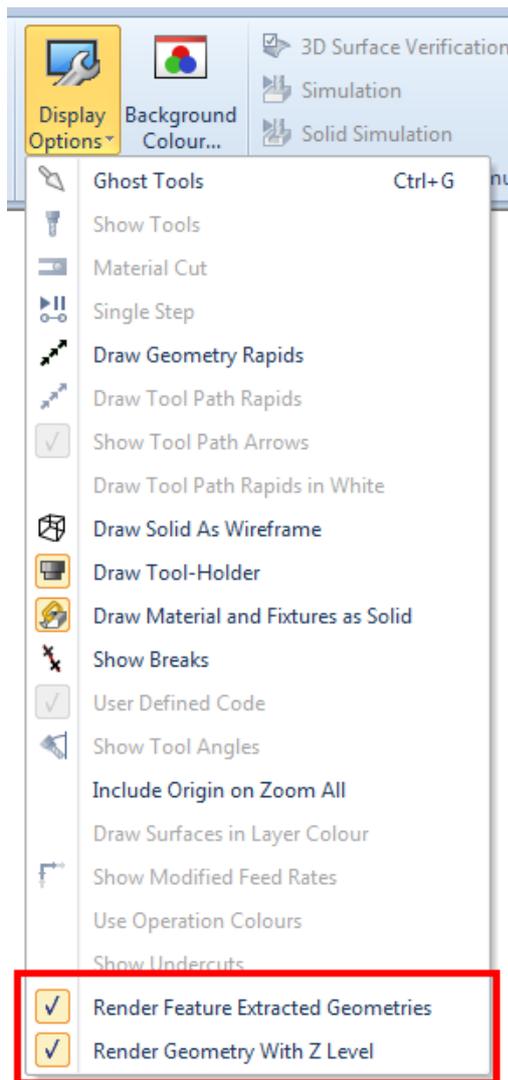


Figure 29 - Display options to see extracted items rendered or not

Using **HOME > Configure > General** 

On the Geometry tab, using the controls below.

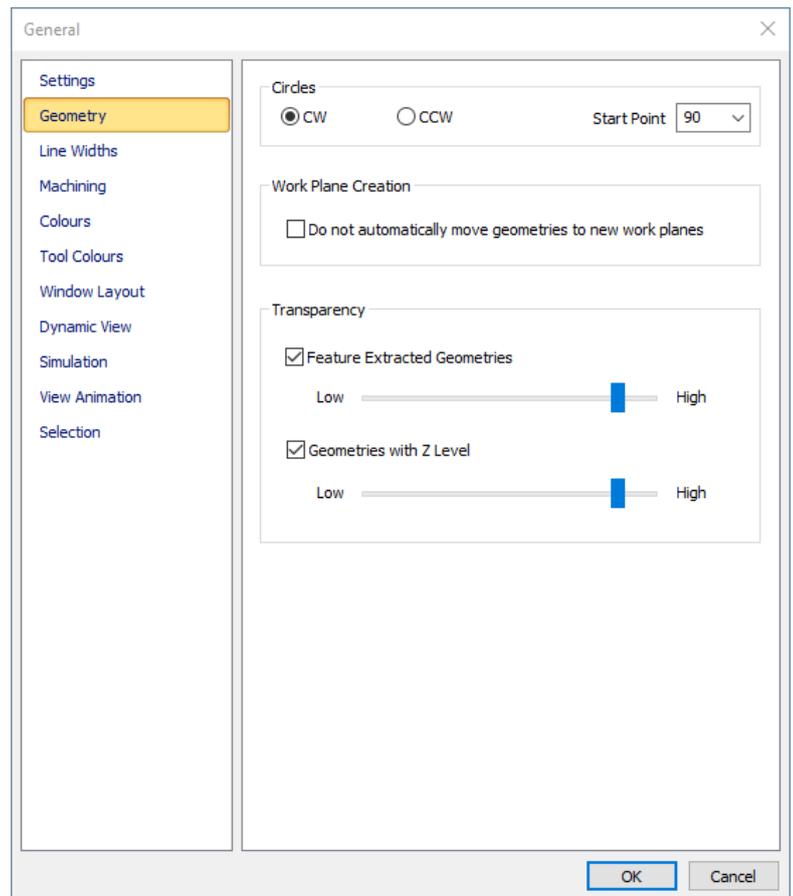


Figure 30 - Configuration slide bars for rendered transparency

Using the tick box allows for the option to be on or off or the slide bars to adjust the transparency settings.

Extract Side Holes on Front Face.

For this method, we will need to create a suitable Work Plane on the model.

Select **WORK PLANE > From Solid Model Face** 
<LClick> on the face shown.

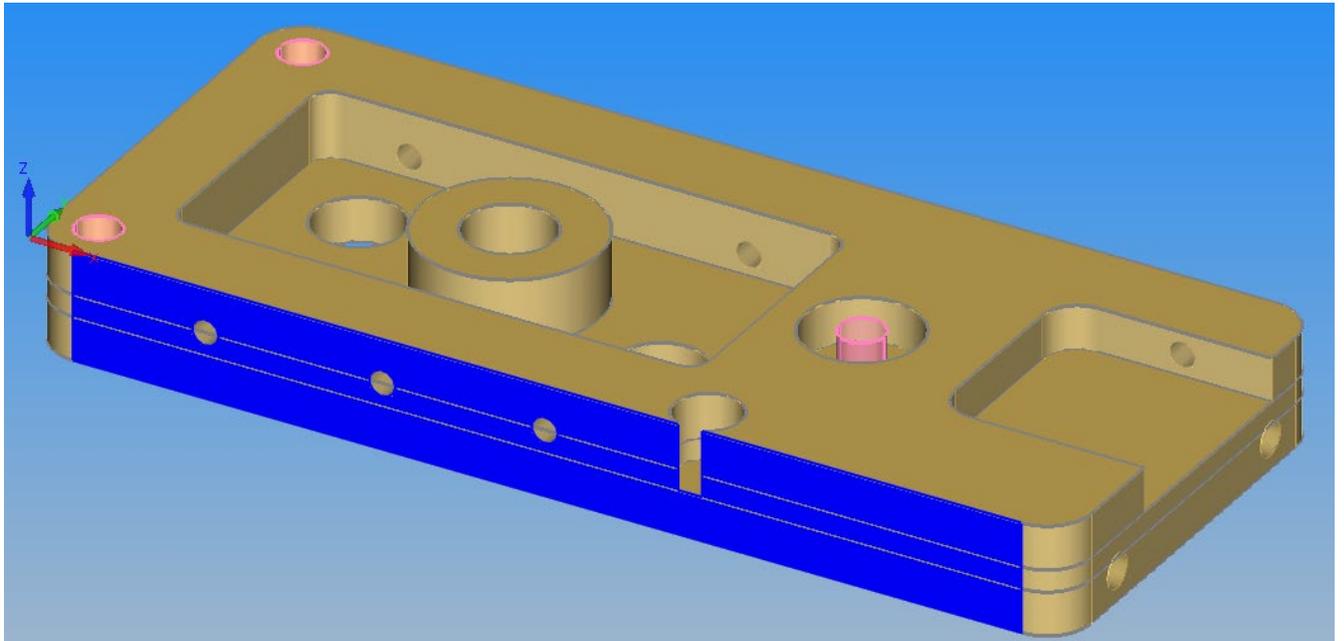


Figure 31 - Creating the Work Plane for one set of side holes

<RClick> to complete the process.

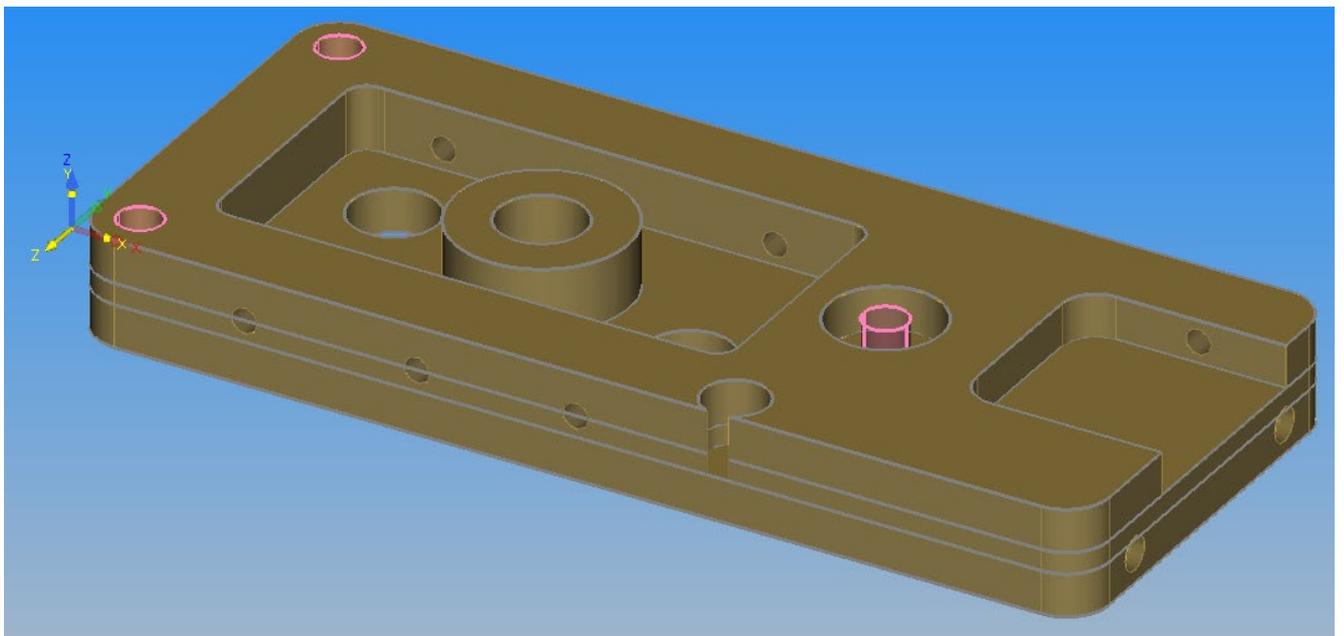


Figure 32 - Work Plane creation on first side detail

Select **SOLID MODEL EXTRACT > Automatic Extraction** 
The options dialogues are displayed.
Selecting only **Drillable Holes** this time and make the options as shown.

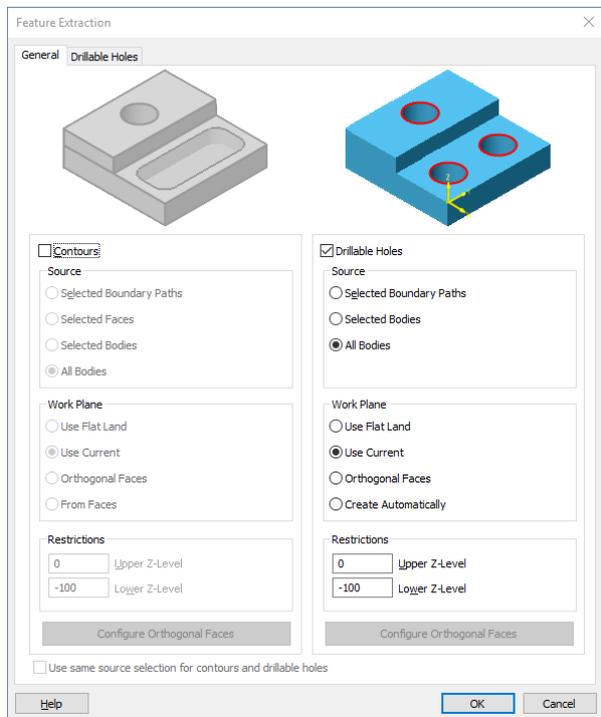


Figure 33 - Feature Extraction Contours turned off

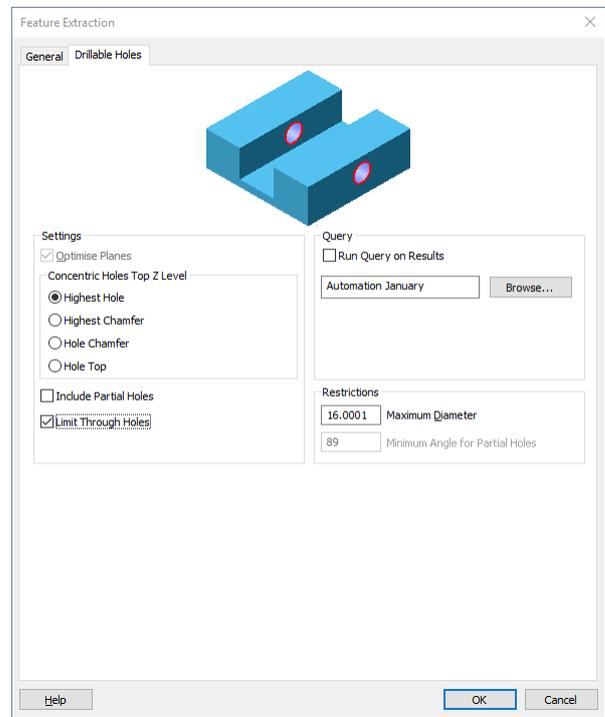


Figure 34 - Drillable Holes set to Limit Through Holes

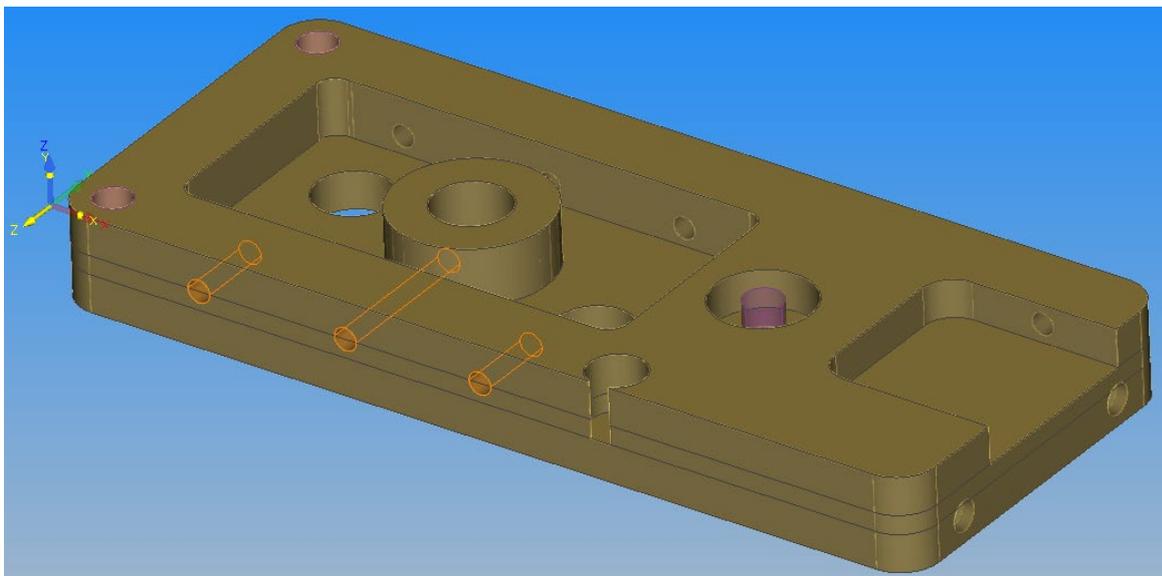


Figure 35 - Side holes extracted

Using the **Limit Through Holes** setting prevents the pair of shorter holes seen in the image from being extracted all the way across the part, as they share a common centreline with the holes on the opposite face. This is very handy if a part is too large to drill all the way through from one side.

Extract Side Holes on Back Face.

For this method, we will need to create a suitable Work Plane on the model.

Select **WORK PLANE > From Solid Model Face** 
<LClick> on the face shown.

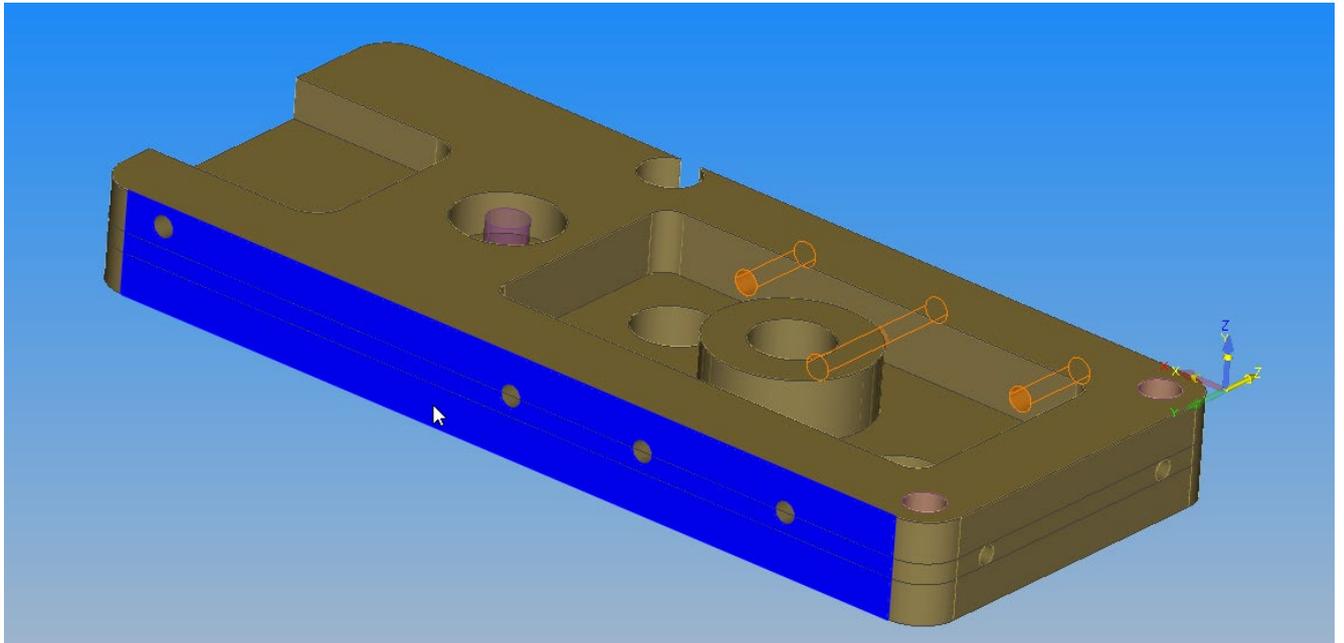


Figure 36 - Creating the Work Plane for the other set of side holes

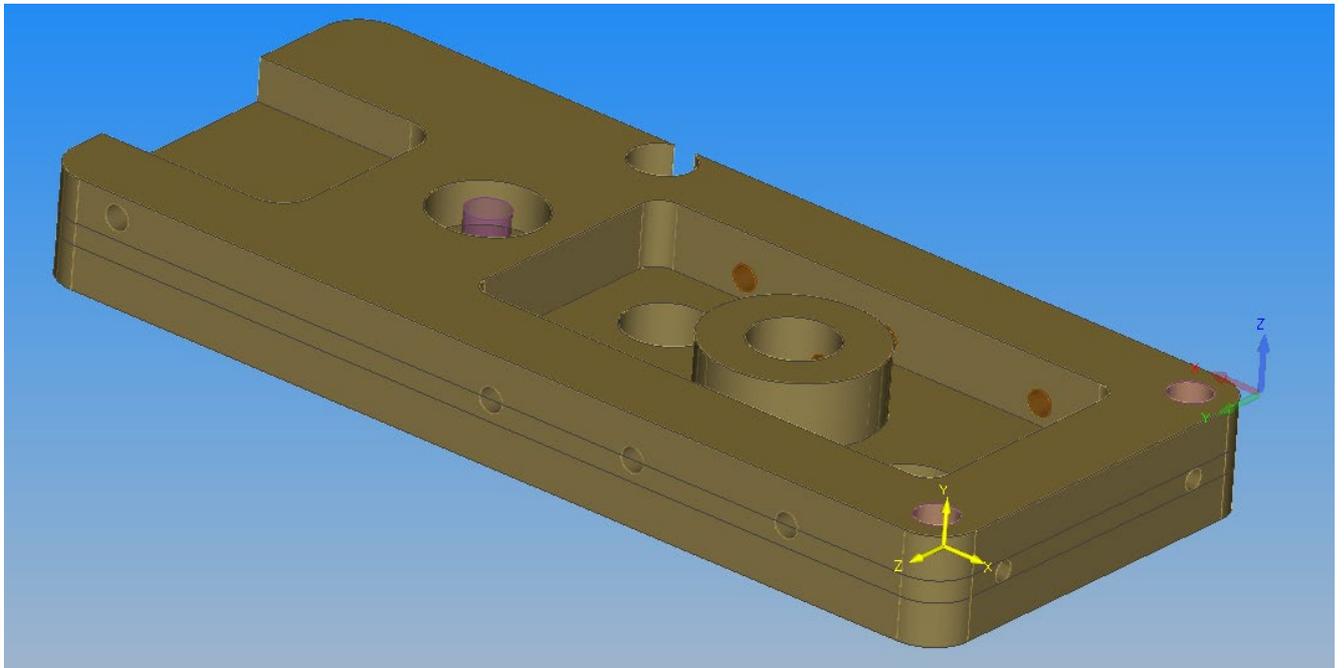


Figure 37 - Work Plane creation on second side detail

<RClick> to complete the process.

Adjust the orientation to make the selection easier select **VIEW > Horizontal Iso 1** 
With the work plane active, you can then extract the hole geometries one by one using

SOLID MODEL EXTRACT > Drillable Hole Extraction. 

Select the options as required for your processing from the dialogue box.

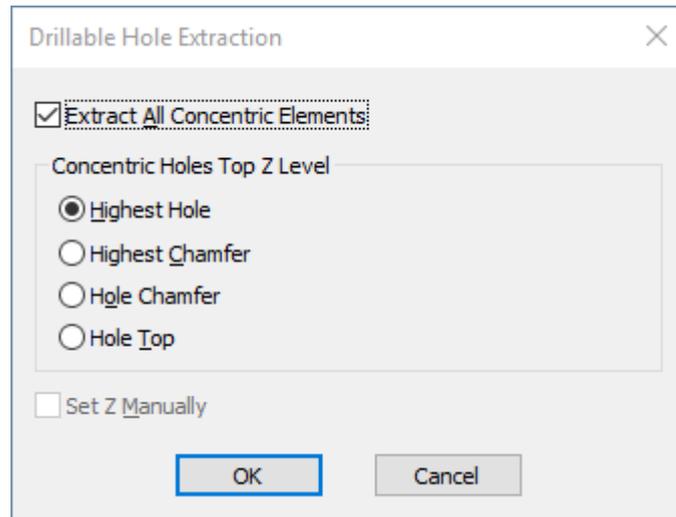


Figure 38 - Drillable Hole Extraction dialogue options

Then **<LClick>** one hole to select it for extraction, **<RClick>** to apply.

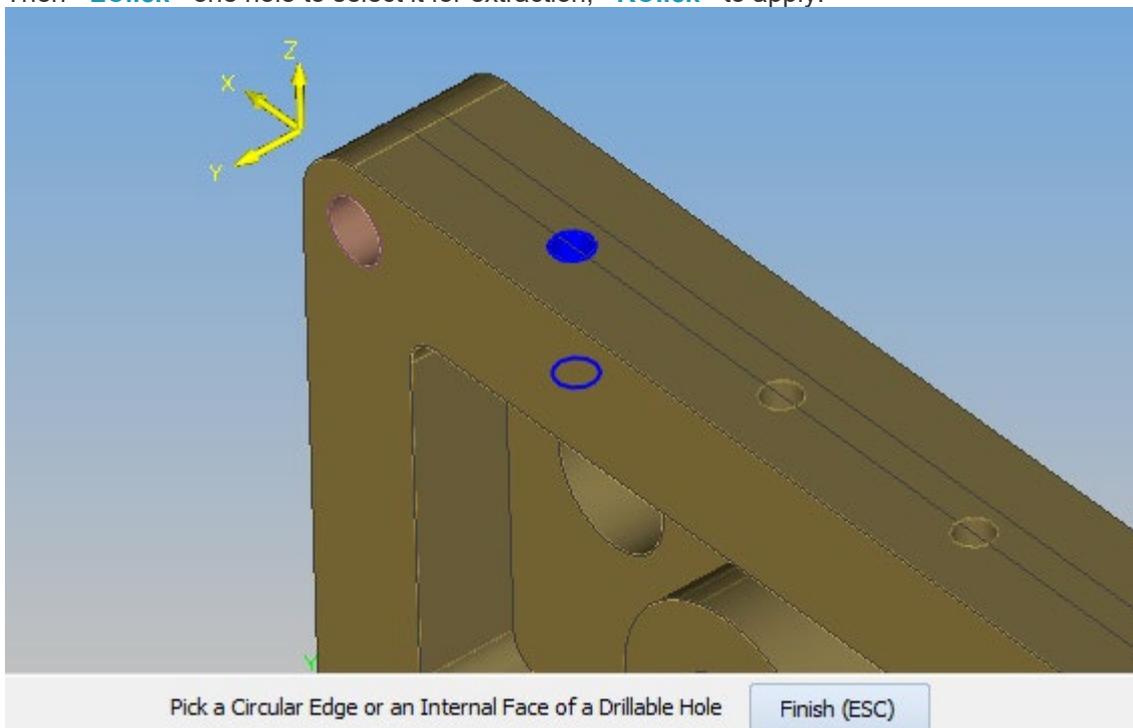


Figure 39 - Selecting the first hole to manually extract

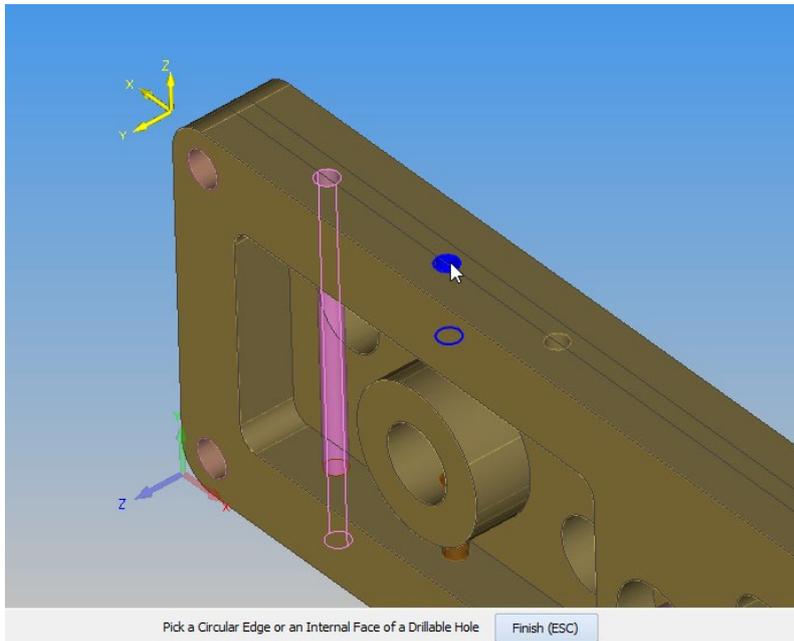


Figure 40 - First side hole extracted

Continue the process to complete the four holes on this side.

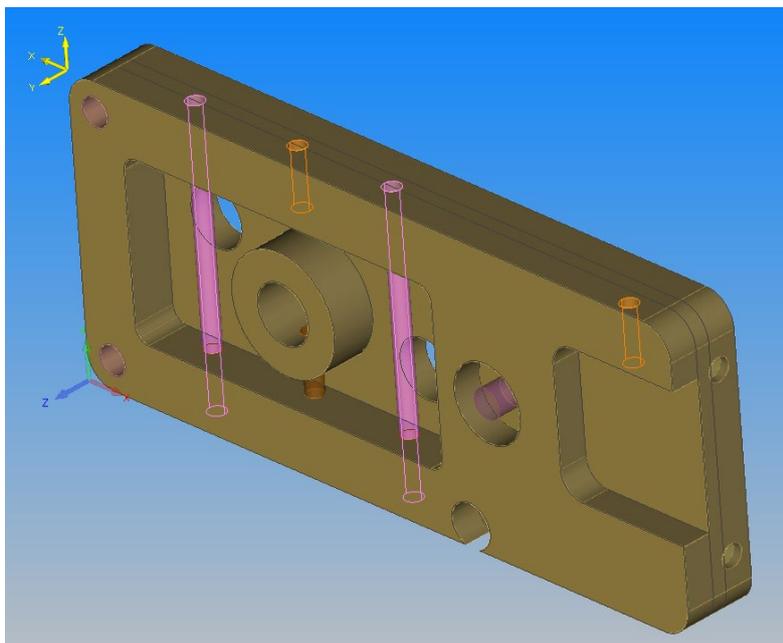


Figure 41 - Holes extracted manually

Note the differences between the holes and their layers, **Through Holes** and **Obstructed Holes** have been applied to the extracted items.



Note that using **Drillable Hole Extraction**, each hole must be selected as an individual item.

Reset the Geometry Z Levels

Because of the way in which the model was created the geometry for two of the second side holes extends through the part, so it will be necessary to adjust the bottom Z level of these two through holes so that during machining the part is only machined through one side wall and not the entire part.

Select **SOLID MODEL UTILITIES > Re-Set Geometry Z Levels** 
<LClick> the two holes on the back face as shown then <RClick>.

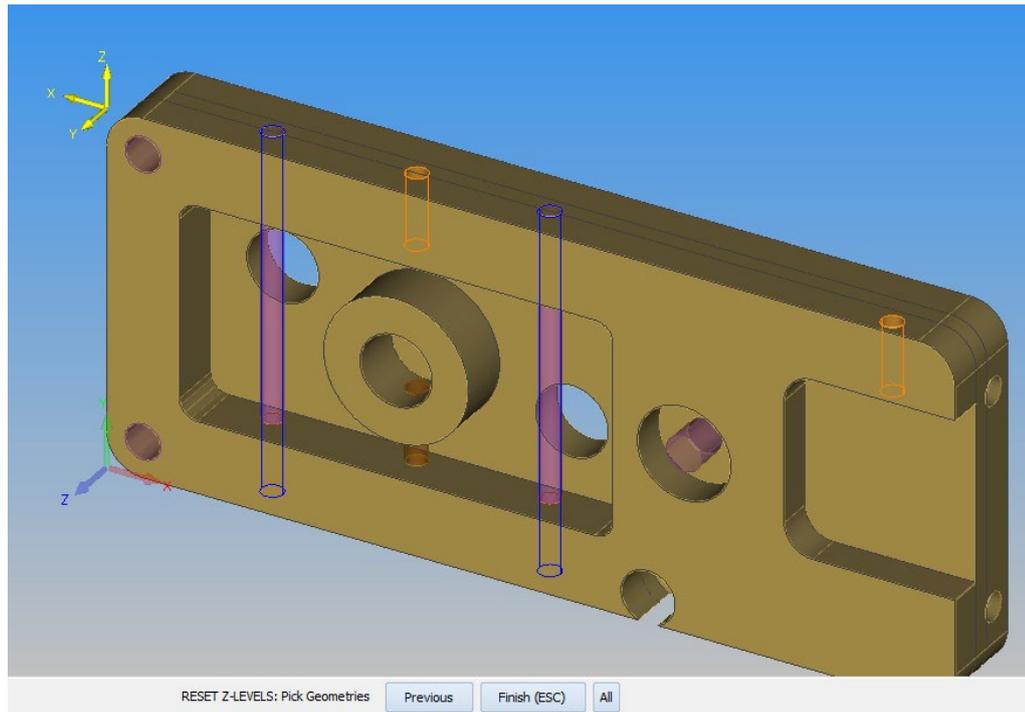


Figure 42 - Two through holes selected for Z level adjustment

Note that in this scenario where you have two sets of geometries from the opposite sides of the part, you are advised to either Hide the geometries on the first side or select the required geometries from their own face. The reason for this is that with there being two geometries on the first side, you may pick the wrong one.

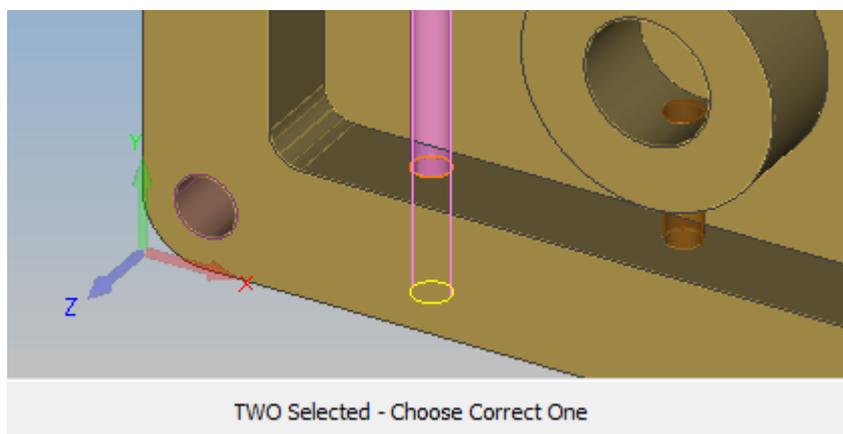


Figure 43 - Overlaid geometry error when selecting

Depending on the nature of the model in question, you may have to adjust the view to see where a specific geometry actually exits into another area.

Whilst you are in this command, the <LClick> options of the mouse to allow you to rotate the part view, are disabled.

You will need to use the cursor keys on the keyboard to view the inside of the part.

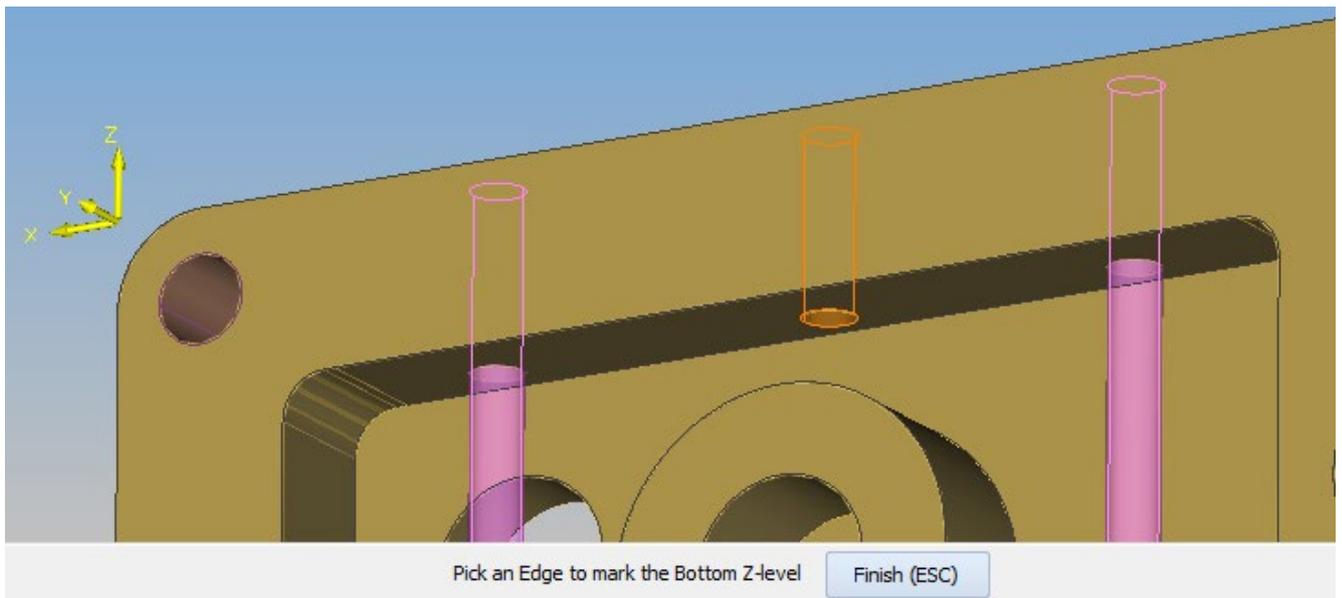


Figure 44 - View altered to see inside the pocket detail

In the case of this model, the walls are vertical inside the pocket, so the bottom edge, hole exit point and top edge are all in the same plane based on the current active work plane.

In this example, it does not matter which of these three edges we select, but in a case of a differing form it could have a serious impact on the geometries created.

 **Important Note**

 It Doesn't matter how many geometries you have chosen to reset, you **ONLY** select a single edge for the bottom and a single edge for the top.

 The locations are applied to **ALL** selected geometries.

 ALPHACAM supports the use of a 3D mouse, in this type of work in can prove more functional to use than the cursor keys.

Select the any edge on the inside of the pocket to set the bottom Z level as shown then **<RClick>**.

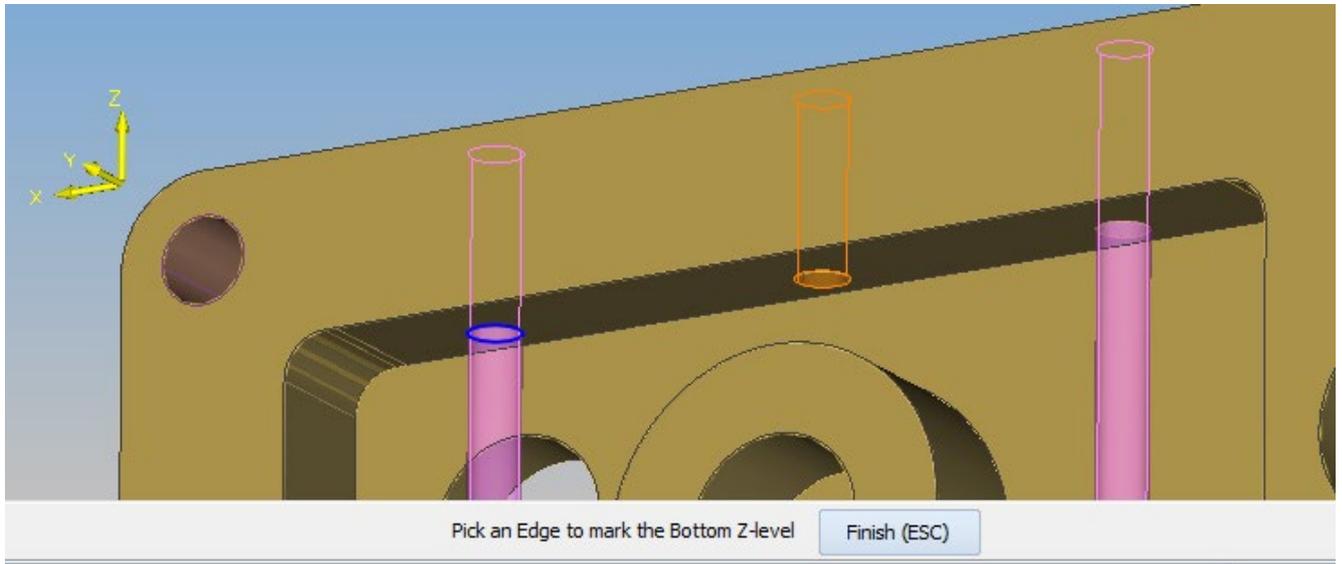


Figure 45 - Selecting the physical edge as the new Bottom Z level

You can select the top edge for the Top Z level then **<RClick>** to reset the holes Z levels.

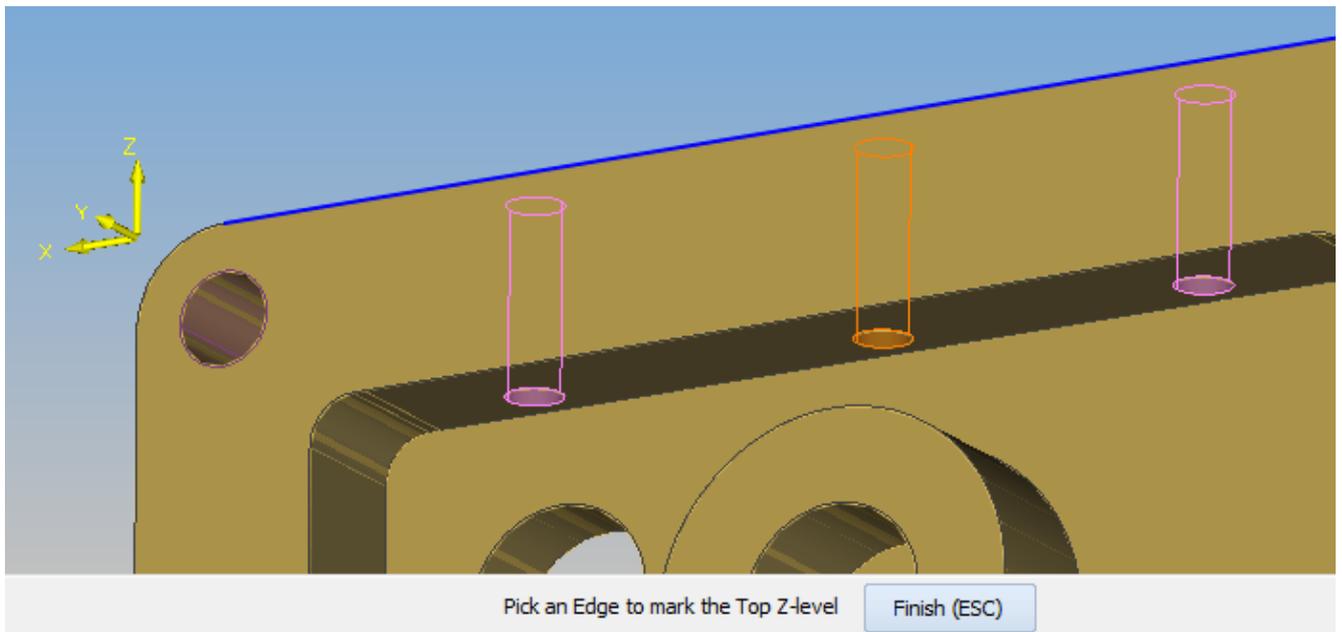


Figure 46 - Selecting the physical edge as the new Top Z level



If a geometry level is already showing as correct, clicking on the **[Finish (ESC)]** option will accept the current values without you needing to select an edge for reference.

#2 Using Orthogonal Faces

Import the model

Select **HOME > Input CAD**  and the following dialogue is displayed.

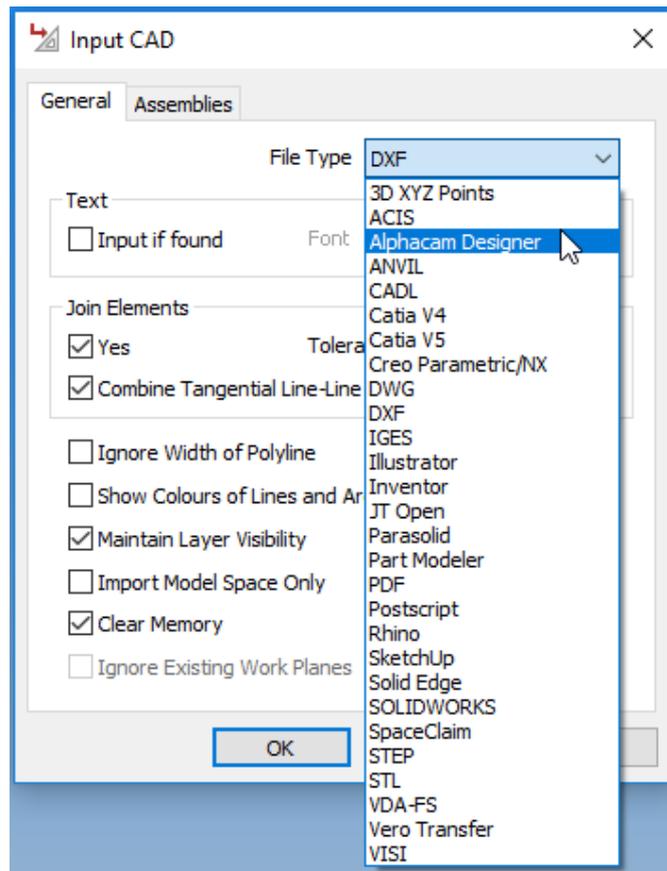


Figure 47 - input Cad dialogue set to ALPHACAM Designer

Set the file type to **Alphacam Designer**, **Clear Memory**, then select **[OK]**.

The file selection dialogue is displayed.

Navigate to the “.....\ALP TRG 114 Feature Extraction 2020\Examples\Alphacam Designer Examples” folder and select the “#1 Feature Extraction Example 1.vdf” then select **[Open]**.

The model is imported and displayed on screen.

Drag & Drop

Note that if you are able to access the required folder of the model via File Explorer, ALPHACAM supports Drag & Drop of files to make the process simpler.

The Input dialogue will automatically be set to the correct file type instead of you manually using the drop down option as above.

Align the Model

For this example, we will set the global datum to the top of the part with the XY datum at the lower left corner.

Select **SOLID MODEL UTILITIES > Auto Align Part** 

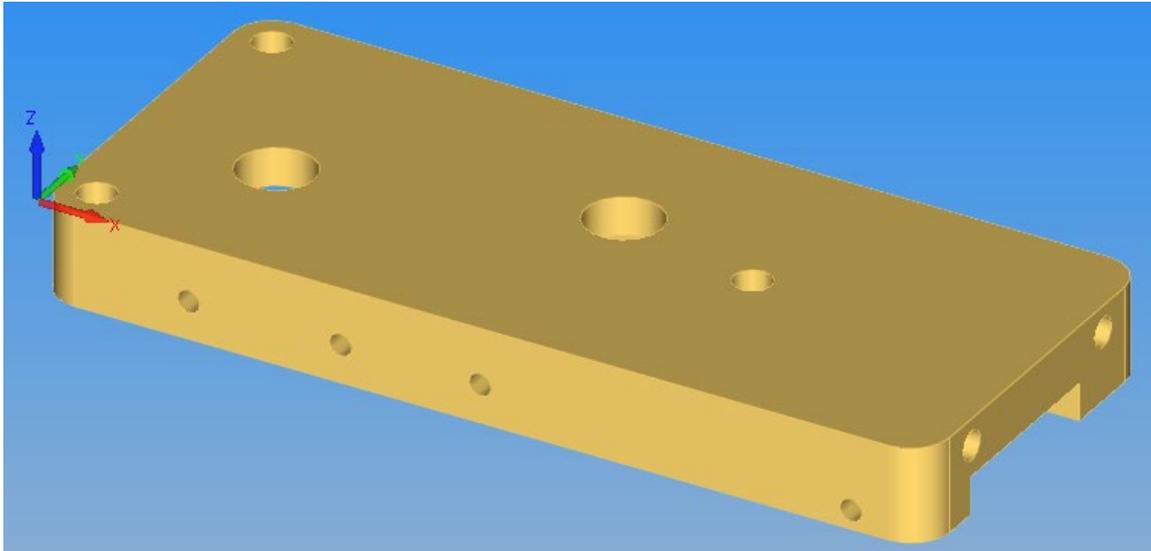


Figure 48 - Part with bottom face Auto-Aligned

If the part is repositioned upside down, select **SOLID MODEL UTILITIES > Auto Align Part**  again and the part will be flipped over.

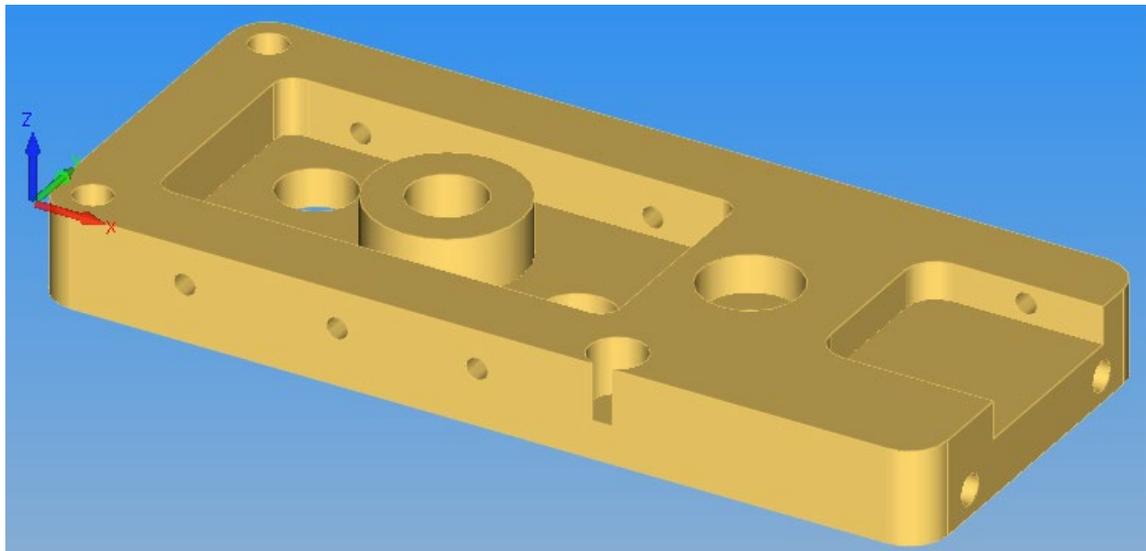


Figure 49 - Part with top face Auto-Aligned



It is worth noting here that Auto Align will only flip the part over 180° along the X axis (in this case).

If the correct face is never displayed face up on the model you are working with, then you will need to use the manual alignment option detailed later in this manual

Extract the Features, All Faces.

To extract the features from the model,

Select **SOLID MODEL EXTRACT > Automatic Extraction**



The options dialogues are displayed. Make sure you have the options set as shown.

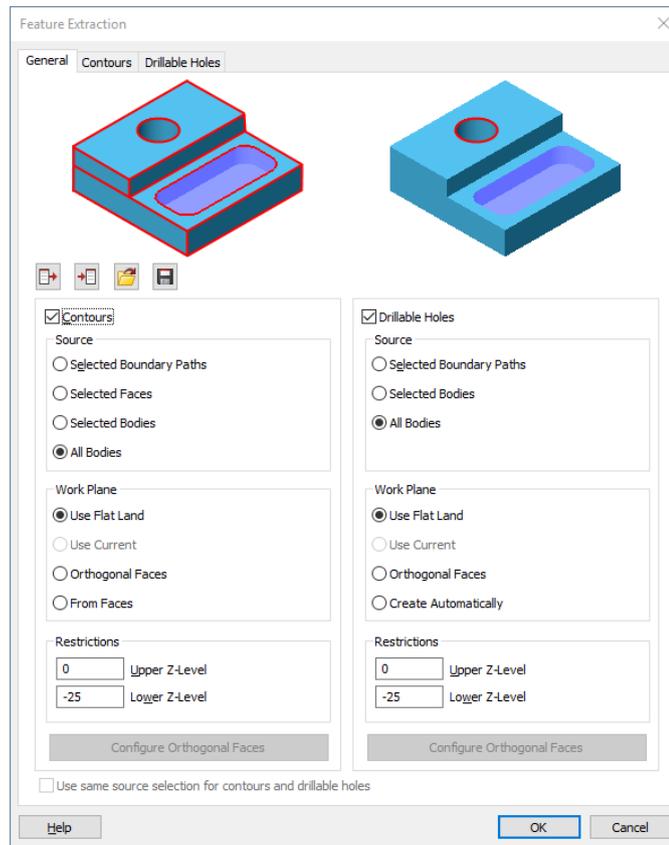


Figure 50 - Automatic Extraction General tab with Orthogonal faces active

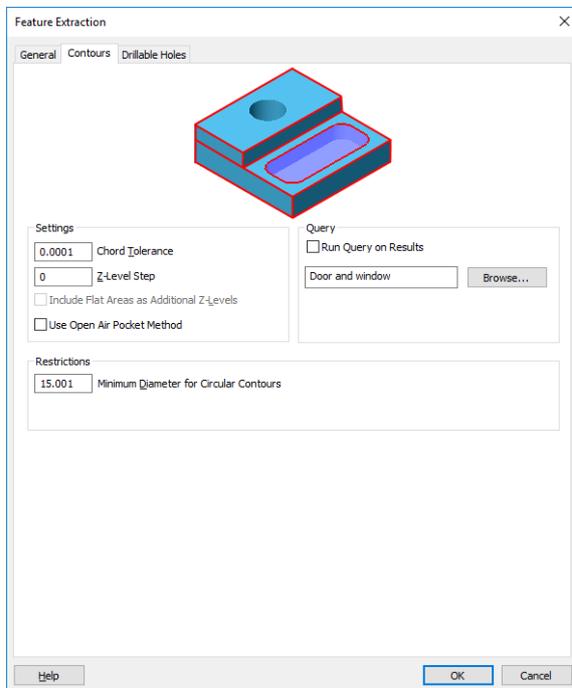


Figure 51 - Contours tab options

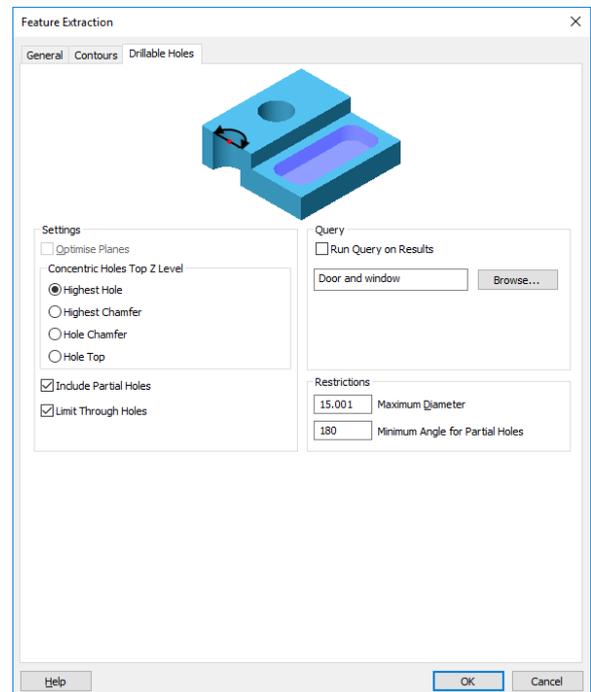


Figure 52 - Drillable Holes tab with Limit Through Holes active

Set the options as shown on each tab.

<LClick> on the **Configure Orthogonal Faces** button and set the options for **Contours** and **Drillable** holes as shown.

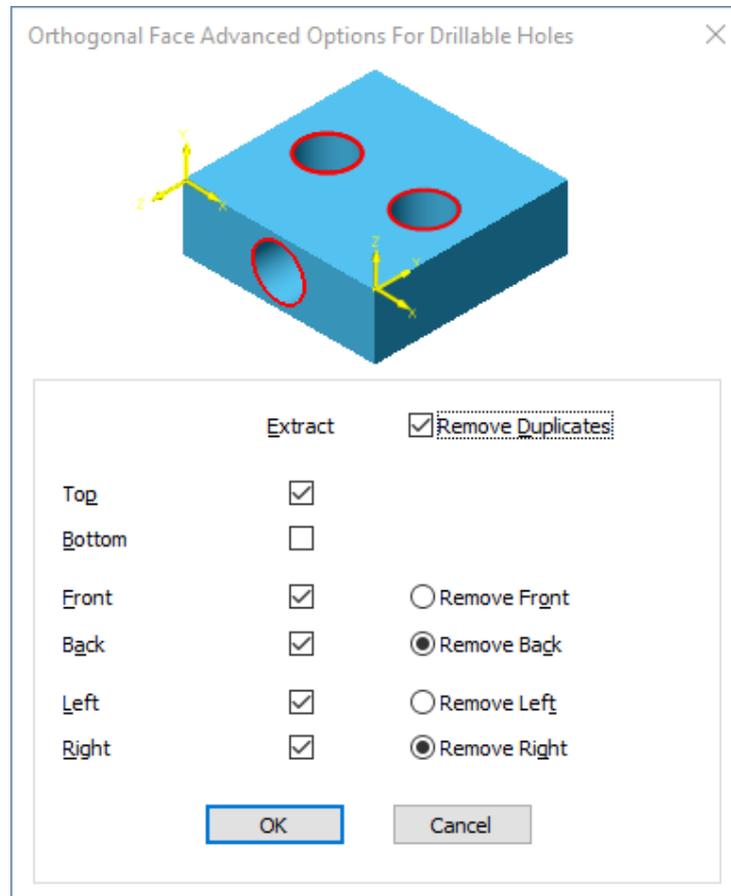


Figure 53 - Orthogonal Faces configuration dialogue options

<LClick> [OK].



Note that if you have the General dialogue option set to **Selected Bodies** the option **Use same source selection for contours and drillable holes** becomes available removing the need to configure the option twice as we have here.

<LClick> [OK].

The progress bar is displayed across the middle of the screen and closes automatically when the feature extraction is complete.

The 2D features will be extracted, holes will be grouped together by size in one of the holes layers depending on their type, closed contours are stored in the **Closed Contours** layer, and any open contours will be stored in the **Open Contours** layer.

All required Work Planes are generated for the four side faces and there is no requirement to reset any Z Levels as we used the **Limit Through Holes** option.

On simple parts such as this one, this method proves the most efficient.

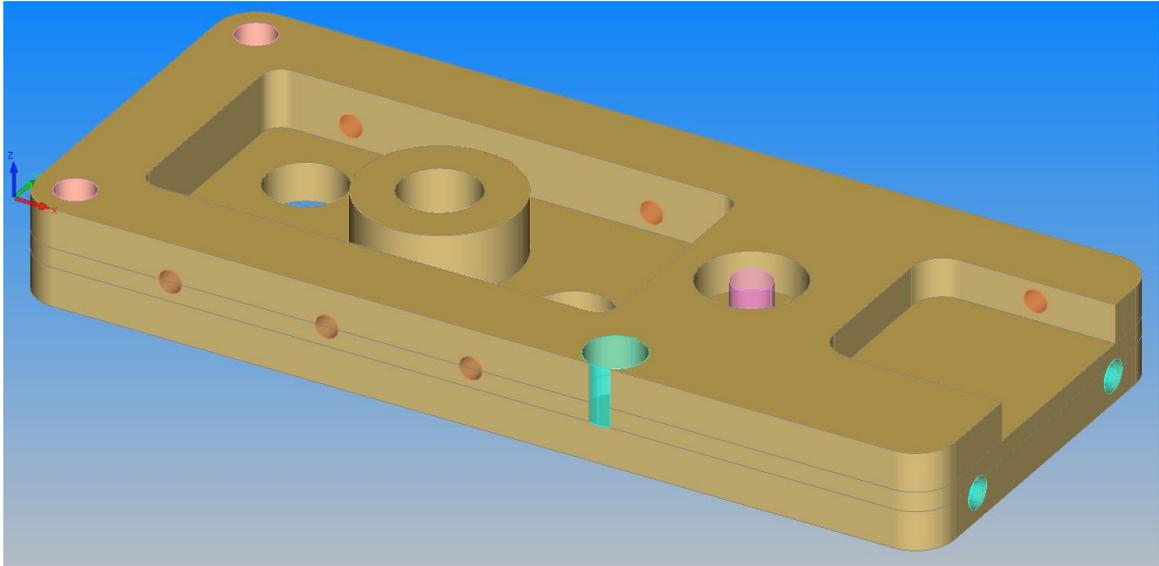


Figure 54 - All faces extracted shown Shaded

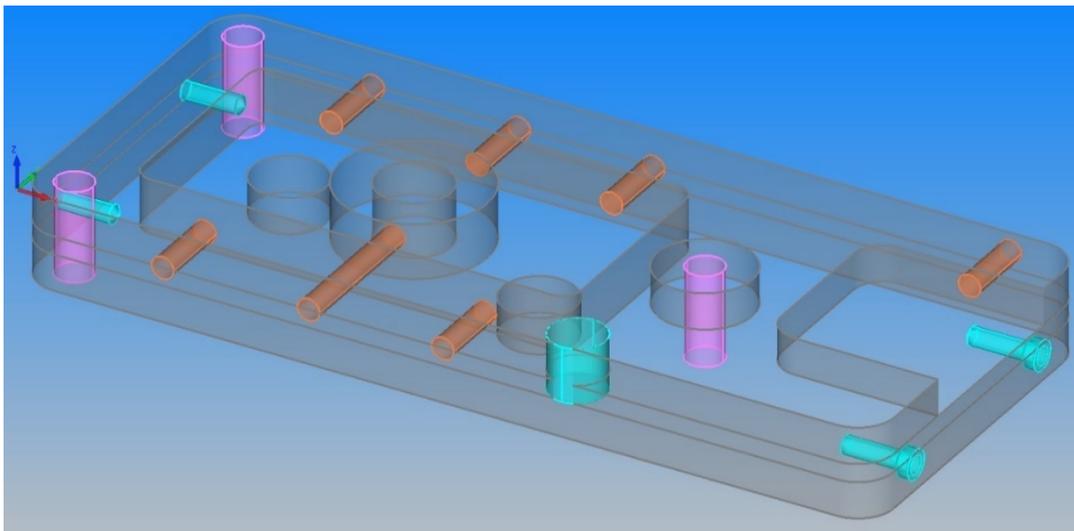


Figure 55 - All faces extracted shown not Shaded

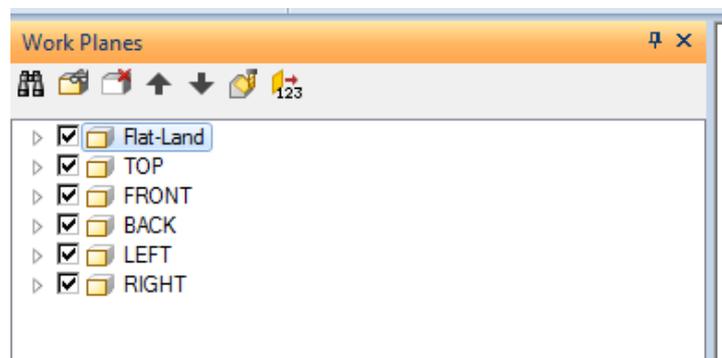


Figure 56 - Automatically created and renamed Work Plane

Also, the Work Planes created are named accordingly to the faces they represent. This can be an extremely useful method of using **Feature Extraction** if your parts are predominantly panel style items working on these side faces.

Prepare the Geometries for Machining

Turn off the **Solid Model** in the Layers Project manager page.



Figure 57 - Solid Model hidden for clarity

Check Tool Directions

Set the view to be XY using **VIEW > XY** 

Turn on the Ghost tools  **<Ctrl> + <G>**

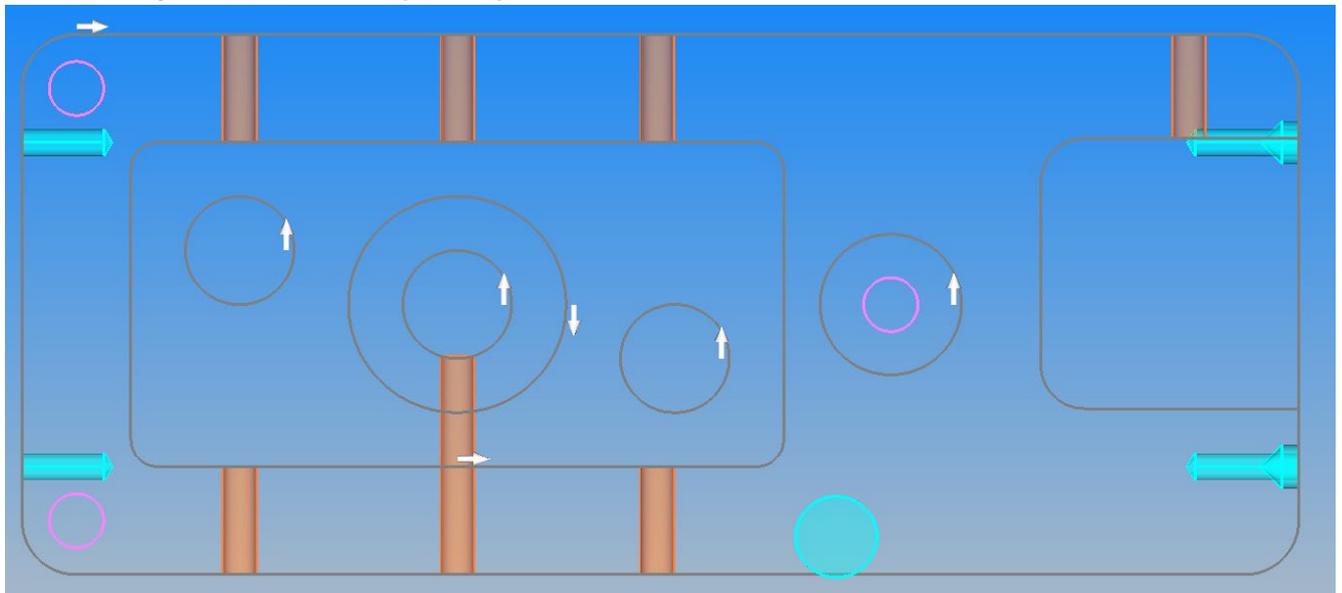


Figure 58 - Ghost Tools visible for start point locations

Check the Start Points

You will notice that the Start point on the rectangular pocket is positioned at the midpoint of the longest side (as set in the Feature Extraction configuration); unfortunately, this is at the narrowest gap between the pocket wall and the central boss.

Possibly moving the start point on this geometry only will make machining the part less prone to errors depending on the style of Lead In/Out that you normally use.



It is worth spending a few moments on your Feature Extracted geometries to check the positions of the Start Point to save you altering them after adding the tool paths.



Remember, you have saved considerable time in geometry creation, a few minutes checking could be beneficial.

Set Start Point

Select **EDIT > Start Order > Start Pt**  then select the midpoint on the left or right side edge.

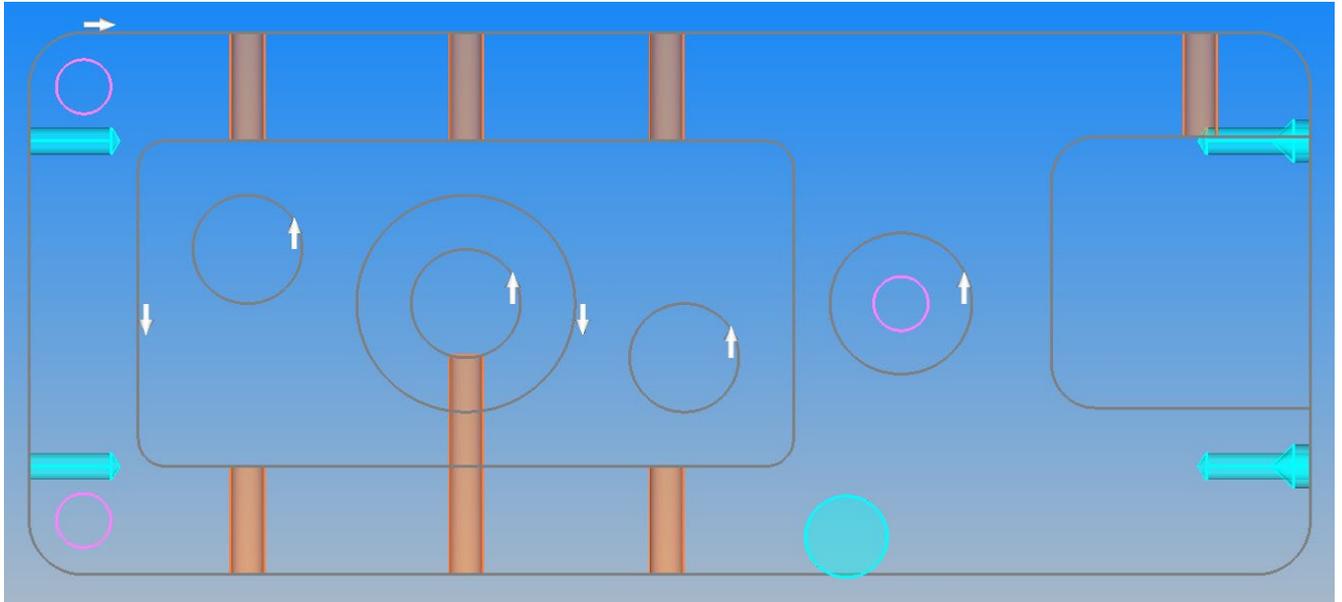


Figure 59 - Start Points altered to suit manufacture

Introduction to Auto Z Machining

When using Feature Extraction methods of geometry creation, the upper and lower Z values are assigned to the geometries using inbuilt attributes. This then allows you to use the Auto Z method of toolpath creation to generate the required cycle data for the part to be machined.

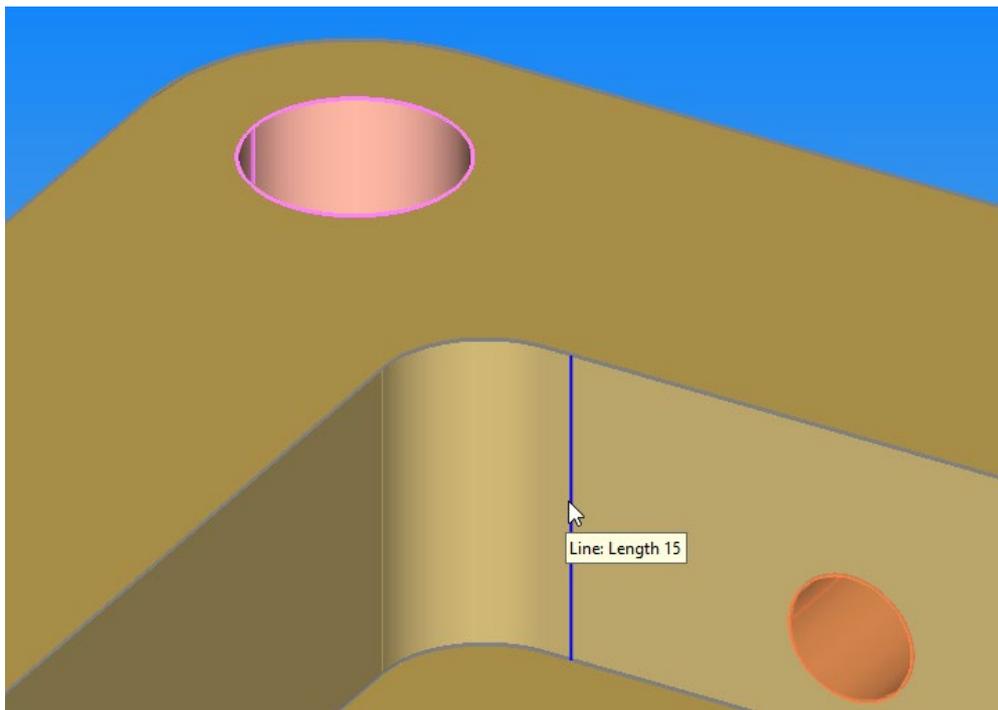
Face Edge Point details

Whilst there is no direct requirement to measure every single section of solid model prior to manufacture to allow you to enter the upper and lower Z values into the Levels and Cycle Data pages of the chosen machining cycle, you will still need to account for drill sizes, flute length and tool radius so that you can produce the part correctly.

Using **SOLID MODEL UTILITIES > Face/Edge/Point Details**



allows you to check how deep a pocket or profile is so that you use a tool with the correct length or to find the fillet radius in a corner to allow you to select the correct diameter tooling.



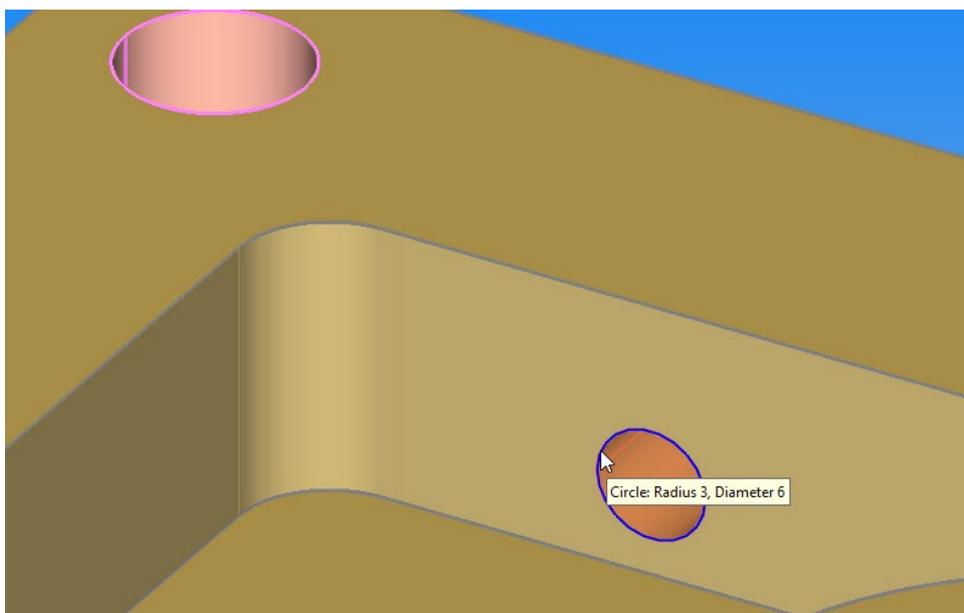
Hovering over a demarcation line to check its length for the required flute length

Figure 60 - Face/Edge/Point Details 1



Hovering over an internal fillet radius to see what size tool is required.

Figure 61 - Face/Edge/Point Details 2



Hovering over a hole edge or the internal face will give you the required diameter.

Figure 62 - Face/Edge/Point Details 3

Auto Z Differences

Cycle Selection

When accessing the required cycle options during toolpath creation for **Pocketing** and **Rough or Finish**, to use Auto Z methods you must alter the selection option from Vertical to Auto Z

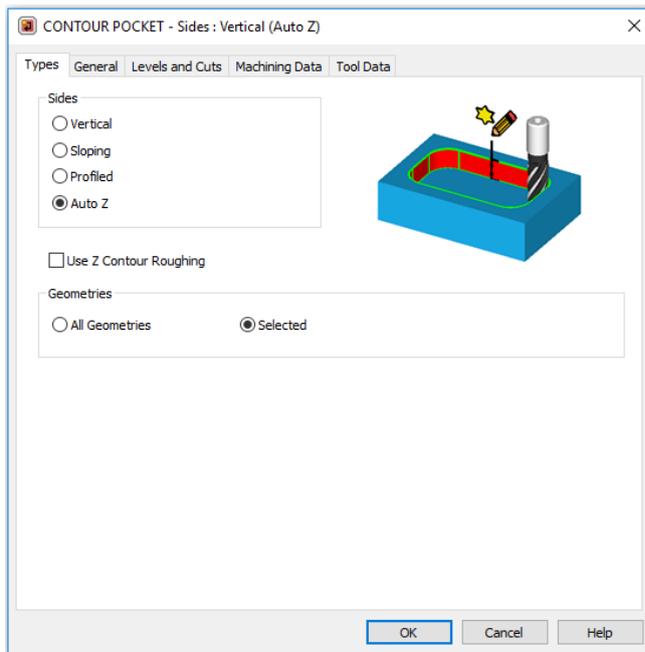


Figure 63 - Auto Z set in Pocketing

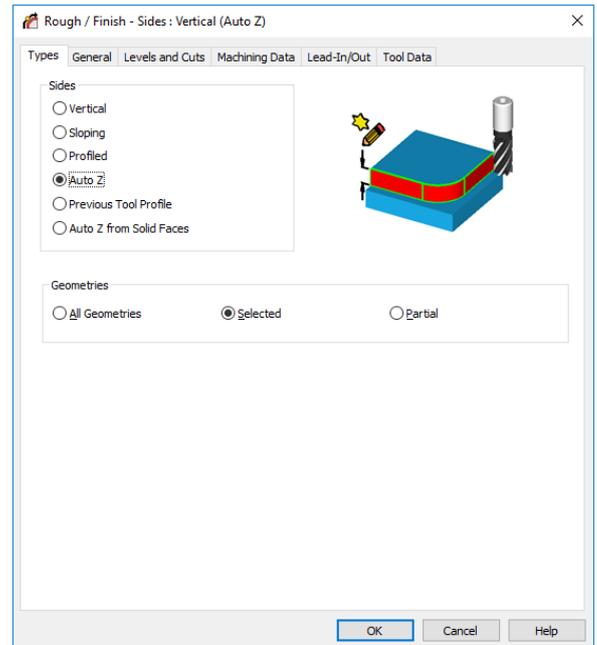


Figure 64 - Auto Z set in Rough or Finish

Drilling, however, is slightly different. Here you must select the correct cycle from the ribbon or drop down menu depending on the level of the software in use.

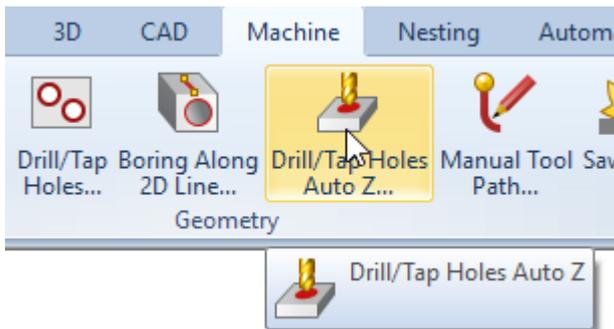


Figure 65 - Auto Z Drilling in Standard

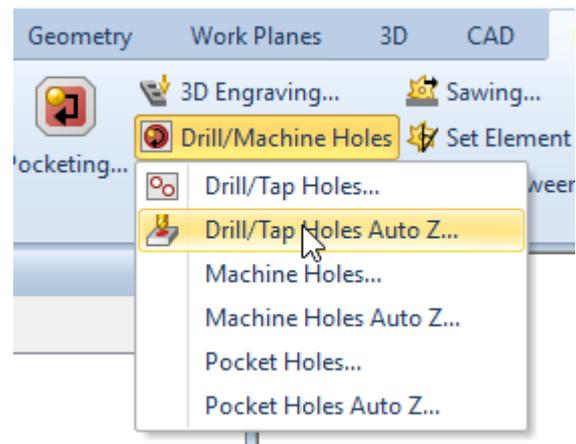


Figure 66 - Auto Z Drilling in Advanced and Ultimate

Levels and Cuts settings Pocketing

Compared to Fixed Z machining, where you must enter the values for Material Top and Z Final Depth on all cycles, Auto Z machining uses the extracted geometry values to calculate the toolpath Z values and as such these two boxes do not appear in any of the Auto Z manufacturing tabs.

In Pocketing, you have options for Max and Min Depths of cut to allow ALPHACAM to calculate how many steps down in the Z axis are required to reach the correct depth of the pocket.

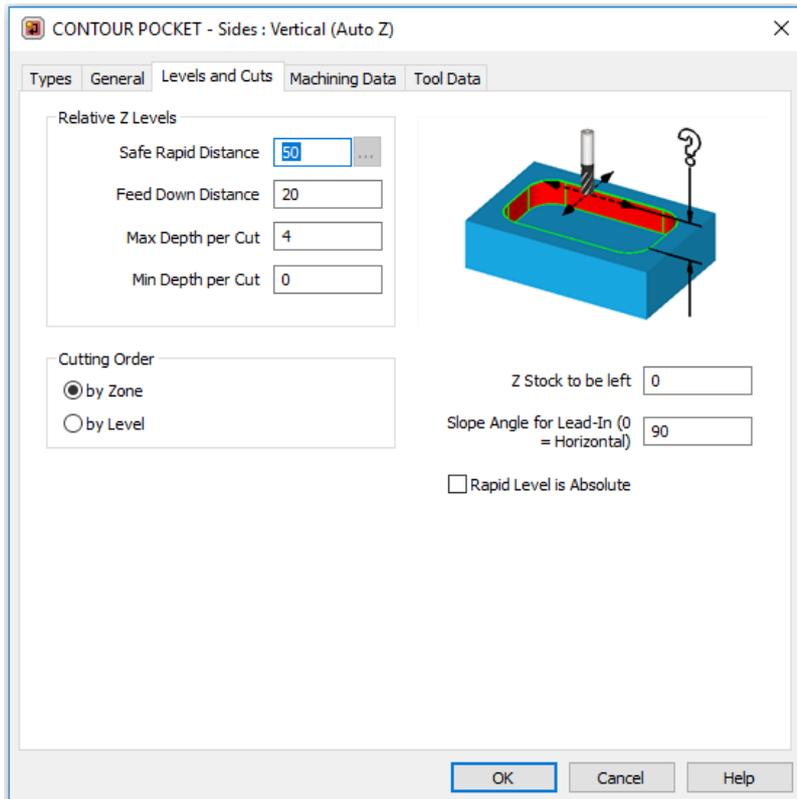


Figure 67 - Auto Z Pocketing Levels and Cuts tab

The Max Depth value can be automatically applied into a cycle by entering a value into the corresponding dialogue option when you define a tool.



Figure 68 - Depth of Cut option on Tool Definition

To further control the depth of the tooling, in Fixed Z you can manipulate the actual values, this is not the case in Auto Z machining.

Using the **Z Stock to be left** option will allow you to machine above the lower geometry value by entering a positive value or to go deeper by entering a negative value.



Note that the value is only the amount you wish to stand off or machine beyond by.

Levels and Cuts settings Rough or Finish

In Rough or Finish you still have control over the actual Z depth using the same **Z Stock Amount** option as used in the Pocketing cycle, but you also have an option to control the depth of each cut using the **Depth of Cut (0=Full)** option.

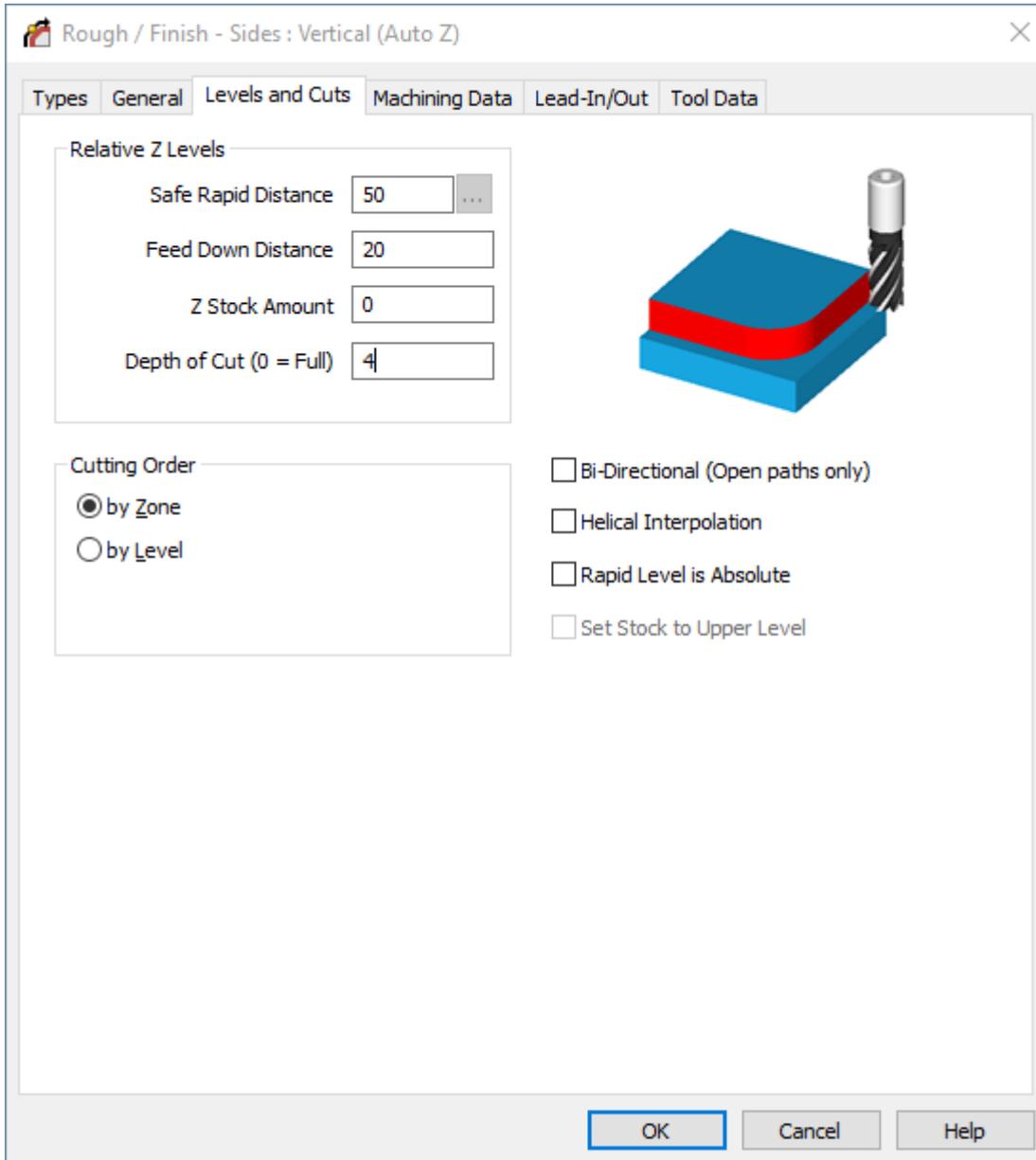


Figure 69 - Auto Z Rough or Finish Levels and Cuts tab

Note that the **Depth of Cut (0=Full)** is in millimetres **NOT** number of cuts.

In the image above the part would be machined in depths of 4mm in the Z axis, should you wish to machine the profile in a single pass, then enter a value of zero.

Levels and Cycle Data settings Drilling

The same manner of depth control is applied to **Drilling** as used in **Pocketing** and **Rough or Finish**. Using the Z Stock Amount allows you to drill shallow or deeper than required based on other option choices.

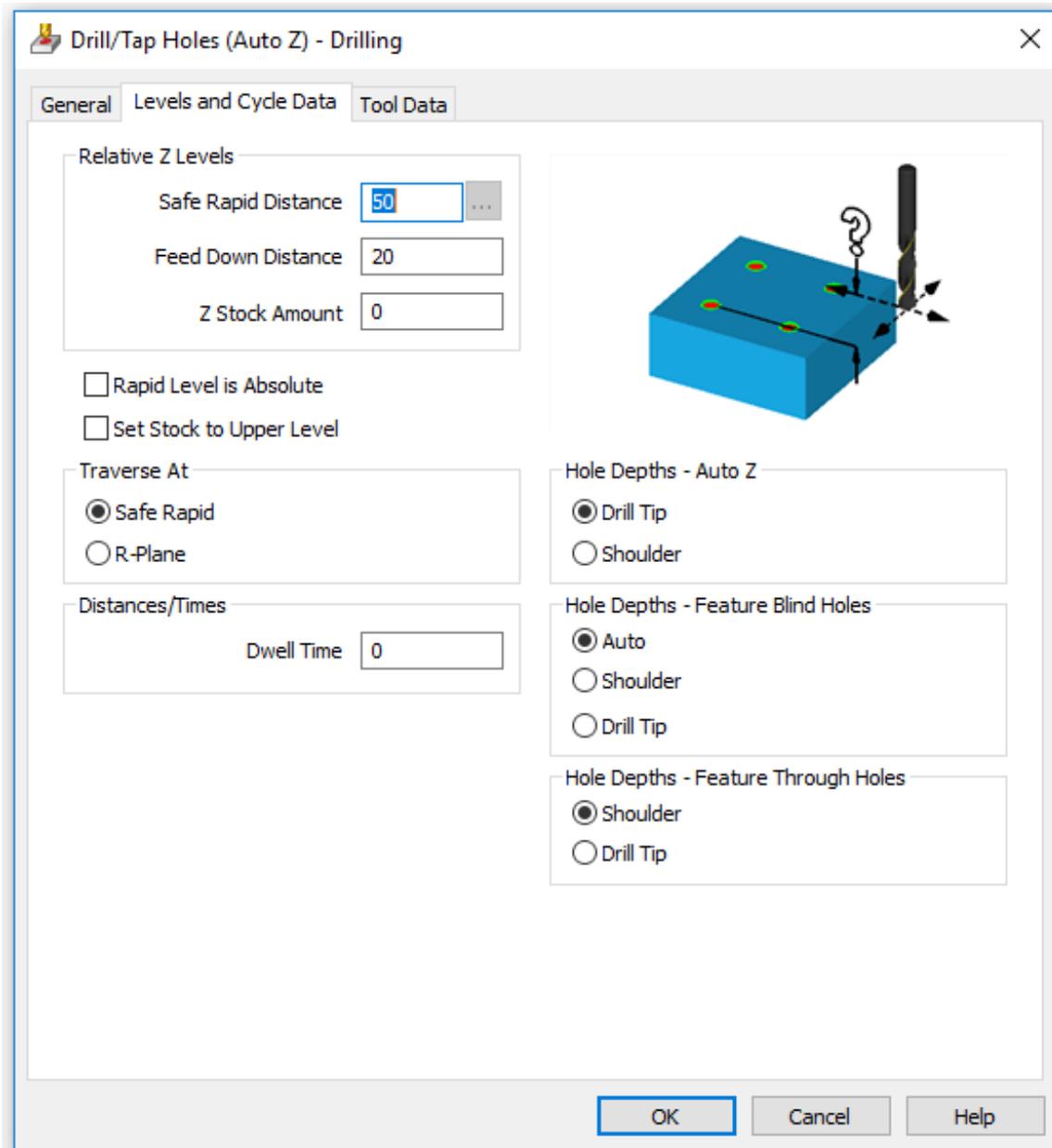


Figure 70 - Auto Z Drilling Levels and Cycle Data tab

Common options on all Drill/ Tap Holes Auto Z options.

Rapid Level is Absolute

This option will alter the **Feed Down Distance** and **Safe Rapid Distance** from being Relative to the current active work plane to be in relation to the Global Origin. This can allow extra clearance when working on multi plane parts.

Set Stock to Upper Level

During all the above cycles, the final depth of the hole has been based on the lower geometry Z value applied to the chosen geometry, selecting this option will now set the depth of the hole to be relevant to the higher geometry instead.

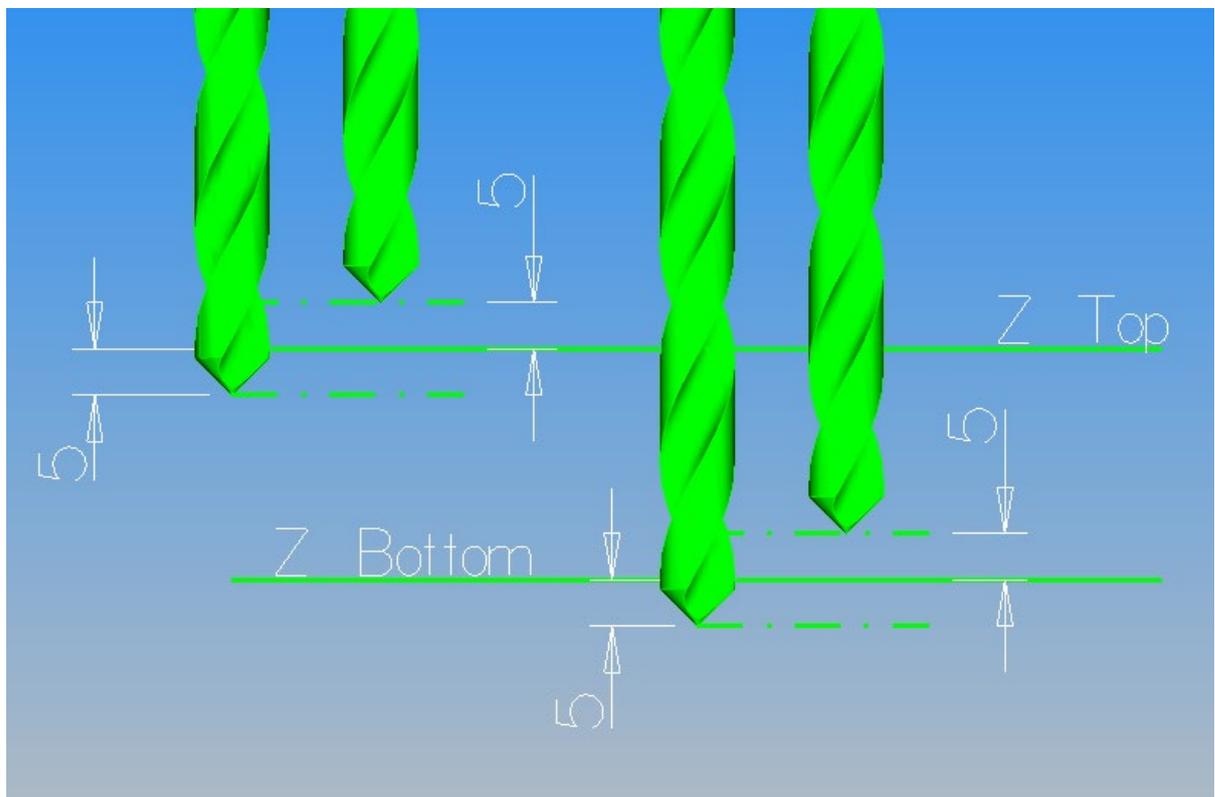


Figure 71 - Setting Stock options in Auto Z Drilling operations

This makes it easier to apply Auto Z hole spot drilling and hole chamfers.

Auto Z Hole Depth Selection

Hole Depths - Auto Z <input checked="" type="radio"/> Drill Tip <input type="radio"/> Shoulder
Hole Depths - Feature Blind Holes <input checked="" type="radio"/> Auto <input type="radio"/> Shoulder <input type="radio"/> Drill Tip
Hole Depths - Feature Through Holes <input checked="" type="radio"/> Shoulder <input type="radio"/> Drill Tip

Figure 72 - Auto Z Hole Depth Options

On **Feature Blind Holes** (Feature Extracted Holes that do not pass through a part) the default option is set to Auto. ALPHACAM will check the described drill angle from the Tool Definition dialogue against the Feature Extracted geometries and stop the drill when the first point of contact is made. You can force Shoulder or Drill point should you wish.

On **Feature Through Holes**, the default option is set to the shoulder.

Process Plan

The process plan for machining this component will vary depending on the machine tool configuration. Produce a process plan with the tutor to machine the component using Auto Z machining processes. In some circumstances, it may be a requirement to create additional geometries that cannot be extracted because the edges do not exist i.e. Pilot holes for the pocketing.

Any geometry created manually may be manipulated on to **User Layers** for ease of selection/machining. Save the Job in the “... LICOMDIR\Alphacam Trainee Files!” with a suitable name.

Once you have created the machining you can consider if the machining strategies would be useful for other jobs.

If the same strategy can be used on other parts then you can save the machining operation as a **Machining Style**. These machining styles can be applied to geometries in other jobs to replicate the machining operations. Because the machining will be Auto Z machining, the machining depths will be associative from the geometry in the new job.

To create a machining style <RClick> on the detail line of the Operation in the Operations project manager page. From the local popup menu select the option to **Save as New Style**.

#3 Dome

Hole Extraction using Any Orientation

Using **HOME > Input CAD** 

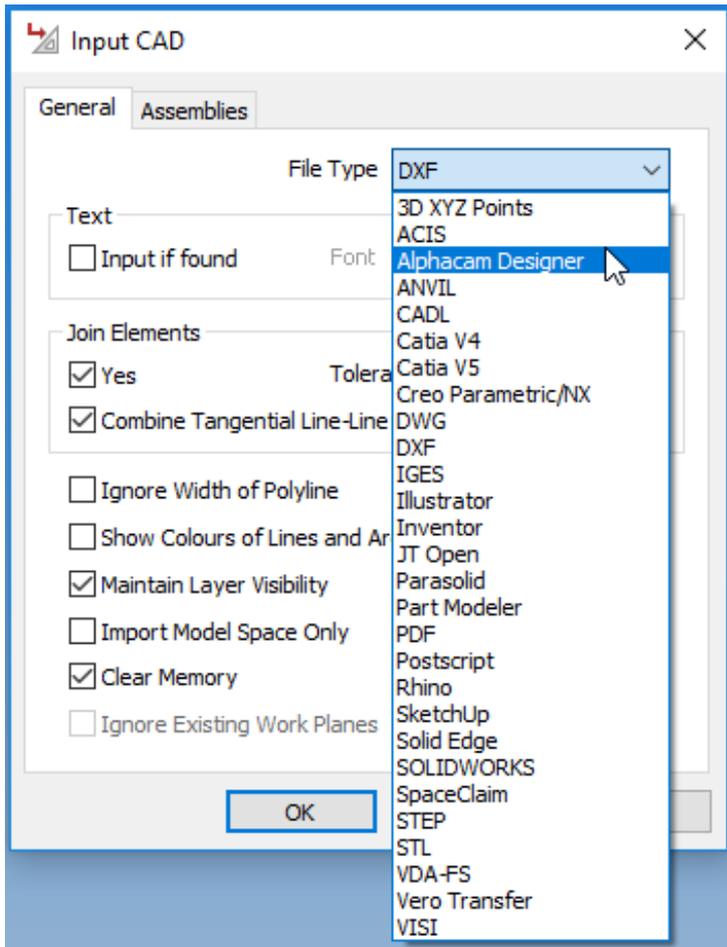


Figure 73 - input Cad dialogue set to ALPHACAM Designer

Set the file type to **Alphacam Designer**, and then select **[OK]**.

The file selection dialogue is displayed

Navigate to “**...ALP TRG 114 Feature Extraction 2020\Examples\Alphacam Designer Examples**” and input the Designer part: -

“**#3 Dome.vdf**”

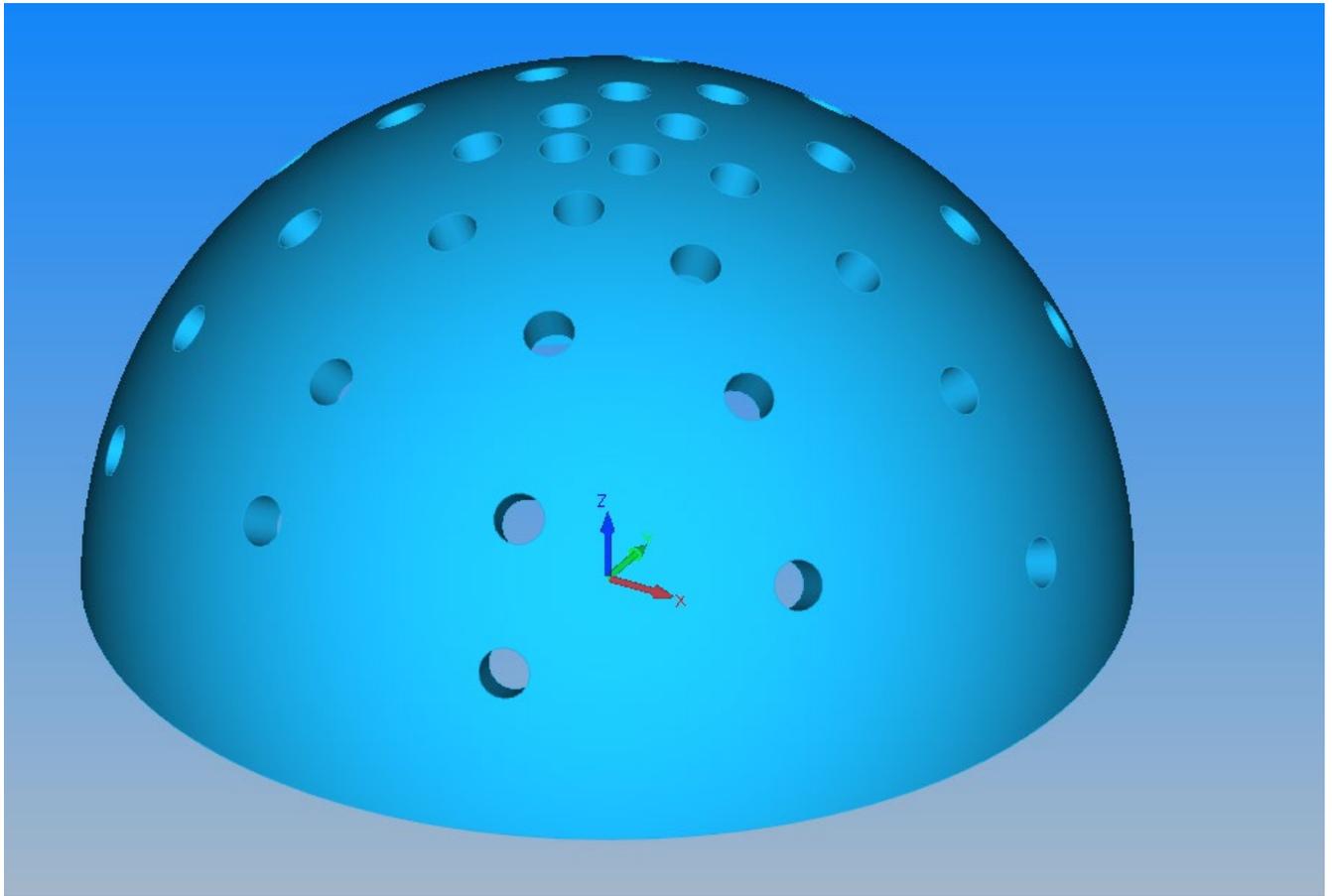


Figure 74 - Dome example file



Remember you can always Drag & Drop to make things easier.

Select **SOLID MODEL EXTRACT > Automatic Extraction** 
The options dialogues are displayed.

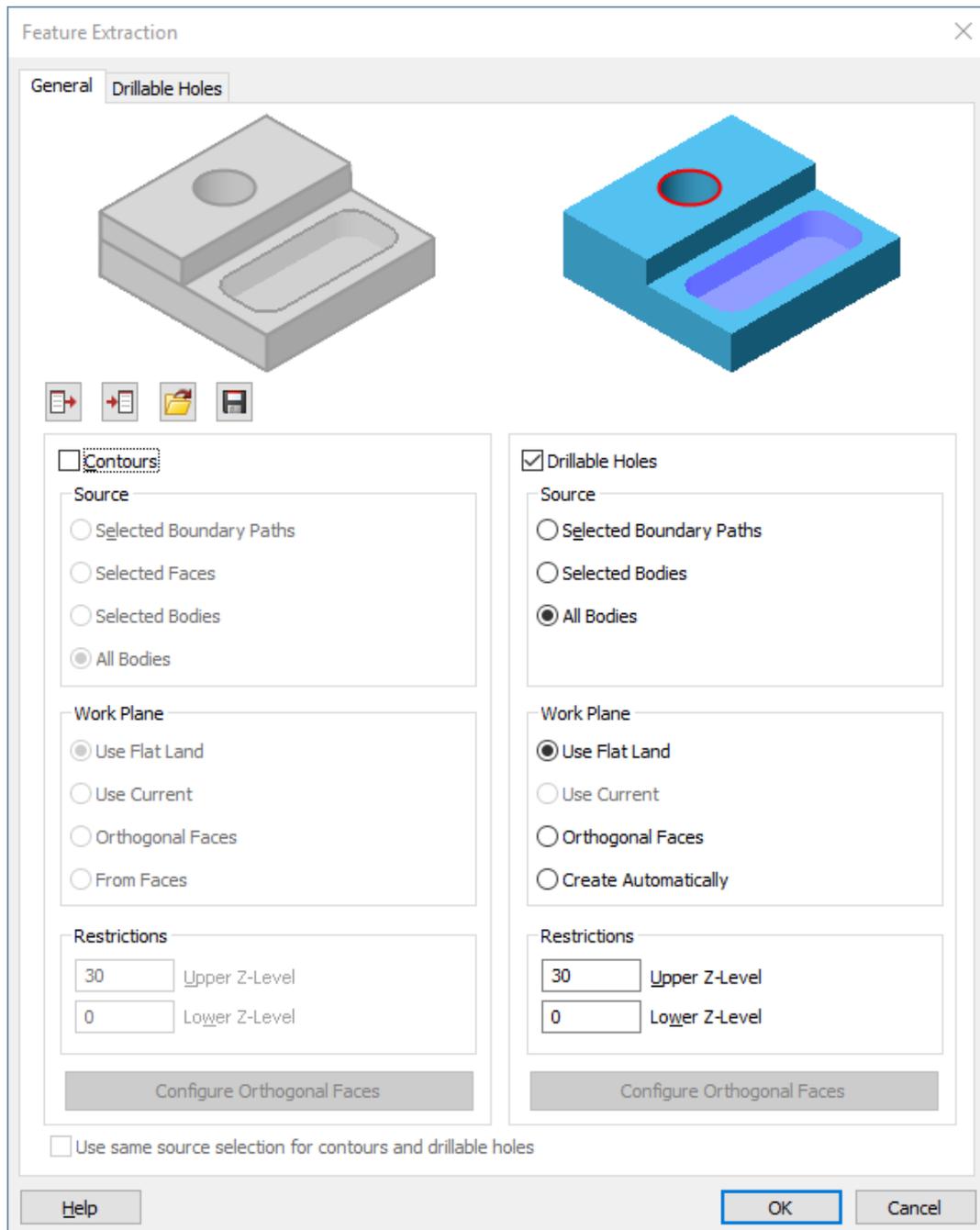


Figure 75 - Automatic Extraction options with Contours disabled

Set the options to **Drillable Holes** only, **Create Automatically** as the changes from the previous use of this dialogue option.

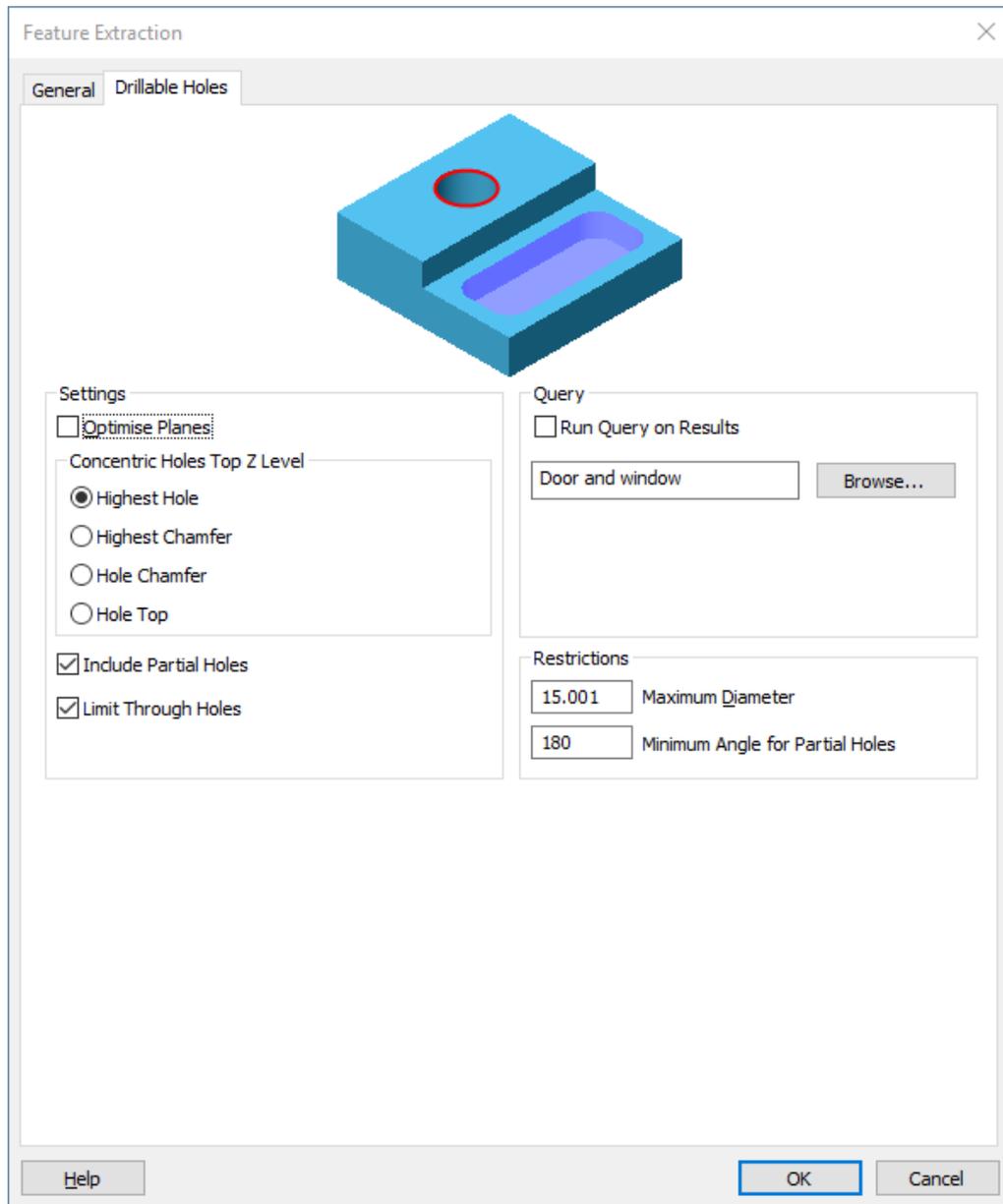


Figure 76 - Drillable Holes options to suit the model requirements

<LClick> [OK].

All the holes will be recognised, and Work Planes created to suit each hole's attitude in relation to the model.

 This short tutorial was just to demonstrate the power and ease of use in creating Work Planes for holes at various angles.

This method can also simplify the creation of compound work planes for other geometry, whilst the work plane datum will be located on the centre of an extracted hole, this is not to preclude it from being altered to suit dimensions on a drawing.

#4 Multi Plane Extraction

Extraction for Contours using From Faces

Using **HOME > Input CAD** 

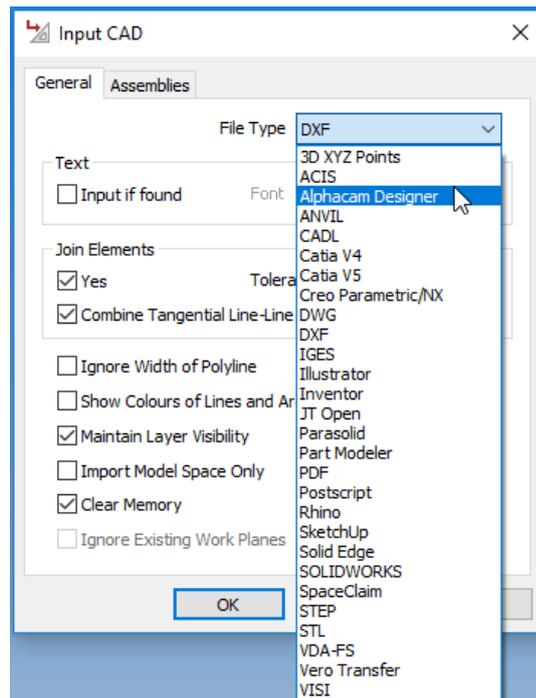


Figure 77 - input Cad dialogue set to ALPHACAM Designer

Set the file type to **Alphacam Designer**, and then select **[OK]**.
The file selection dialogue is displayed.

Navigate to “...\\ALP TRG 114 Feature Extraction 2020\\Examples\\Alphacam Designer Examples\\” and input the Designer part: -
“#4 Multi Plane Extraction.vdf”

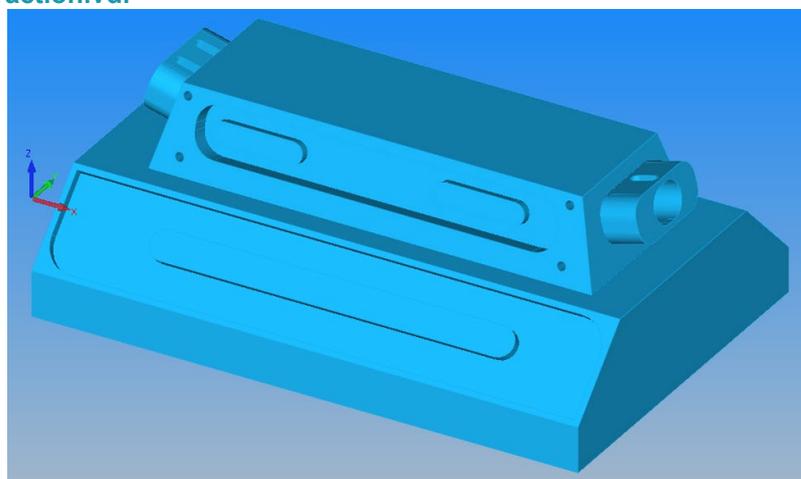


Figure 78 - Multiplane example file

Select **SOLID MODEL EXTRACT > Automatic Extraction** 
The options dialogues are displayed.

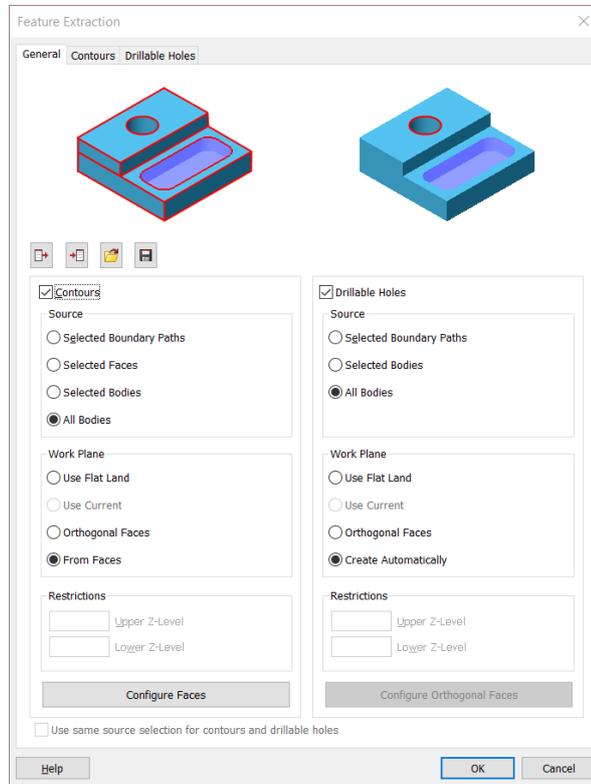


Figure 79 - Feature Extraction options set for multiplane extraction

Using the Configure Faces button will allow you to alter the face selection process and dictate how the work planes will be generated for the extracted geometry.

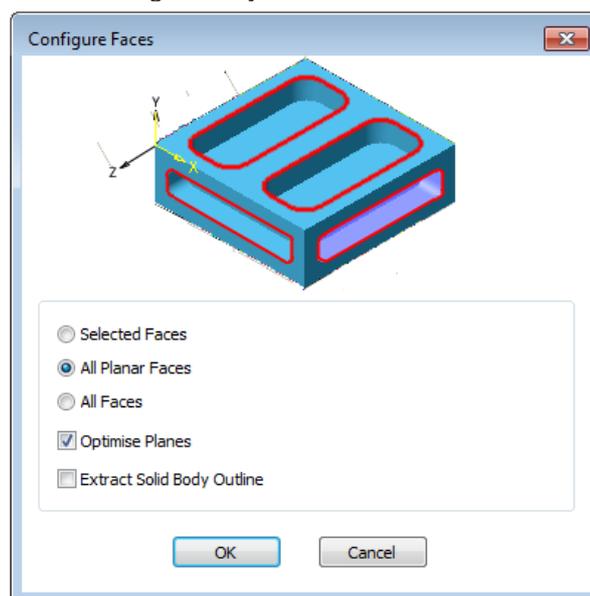


Figure 80 - Face Configuration dialogue

<LClick> [OK] to accept the face configuration options.
<LClick> [OK] to start the **Feature Extraction** process.

Below is the part immediately after clicking [OK] to apply the Feature Extraction options.

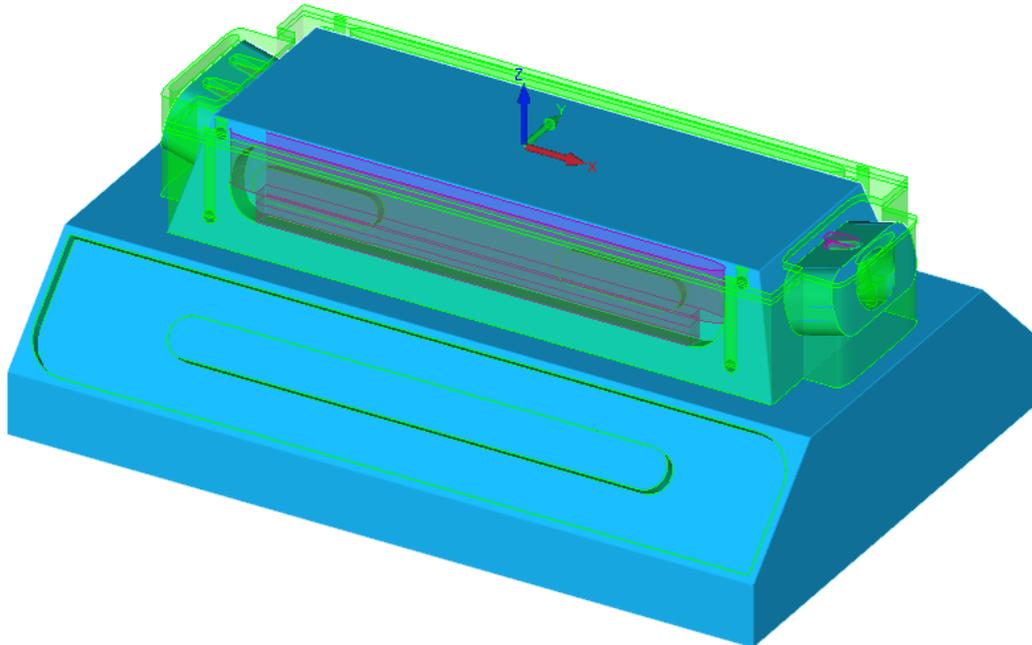


Figure 81 - Multi Plane part with all geometries shown



Remember, **Feature Extraction** looks at absolutely everything, so you may gain unwanted or strange geometries.

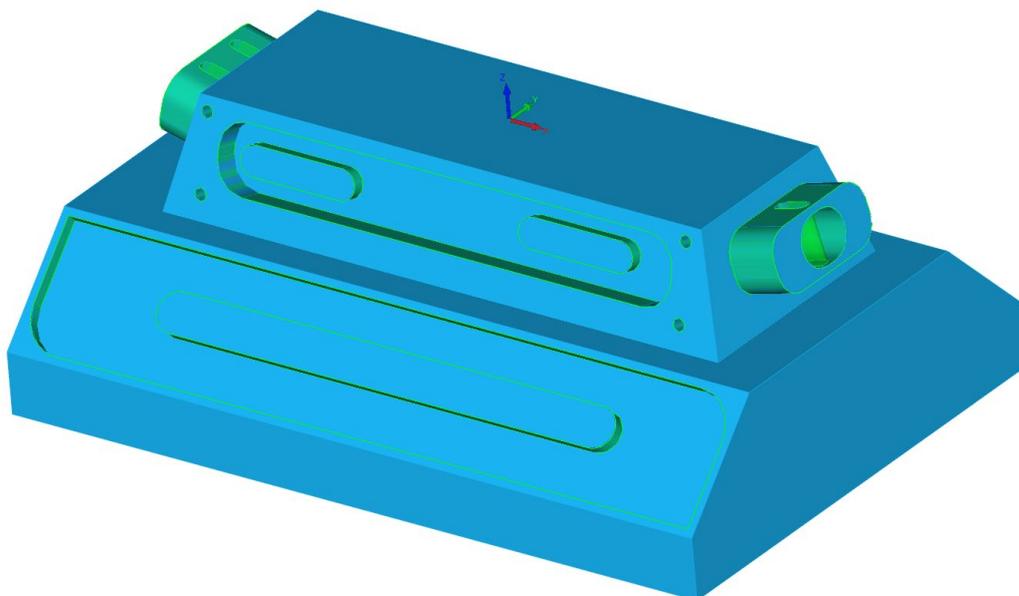


Figure 82 - Extracted face details with work planes correctly generated

The image above shows the geometries after Hiding or Deleting unwanted items for clarity.

Use of the Find option

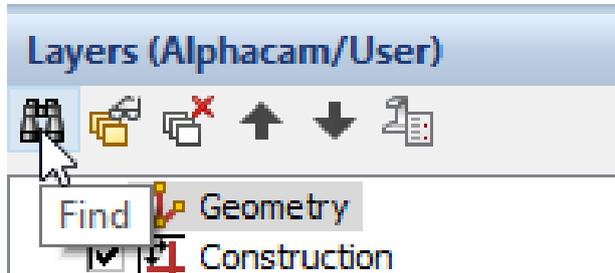


Figure 83 - Using Find on the Layers tab

Using **Find** on the Layers tab will expand the listings to show you which Layer the chosen geometry is located on.

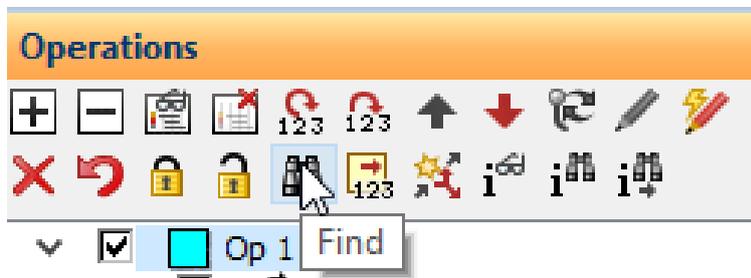


Figure 84 - Using Find on the Operations tab

Using **Find** on the Operations tab will show you which machining operation the geometry is associated with.

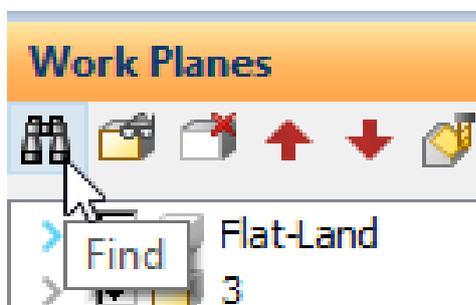


Figure 85 - Using Find on the Work Planes tab

Using **Find** on the Work Planes tab will expand the Work Plane listing on which the geometry is located.

Using the **Find**  option will allow you to easily identify where a particular geometry or tool path is located and then allow you to deal with it as you see fit.

#5 Other Extraction Techniques

Other geometry extraction and model manipulation techniques

Using **HOME > Input CAD** 

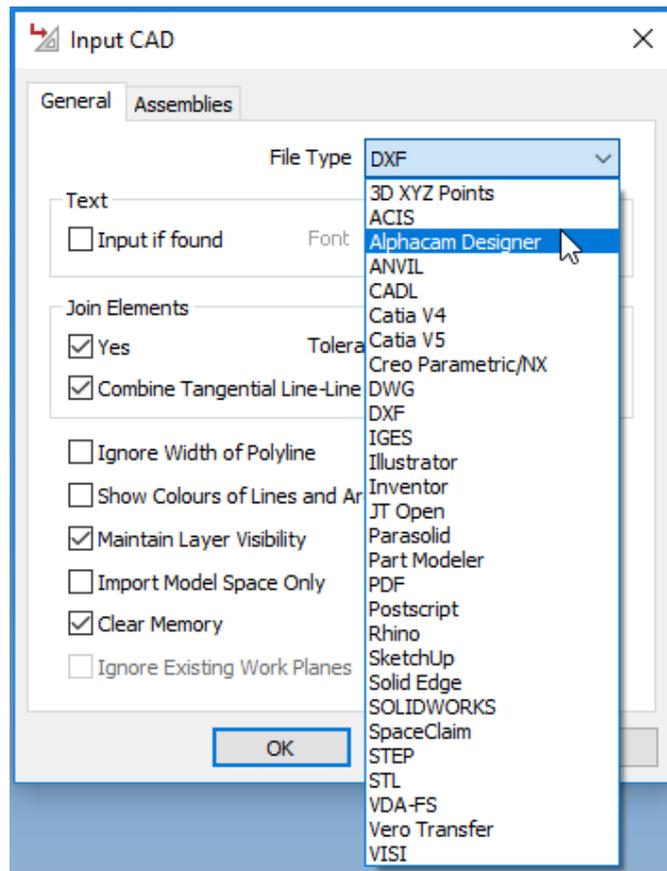


Figure 86 - Input Cad dialogue set to ALPHACAM Designer Files

Set the file type to ALPHACAM Designer, and then select **[OK]**.
The file selection dialogue is displayed.

Navigate to “...ALP TRG 114 Feature Extraction 2020\Examples\Alphacam Alphacam Designer Examples\” and input the part:

“#5 Other extraction techniques.vdf”

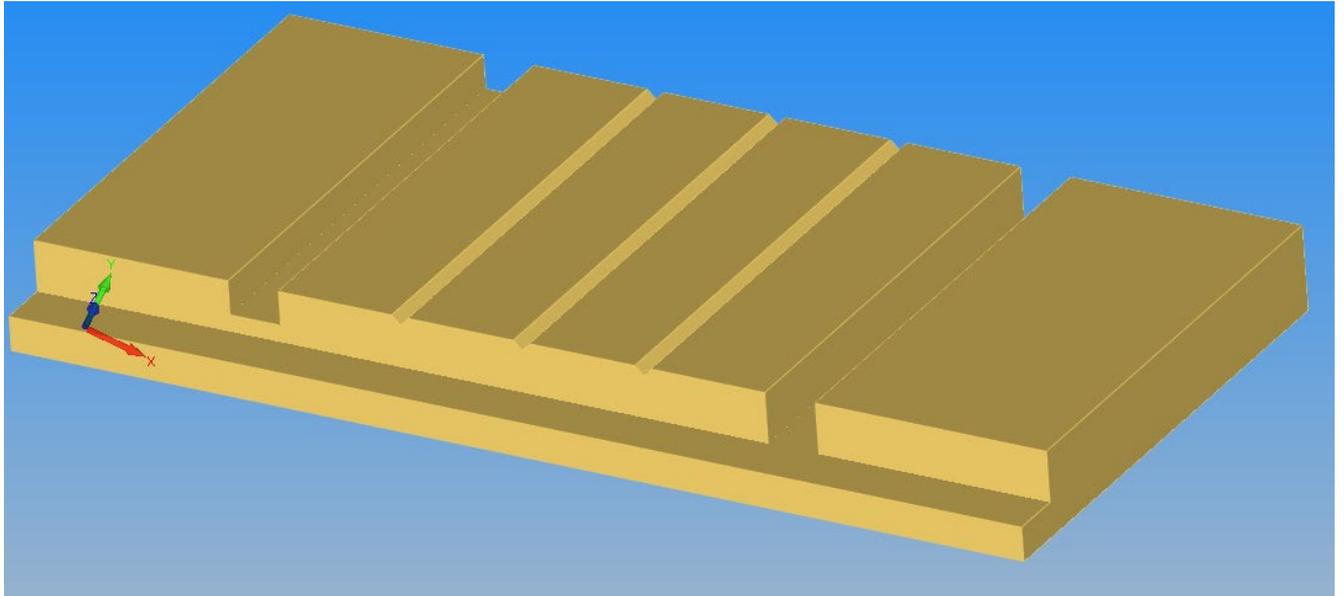


Figure 87 - Other Extraction Techniques 2019 example file

Alignment methods

There are a number of techniques in order to re align the part with the rebate at the front.

Using Auto Align

Firstly, use the **SOLID MODEL UTILITIES > Auto Align part**  to set the datum and alignment.

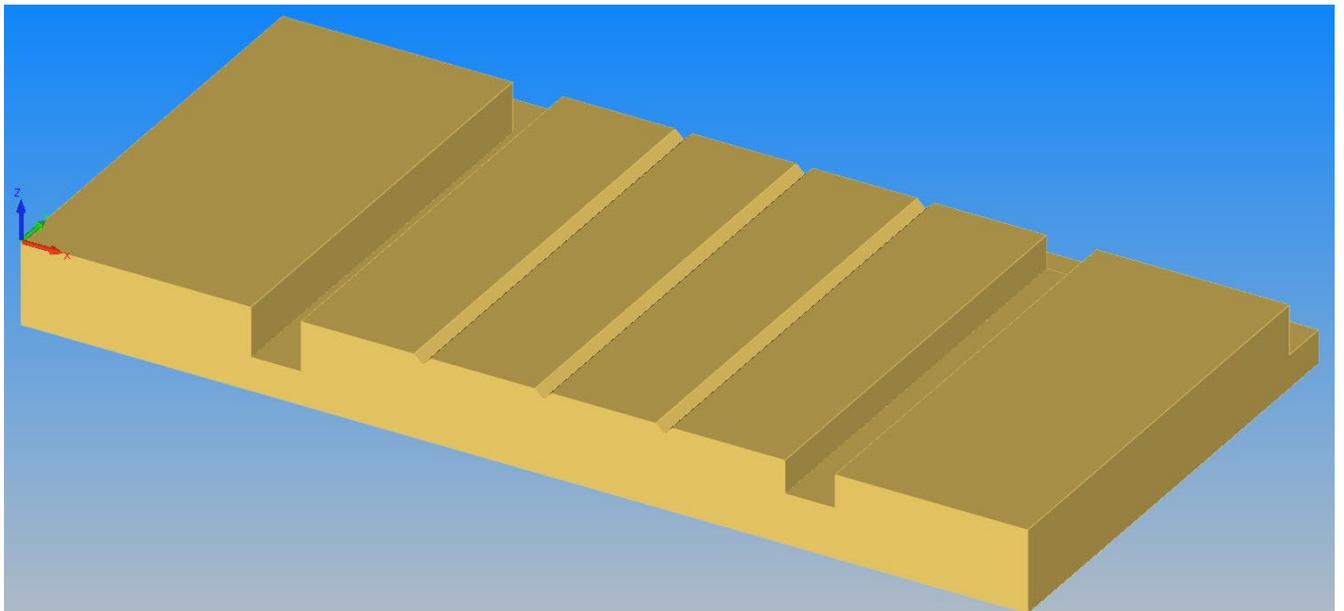


Figure 88 - Automatic Alignment correct face up

In this current view, the part is aligned with the correct face to the top but with the rebate at the back. In this example, we would like the rebate detail to be at the front.

Using Align Part

The technique when using **Align Part** is not to have the part in a similar orientation to which you require the part to be in.

As it has already been aligned using **Auto-Align**, we need to align it in another orientation. This can be done by using **Align Part**.

Select **SOLID MODEL UTILITIES > Align Part** 

For the face to point down **<LClick>** on the right end face then **<RClick>**.

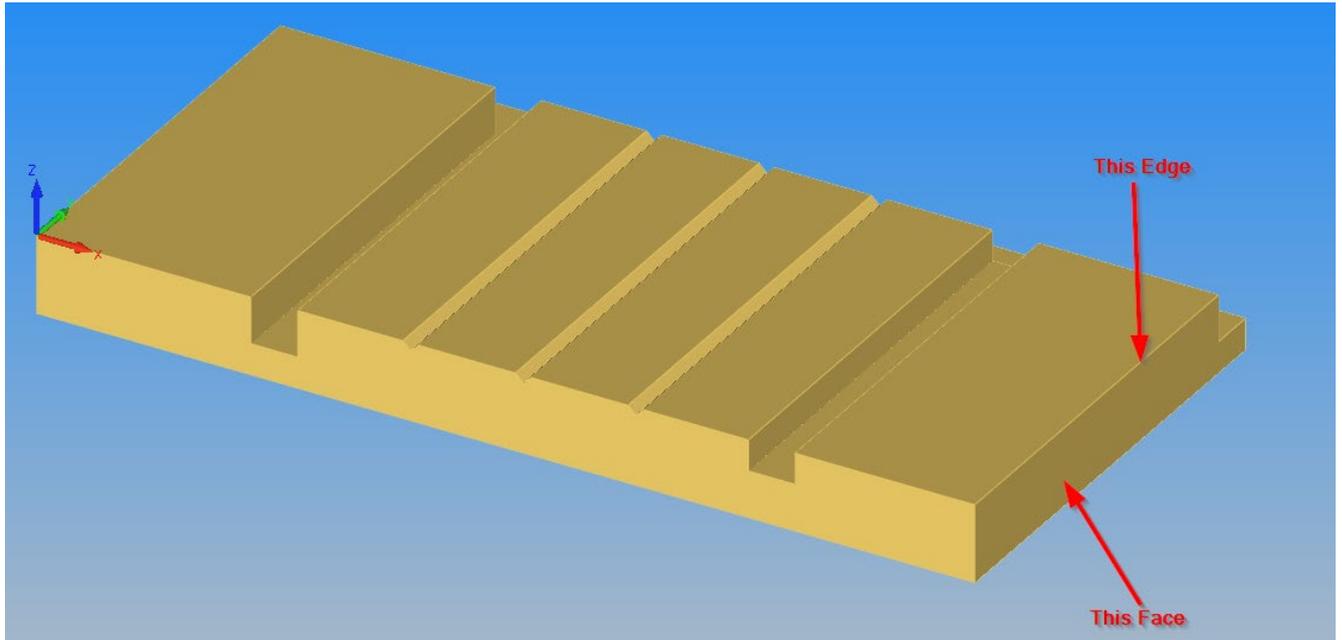


Figure 89 - First phase of manual alignment to set the part incorrectly

For the edge to align **<LClick>** on the right upper edge then **<RClick>**.



The positioning of the mouse pointer needs to be accurate as the edges are not highlighted to indicate you are considering them.

The part will be aligned on its end.
Rotate the part visually on screen and use **Align Part** again.

Select **SOLID MODEL UTILITIES > Align Part** 

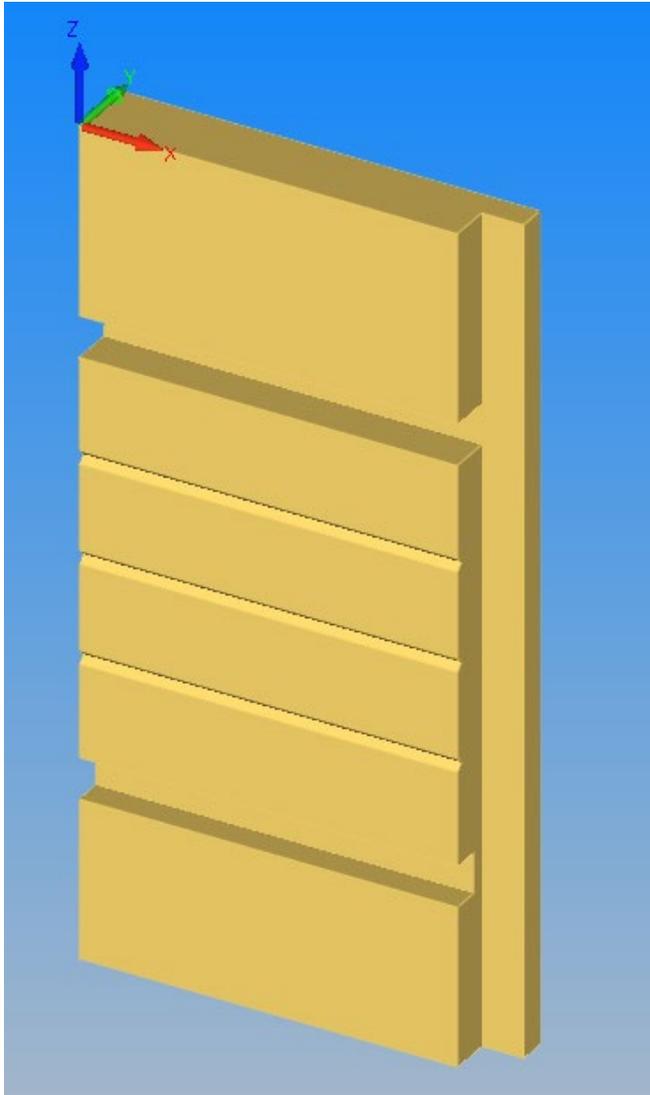


Figure 90 - Part aligned incorrectly

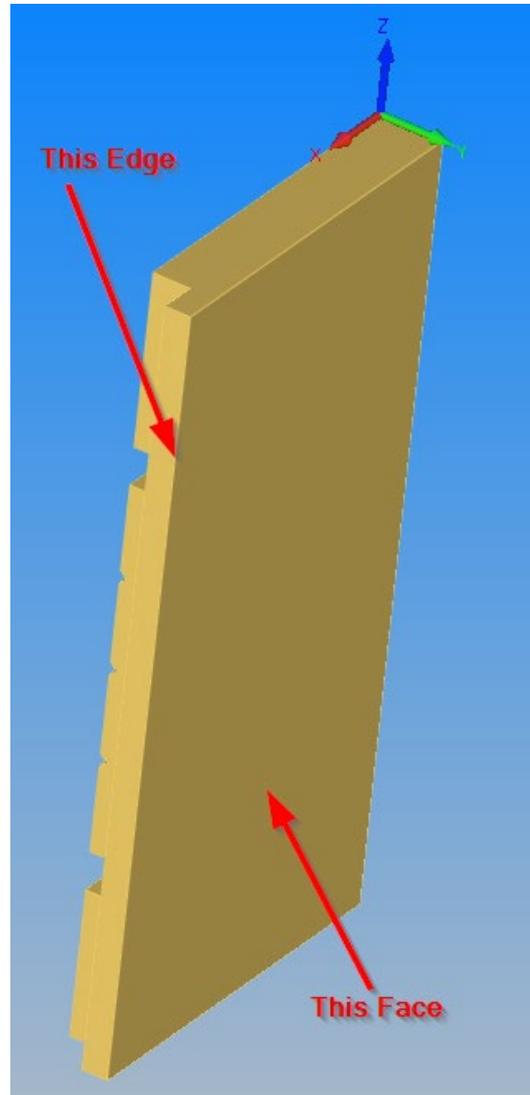


Figure 91 - Selecting the options for the correct alignment

For the face to point down **<LClick>** on the underside face then **<RClick>**.
For the edge to align **<LClick>** on the right upper edge then **<RClick>**.

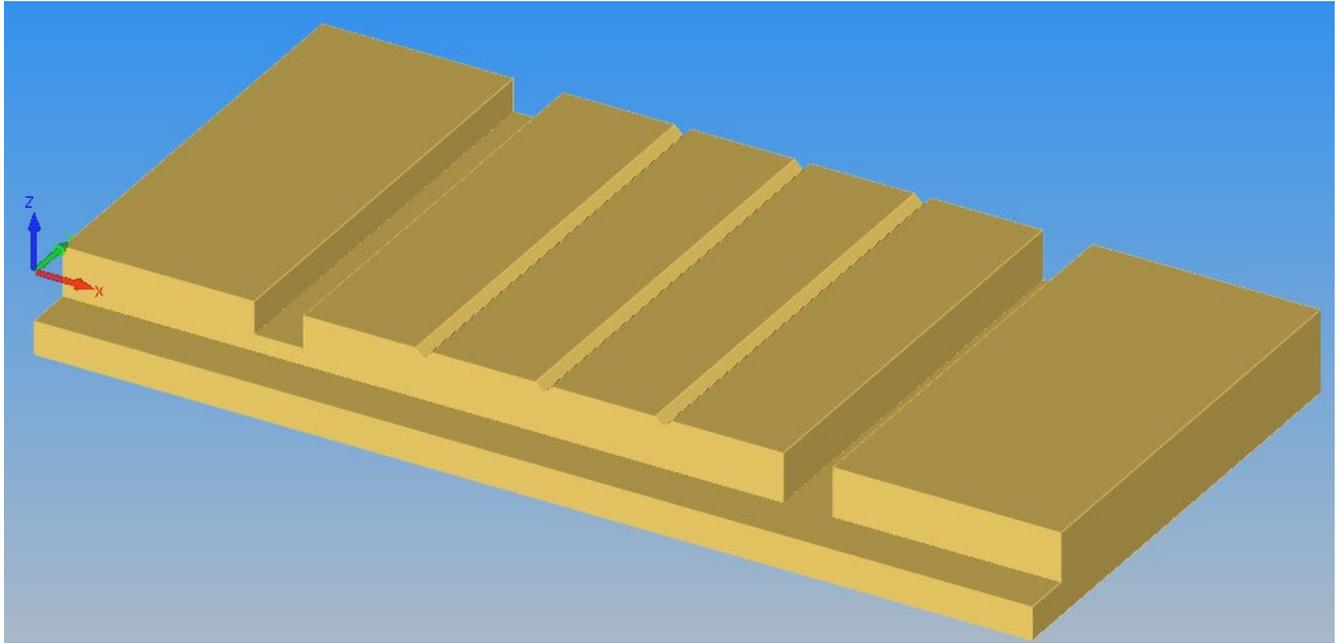


Figure 92 - Part Aligned correctly using multiple Align Part options

Using Manual Manipulation

Select **SOLID MODEL UTILITIES > Auto Align Part**  to place the part flat with the correct face uppermost.

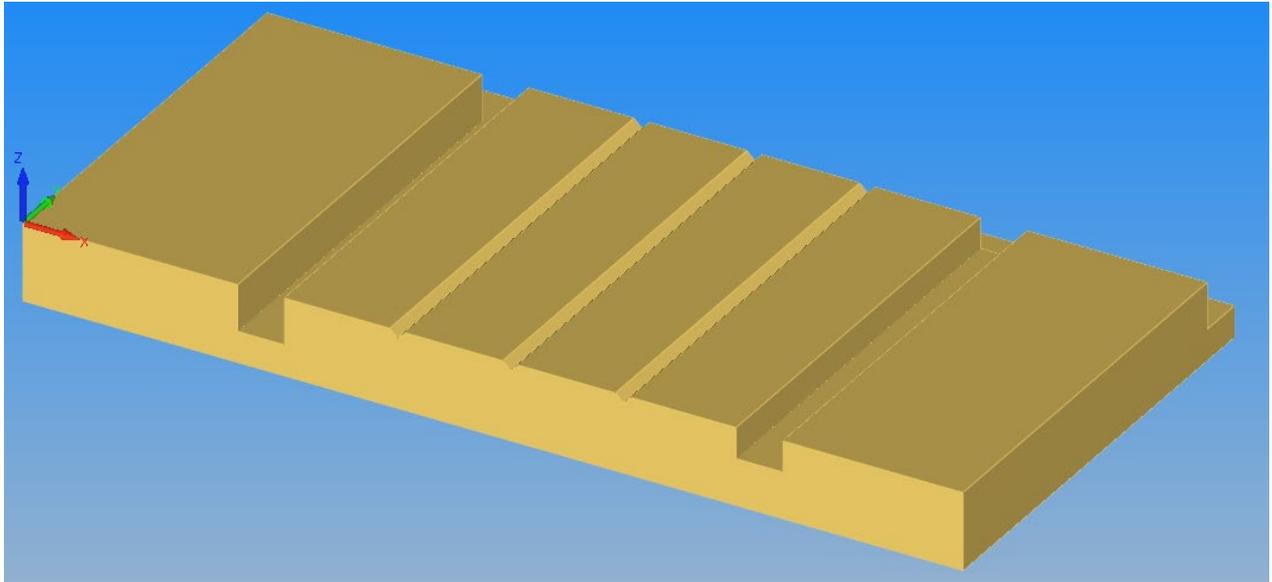


Figure 93 - Part set using Auto Align Part

Select **SOLID MODEL UTILITIES > Create Bounding Work volume**  .
The bounding work volume gives you selectable points at the models' extremities.



The theoretical intersections of the models' edges cannot be picked using any **Snap** options if the physical point does not exist.

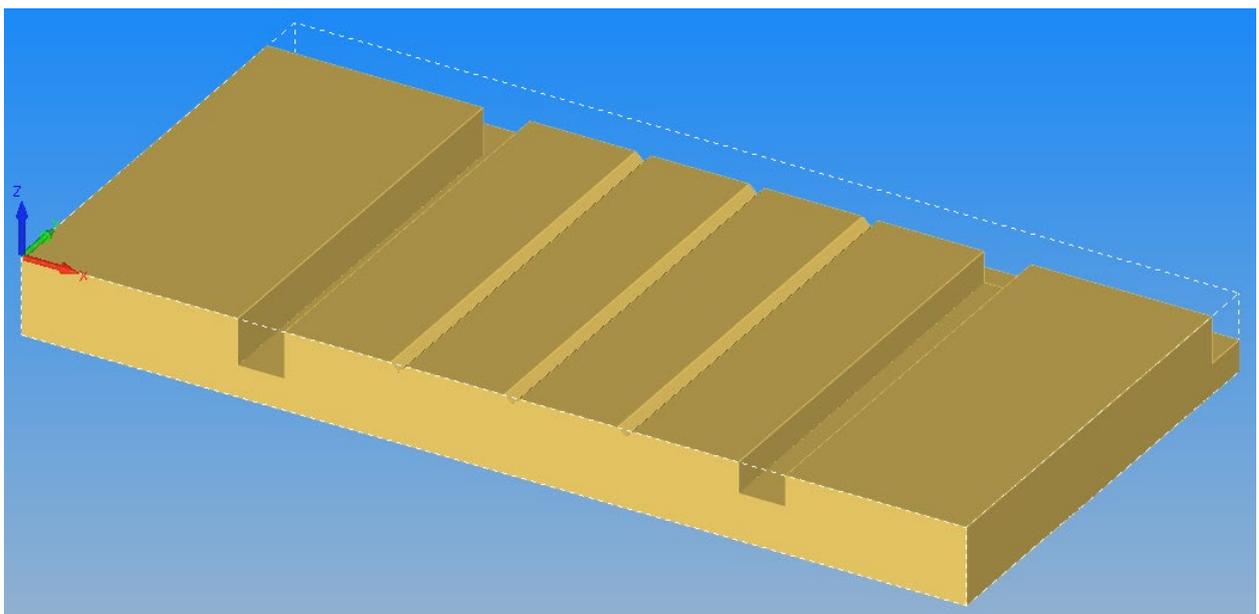


Figure 94 - Solid model with a created Bounding Work Volume

Select **EDIT > Move Copy Etc > Move** 
Select **[All]** and **[Finish (ESC)]** at the bottom of the ALPHACAM drawing window.

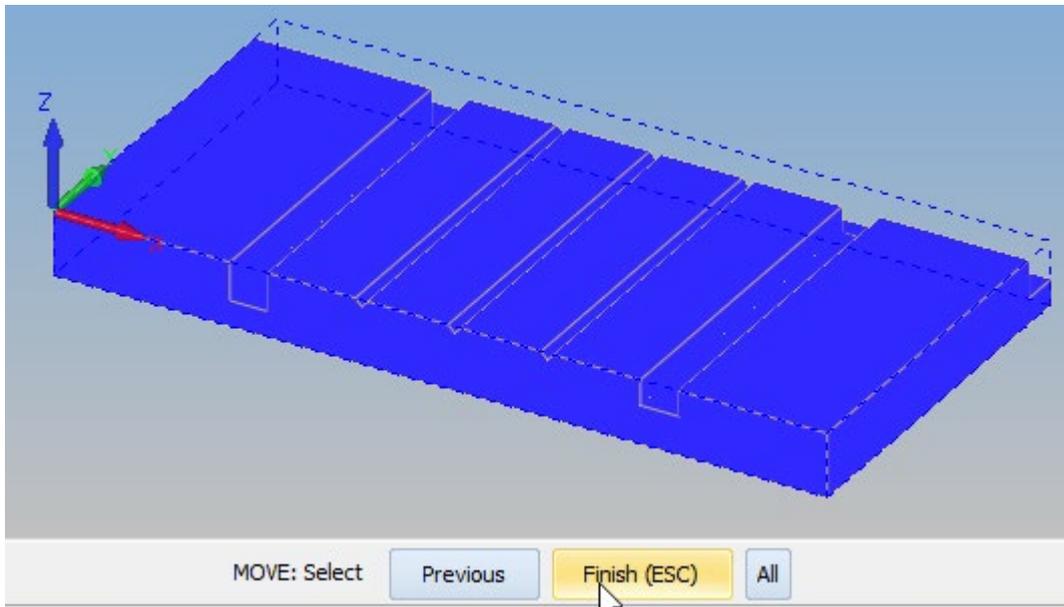


Figure 95 - Selecting the items to Move

For the base point select the top right corner of the work volume diagonally opposite the global origin.

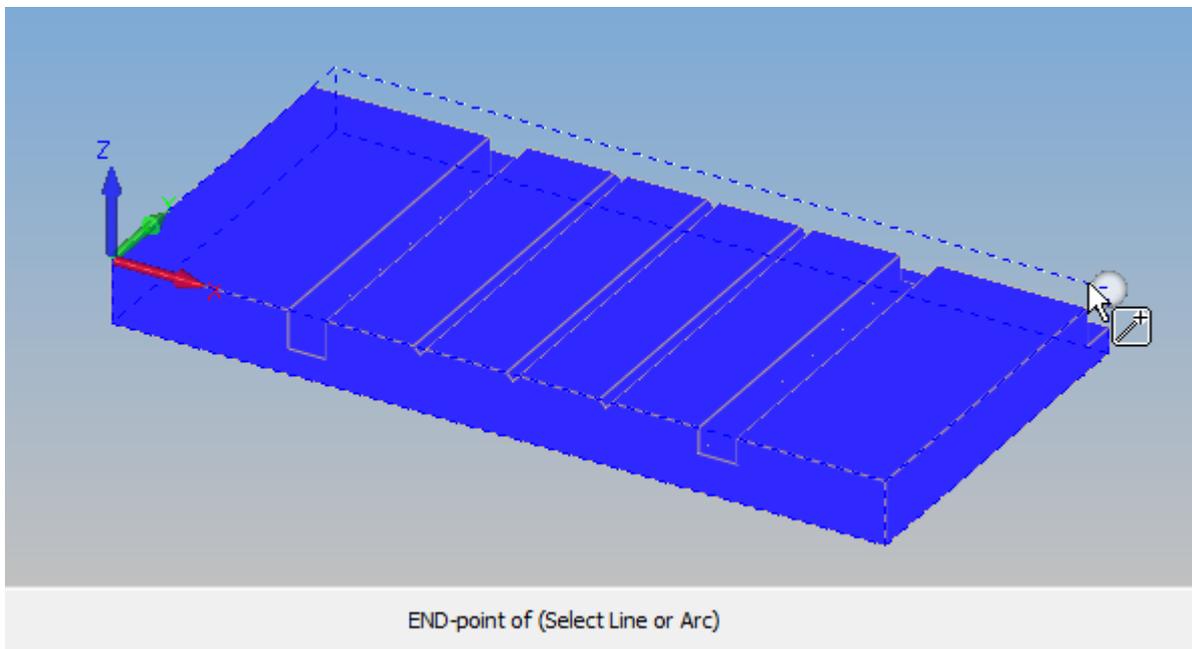


Figure 96 - Selecting the base point

For the new position enter **X0, Y0, Z0** ←
<RClick> to finish.

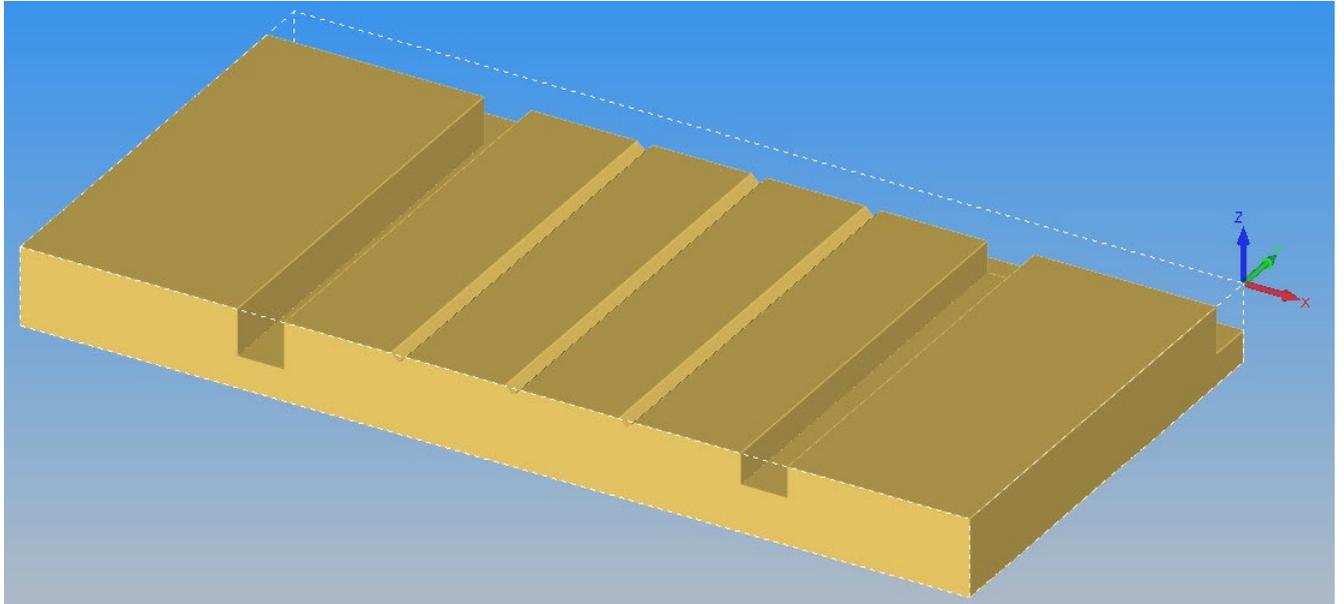


Figure 97 - Part relocated to new datum position on the theoretical corner

3D Part Rotation

Select **3D > 3D Part rotation**  and the dialogue is displayed

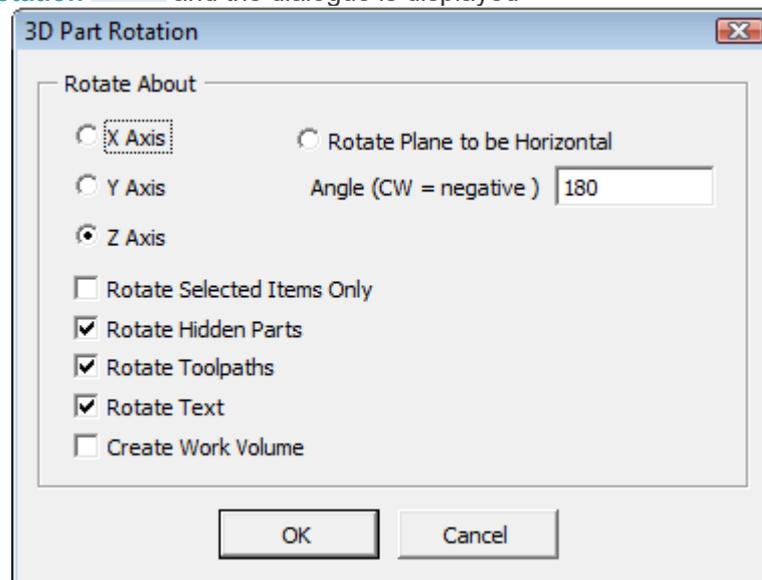


Figure 98 - 3D Part Rotation dialogue options

Select the **Z Axis** as the option to rotate about and set the angle to **180** then select **[OK]**

For the point of rotation enter **X0, Y0, Z0** ↵

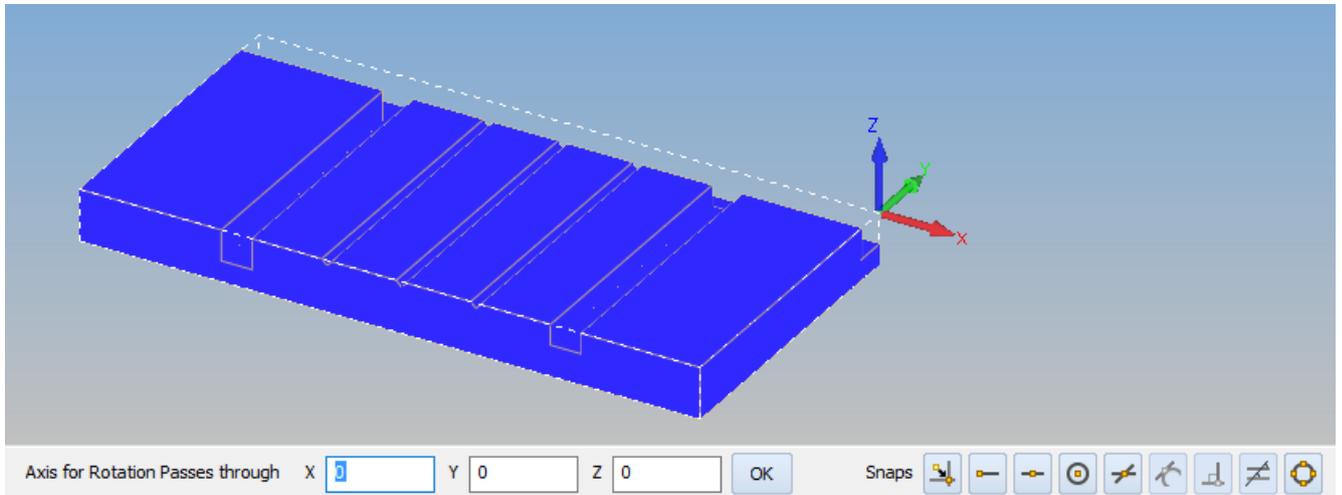


Figure 99 - Setting the axis rotation point

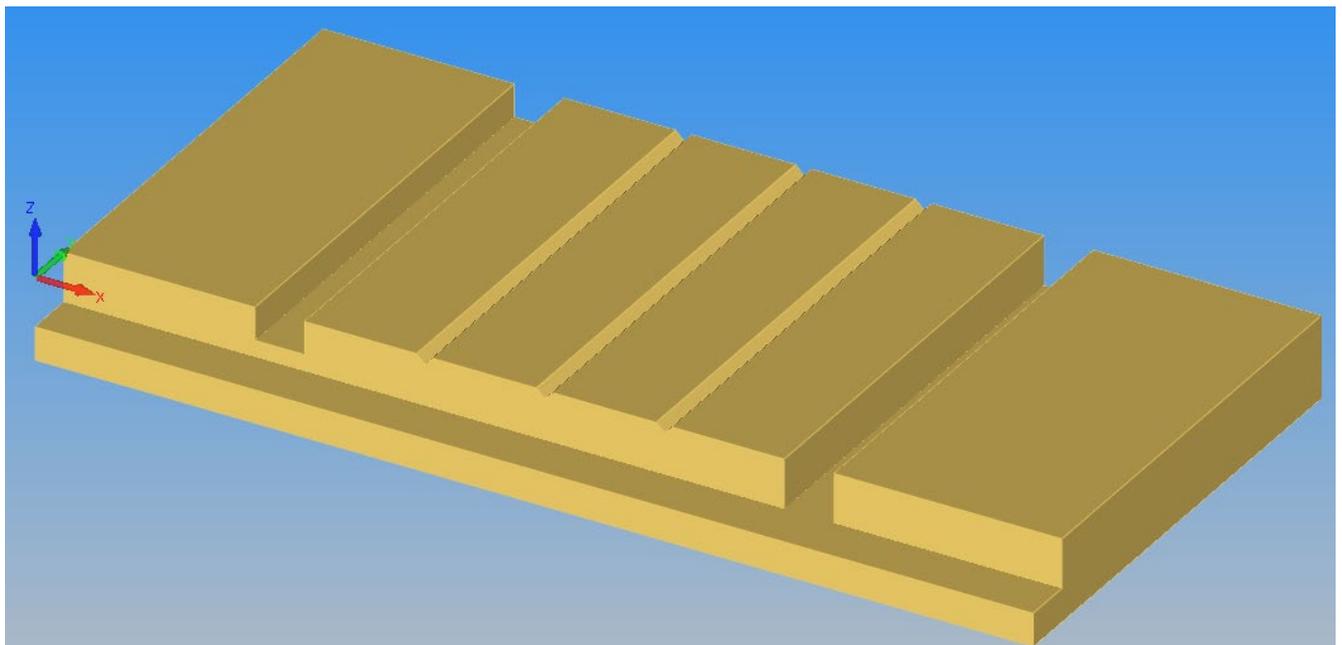


Figure 100 - Part rotated to correct attitude

The part is re aligned.



On some complex parts where there are no flat faces or straight edges, this manual method is the only means of aligning the part to suit you manufacturing techniques.

Geometry extraction not using the Open Air pocket option

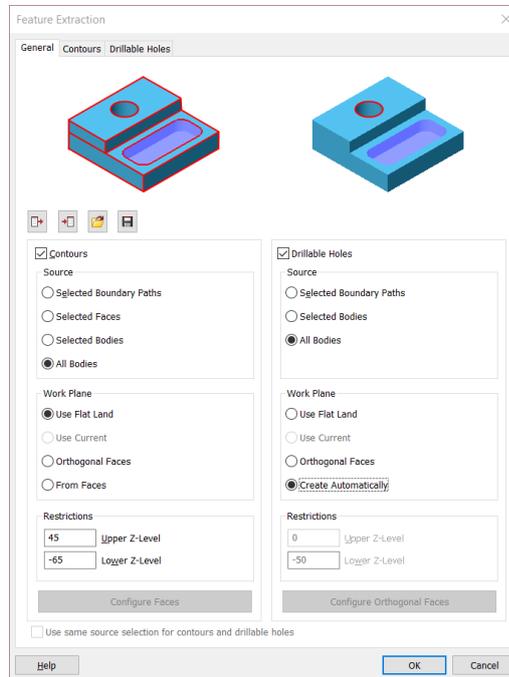


Figure 101 - Automatic Extraction options set to suit

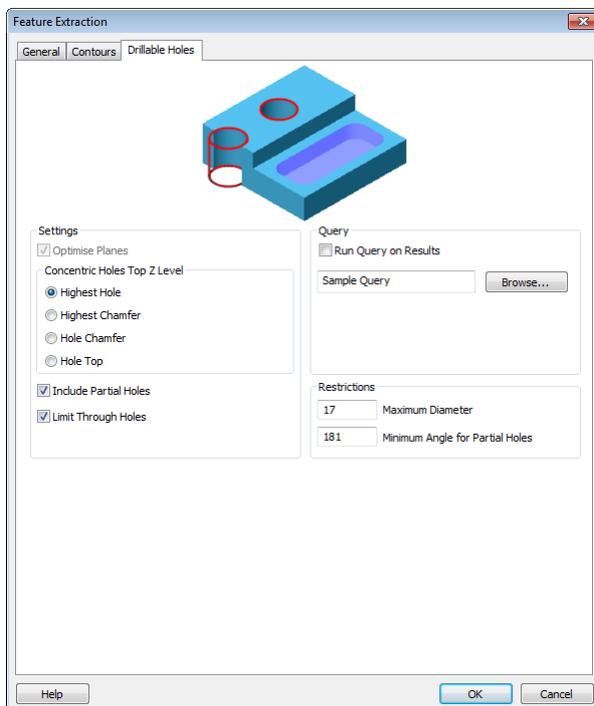


Figure 102 - Drillable Holes options

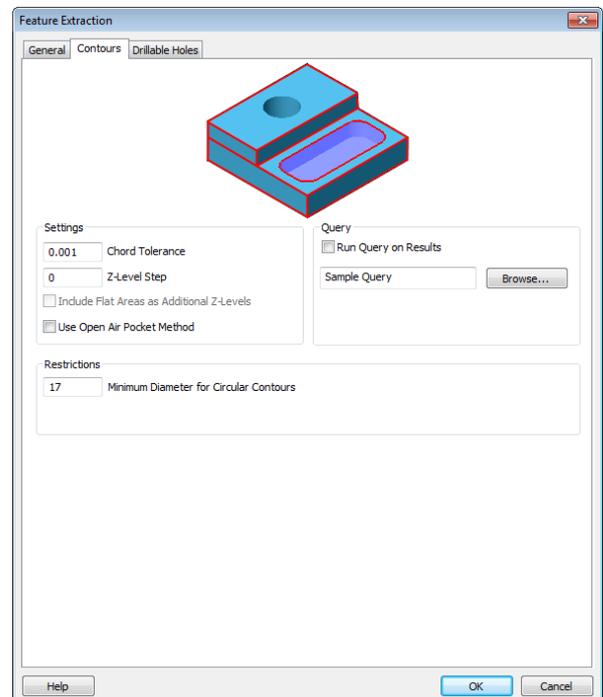


Figure 103 - Contours options

The Automatic Extraction on this part gives geometries that are not useable for machining if the **Use Open Air Pocket Method** is switched off.

The image below shows how the geometries have been extracted. The colours have been adjusted to make the image easier to understand.

All of the geometries have been placed on the **Closed Contours** layer.

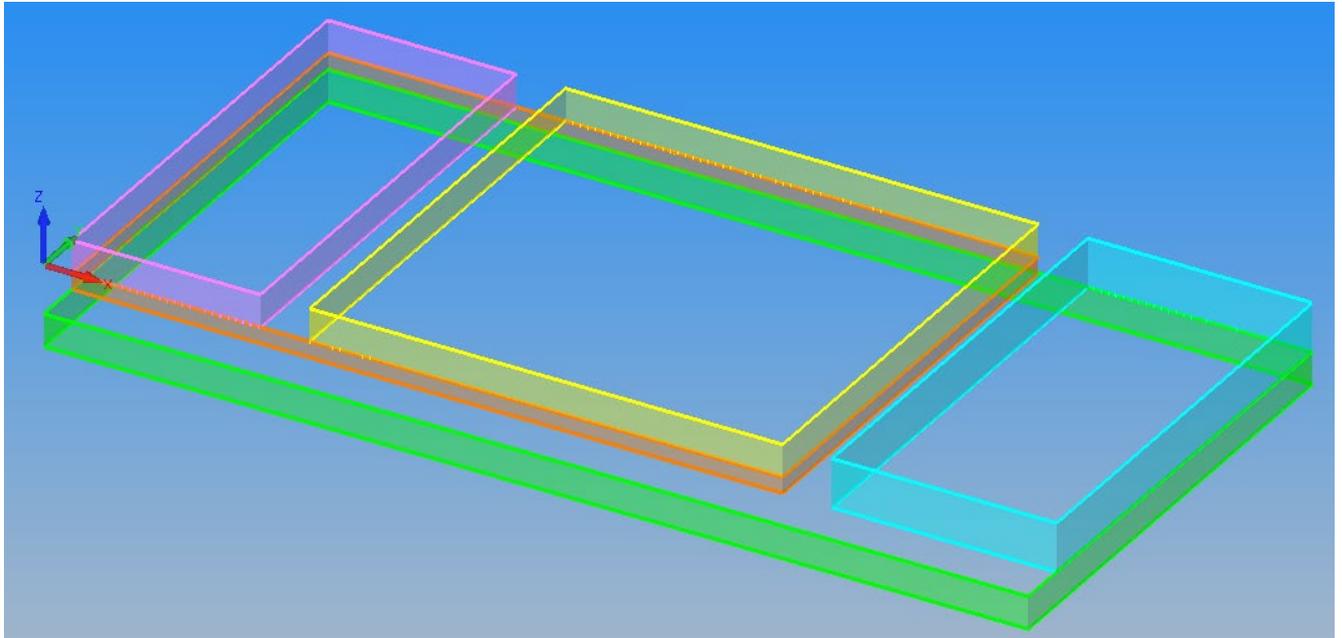


Figure 104 - Extracted Contours when not using the Open Air Pocket method



While it would be perfectly possible to machine this part, some sides would be cut more than once and this may lead to a poor quality finish or damage to the part.

Automatic Geometry extraction using the Open Air Pocket method

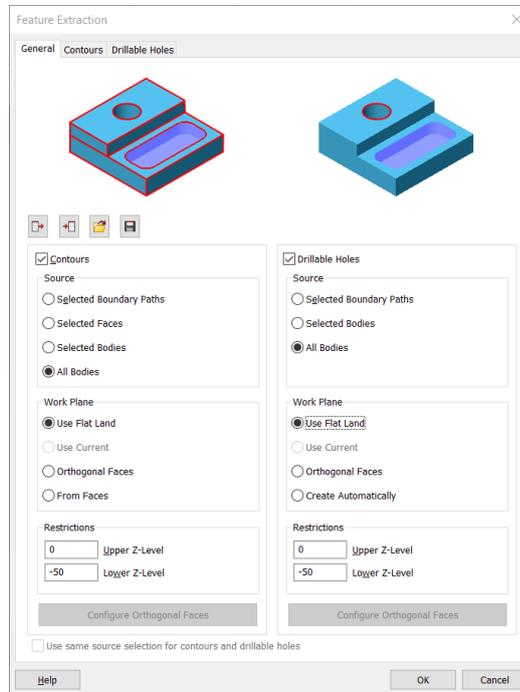


Figure 105 - Automatic Extraction General options to suit

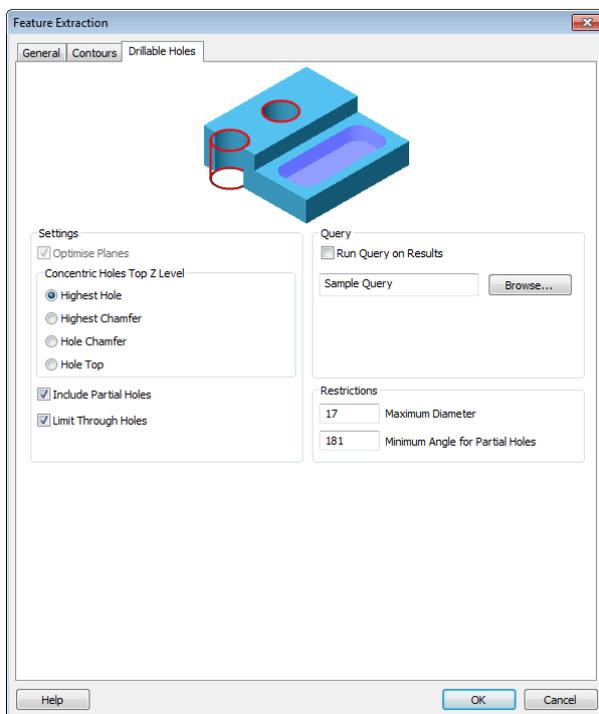


Figure 106 - Drillable Holes options to suit

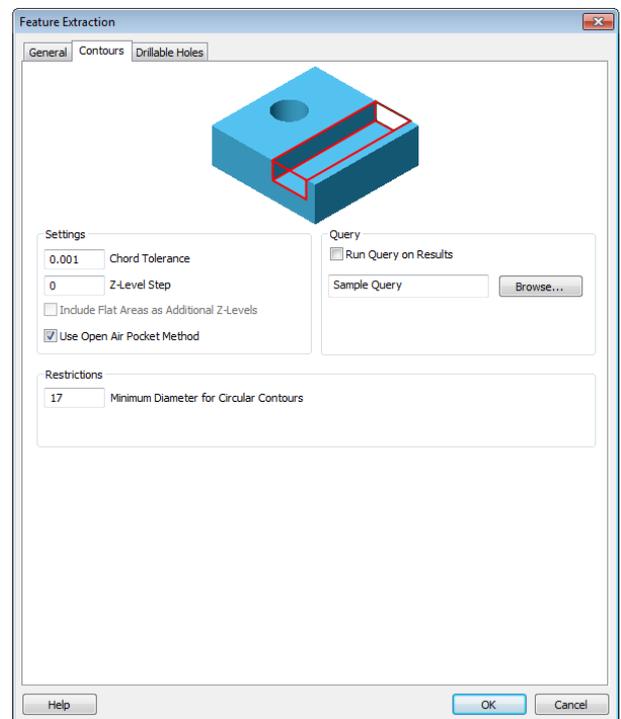


Figure 107 - Contours options with Open Air Pocket Method active

The Automatic Extraction on this part gives geometries that are useable for machining if the **Use Open Air Pocket Method** is switched ON.

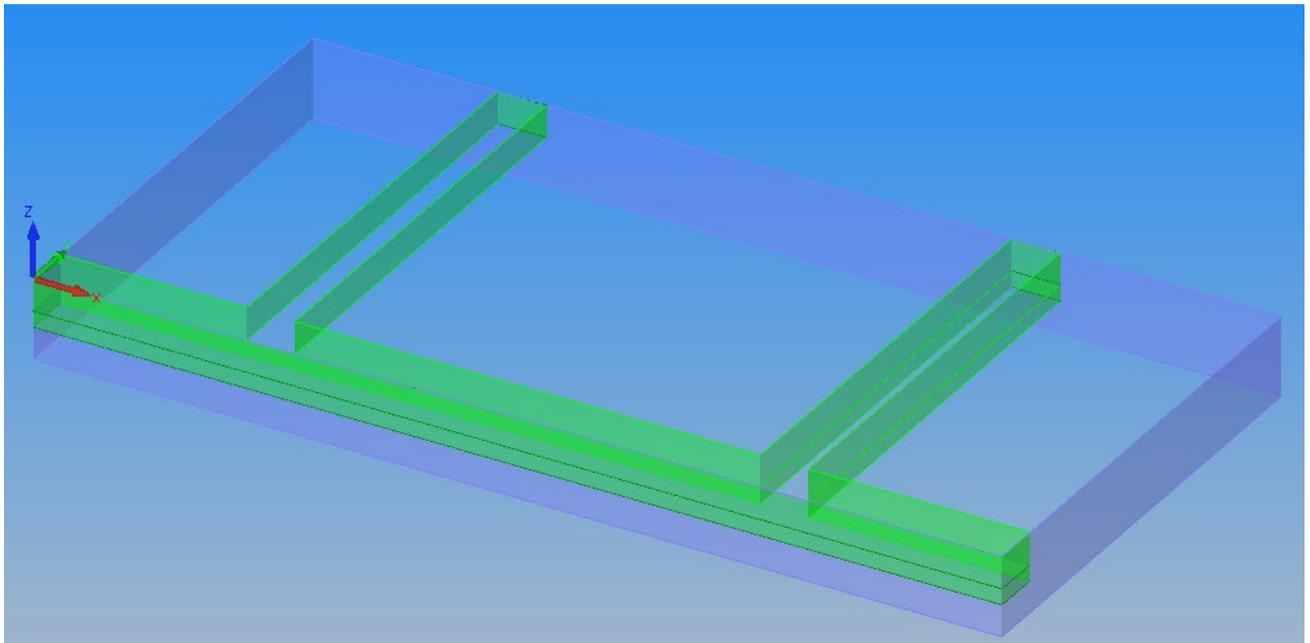


Figure 108 - Example extracted geometries when using Open Air Pocket method



The open sides of all the slot details are automatically set to **Open Elements**.

Using this method creates a type of geometry where, if the actual edges of the model are the physical outer element of the part, they will be set as an **Open Element**.

An **Open Element** is one where the tool can pass across a geometry instead of being contained by it.

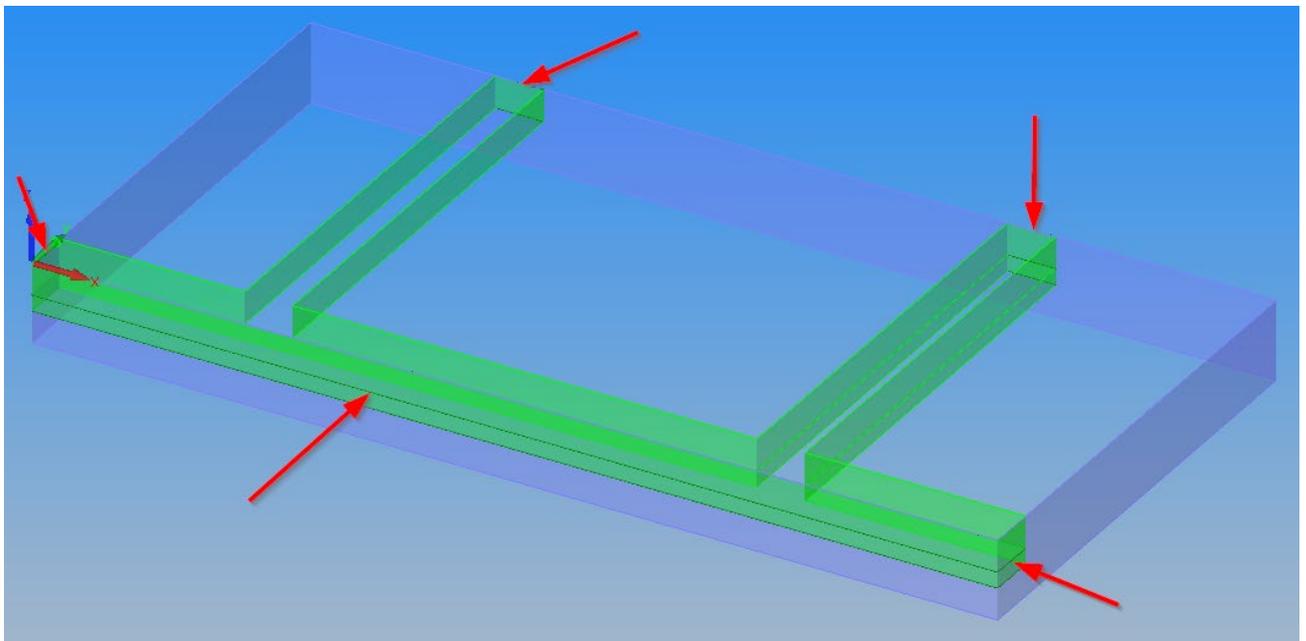


Figure 109 - Arrows indicate the Open Elements

Whilst this has given us more beneficial geometries in certain areas, there are sections which may be cut more than once, resulting in the same possible damage as the previous example shown. To overcome this, you can extract the required geometries manually.

Manual Geometry Extraction

Left Slot

To extract the left slot geometries select

SOLID MODEL EXTRACT > Contour around Picked Faces 

Select the bottom face of the left hand slot; this may be easier in the XY view.

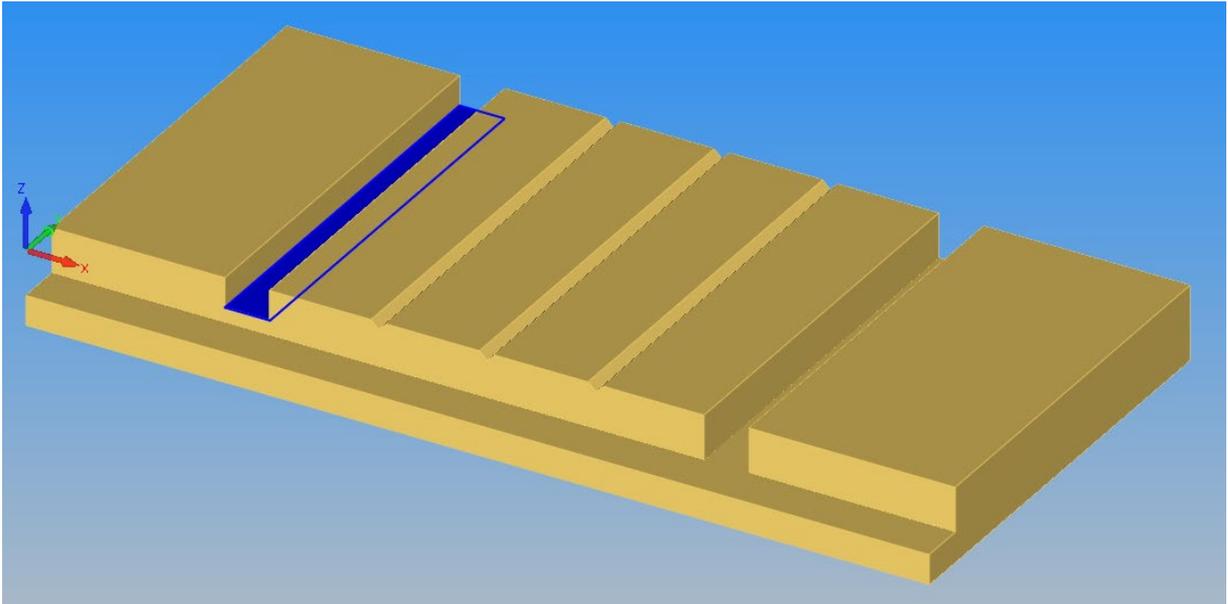


Figure 110 - Slot face selected for extraction manually

You will also be presented with the Face Selection dialogue box.

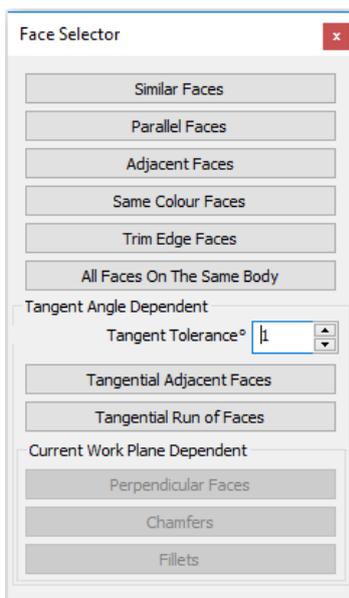


Figure 111 - Face Selector dialogue

This dialogue also allows you to select faces via various methods to make the process easier.

<RClick> to continue as this is the only face we need for this section.
The geometry will be created and the command line will prompt you to select the bottom edge.

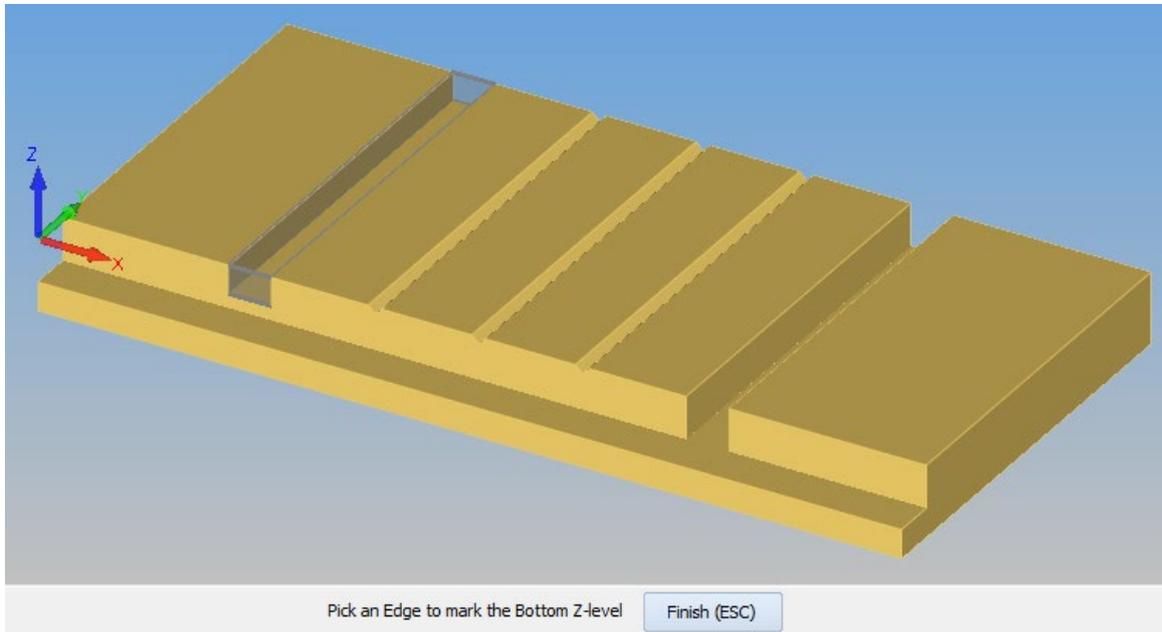


Figure 112 - Selecting the correct Bottom Edge

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

The geometry is already created at the correct Bottom Z level, if it was not at the correct Z level then select any edge that is at the required Z level.

The command line will prompt you to select the top edge.

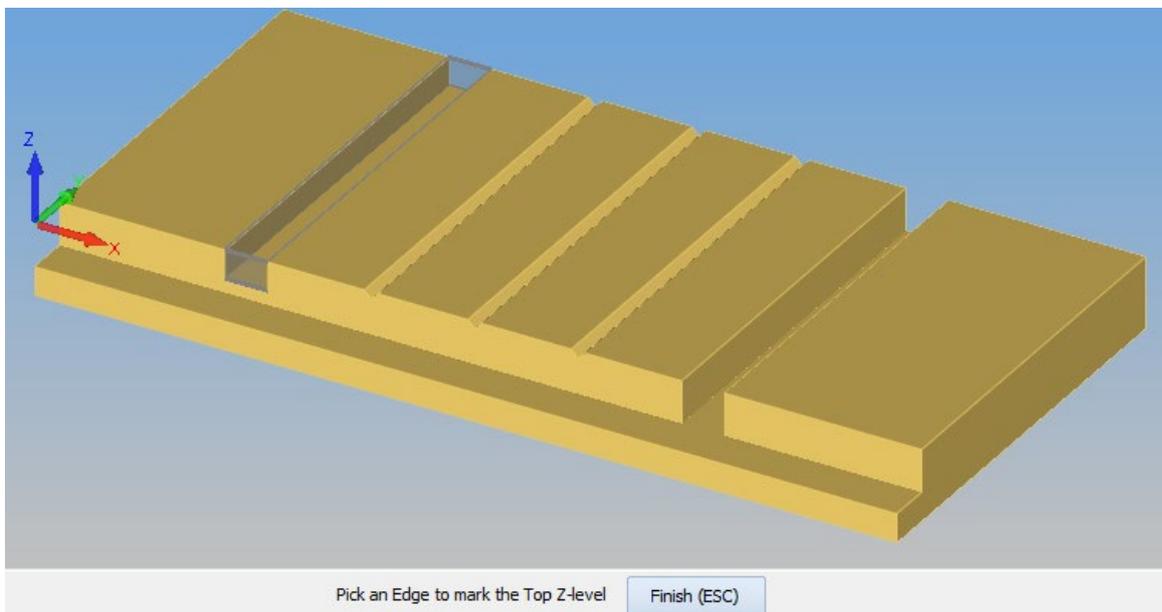


Figure 113 - Selecting the Top edge

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

The geometry is already created at the correct Top Z level, if it was not at the correct Z level then select any edge that is at the required Z level.

Altering Tool sides



Remember to check the **Tool Sides** of any geometries that you extract using this method, they may or may not be correct for machining.

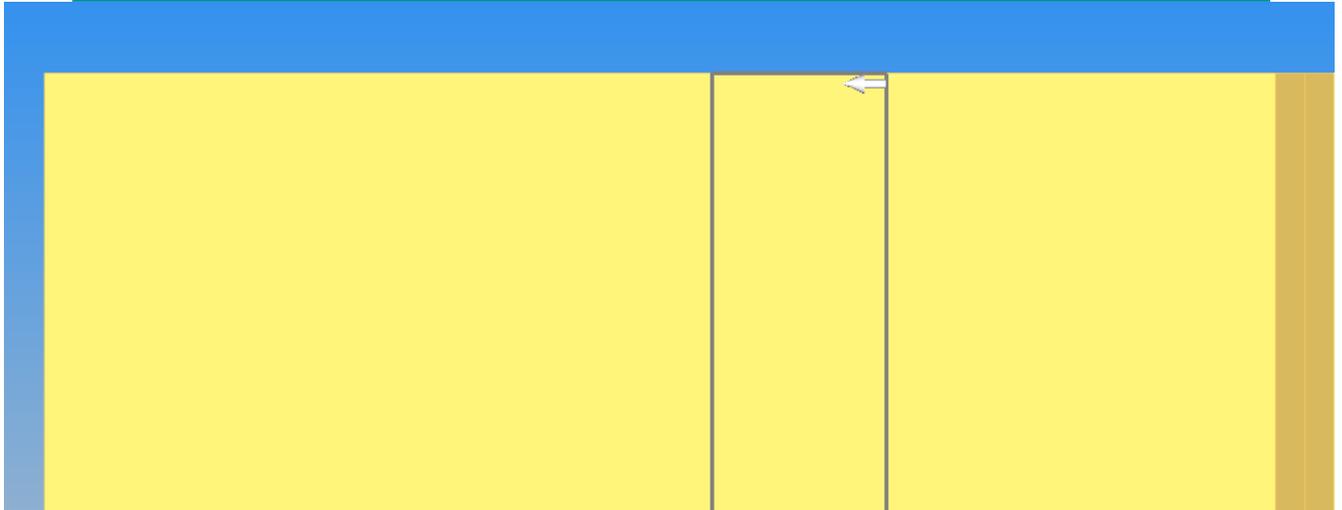


Figure 114 - Ghost Tool correctly extracted

In this case, the manual method of extraction has placed the Ghost Tool on the correct side for manufacture.

You may also wish to set the start point of the pocket to the centre of one of the open ends to remove any possibility of dwell or burn marks on the finished part.

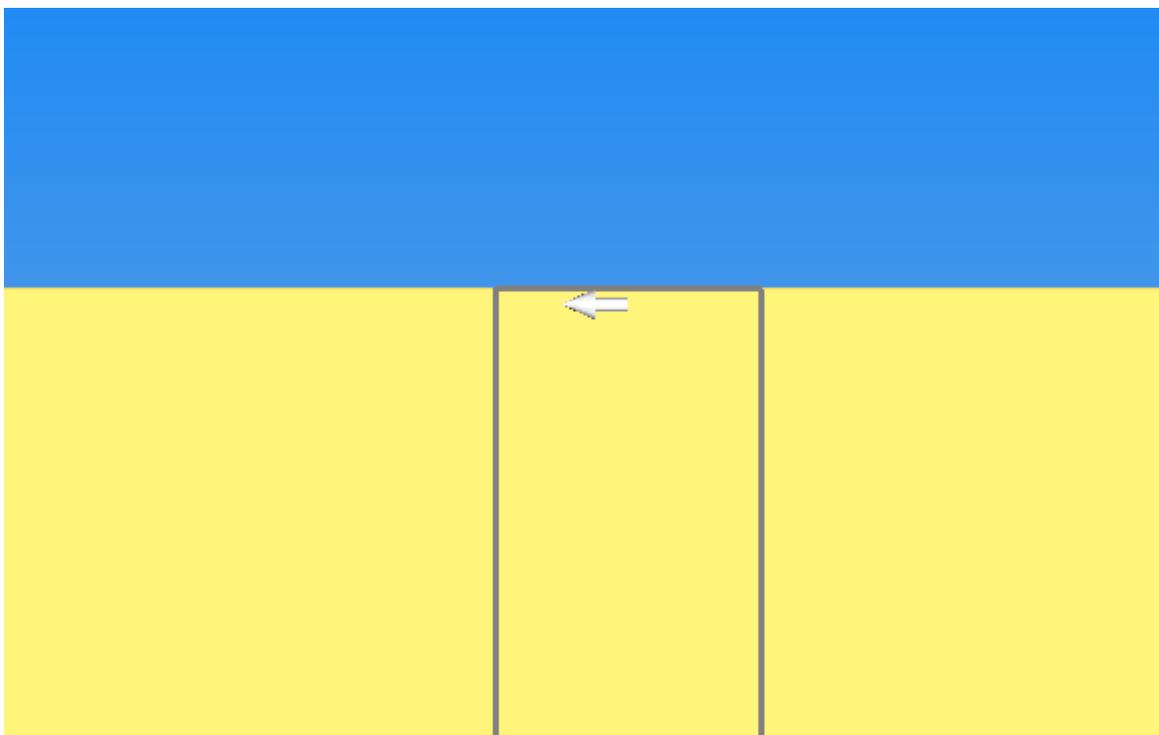


Figure 115 - Correctly adjusted Start Point

Considering the nature of the extracted pocket, if we applied a toolpath to the geometry the following would happen.

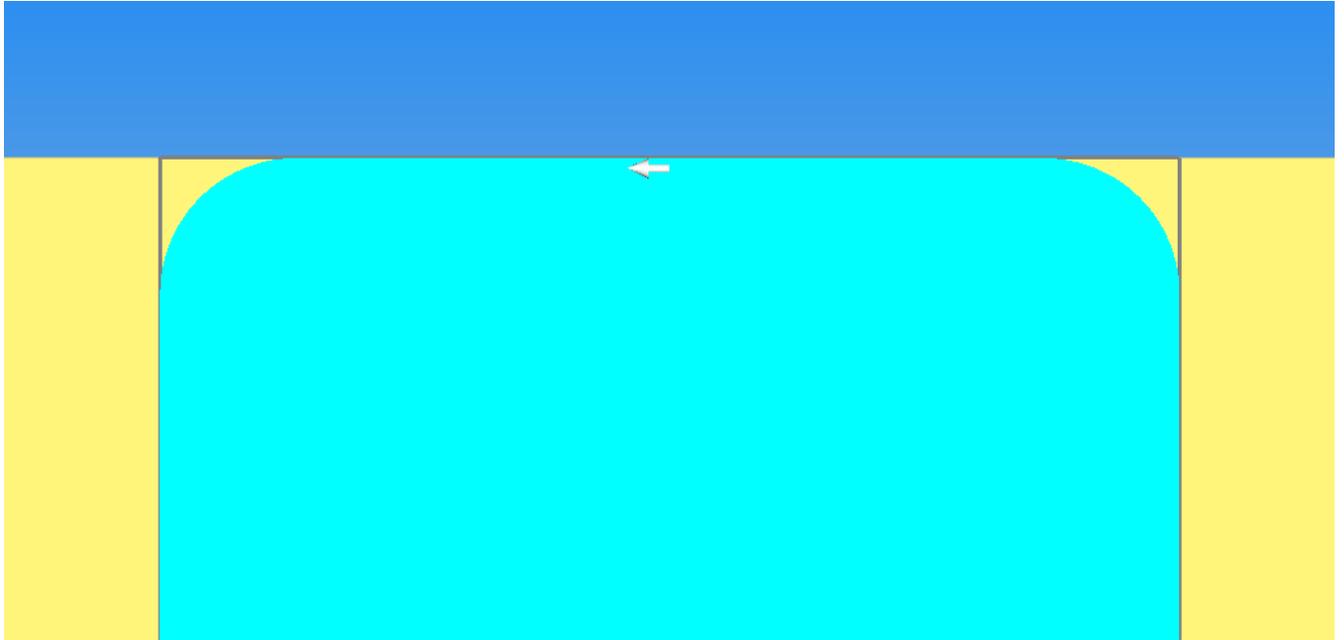


Figure 116 - Machining not created as desired

All of the machining is contained within the extracted geometry.

Setting Open Elements

To generate a more beneficial toolpath, we need to modify the existing geometry to allow the tool to pass across the open ends of the actual pocket geometry.

To do this we make use of the **MACHINE > Set/Unset Open Elements**  command.

With this command, you need to make sure that you know in which direction the Ghost Tool is pointing so that you can make the correct selection of elements.

Element selection requires the choice of a **First** and **Last** element of an area to set to Open.

In the example shown below, **<LClick>** the right hand element behind the Ghost Tool.

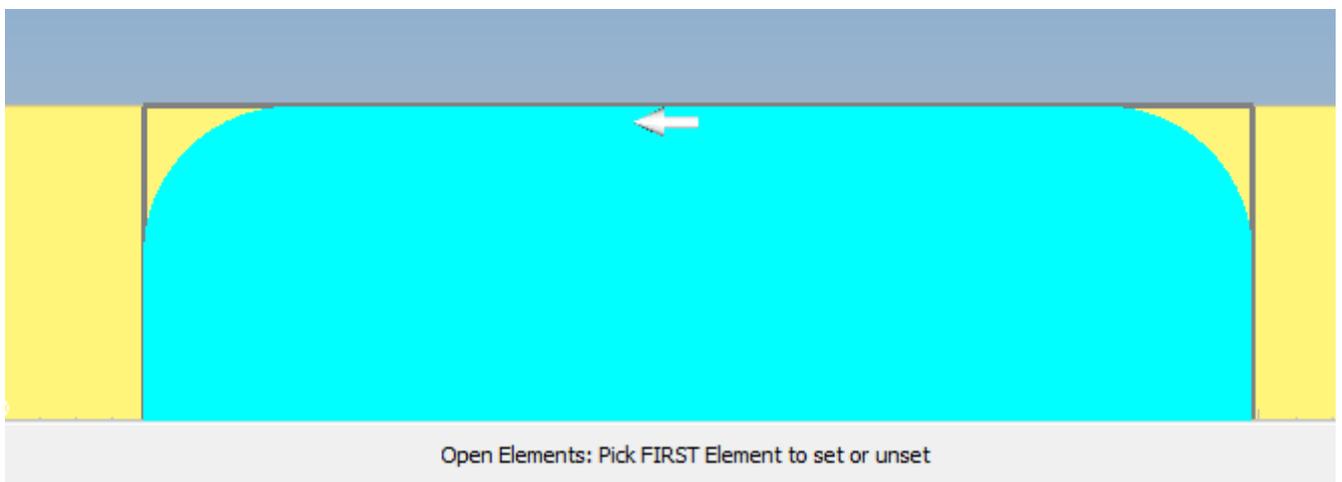


Figure 117 - First Element selected behind the Ghost Tool

For the Last element, **<LClick>** the left hand element in front of the ghost tool.

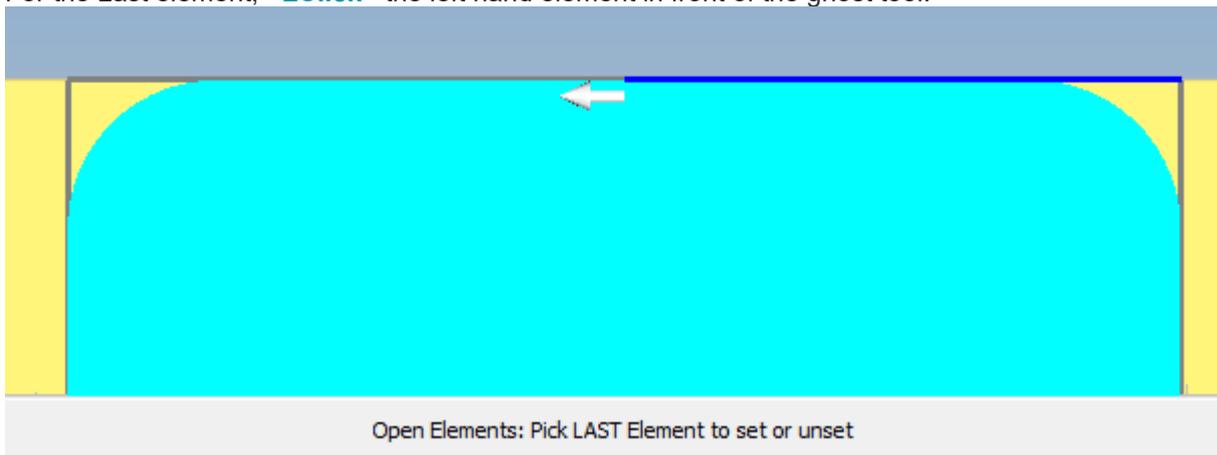


Figure 118 - Last Element selected in front of the Ghost Tool

For the opposite end and for any single element side, **<LClick>** the same element twice. Once to set it as the **First element**, then again to set it as the **Last element**.

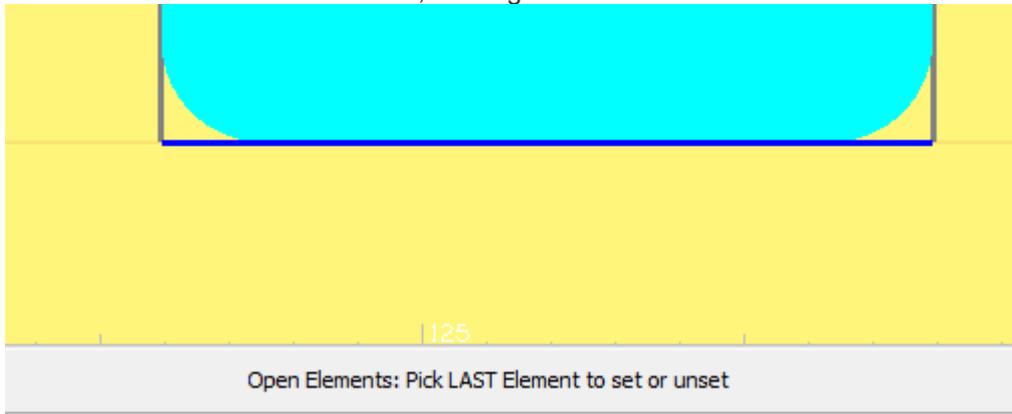


Figure 119 - **<LClick>** the same element twice for single sides

<RClick> to complete the process.

Using the **Update Toolpaths**  command from the top of the Operations Project Manager page will then adjust any toolpaths associated with the geometry and allow the tool to pass across the now open elements.



Figure 120 - Overlapped toolpath.

All overlap values are controlled using the **Overlap on Open Elements** dialogue found in **Pocketing** and **Rough & Finish** on the **Machining Data** pages.

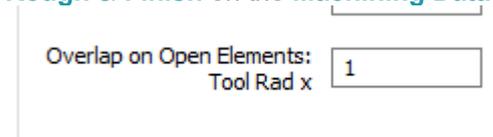


Figure 121 - Overlap on Open Elements

The default option value is always **x1** of the Tool Radius, but a more beneficial value would be slightly more, perhaps **x1.25** just to force the tool a little over the open element.

Right Slot

The **Contour from Picked Faces** is not suitable for the right slot as this would create a T shape because the slot and the rebate are the same depth.

For the right slot geometry select

SOLID MODEL EXTRACT > Contour from Picked Edges



The command line will prompt you to select the first edge (you can pick the top or bottom edge of the slot). The command line will prompt for further edges, as this is the only required edge, **<RClick>** to finish.

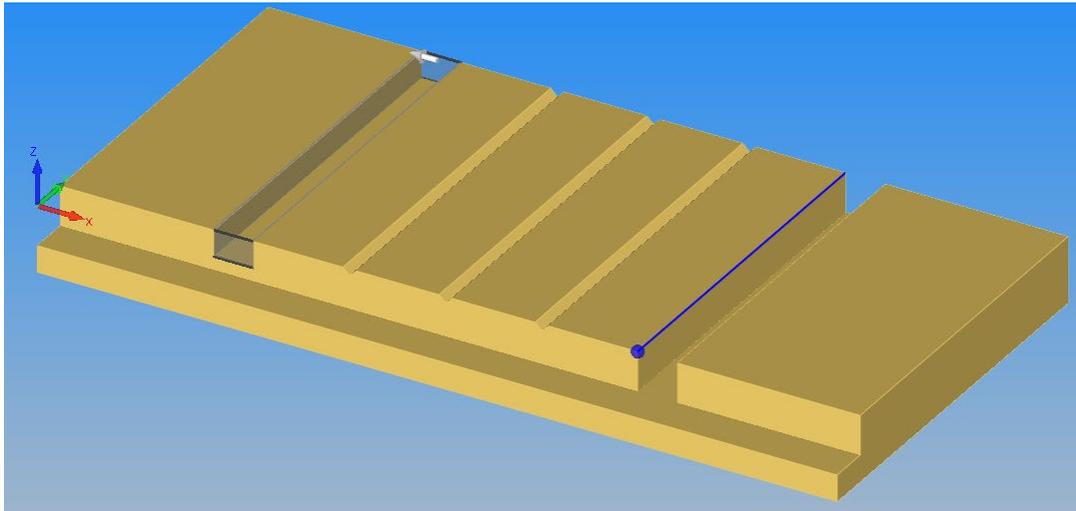


Figure 122 - Single edge selection for extraction

The geometry will be created, and the command line will prompt you to select an edge to mark the Bottom Z Level.

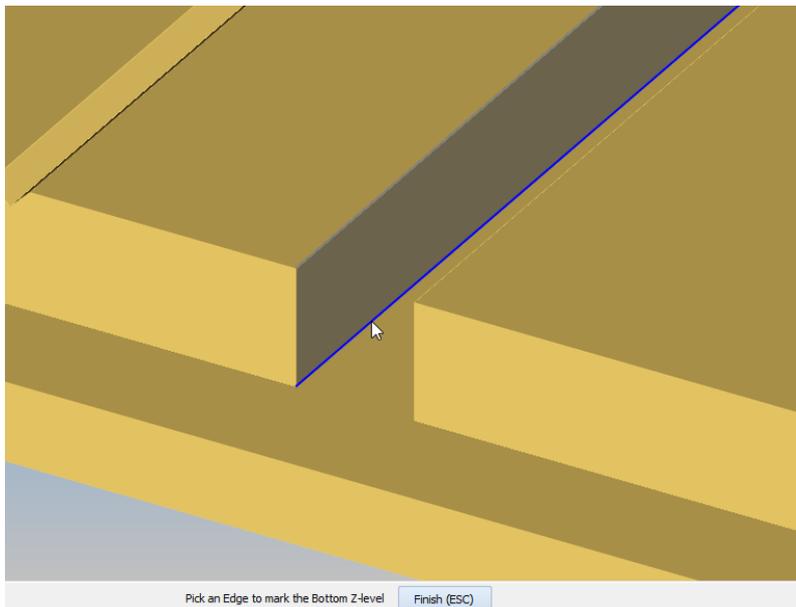


Figure 123 - Setting the Bottom Z Level

<LClick> on any edge that is on the same level as the bottom of the slot then **<RClick>** or click on **[Finish (ESC)]**.

The command line will prompt you to select an edge to mark the Top Z Level.

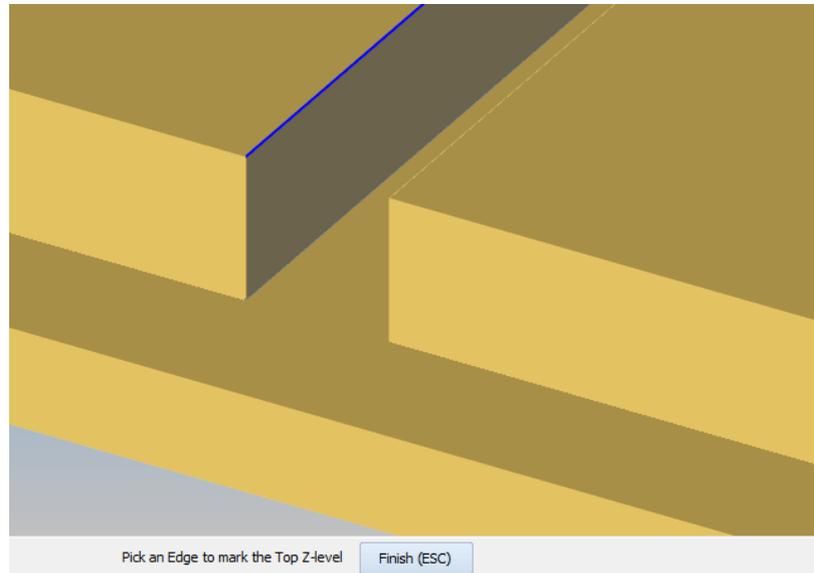


Figure 124 - Setting the Top Z Level

The Top level is where we need it to be, <RClick> or click on [Finish (ESC)]. The geometry is created on the **Open Areas** Layer.

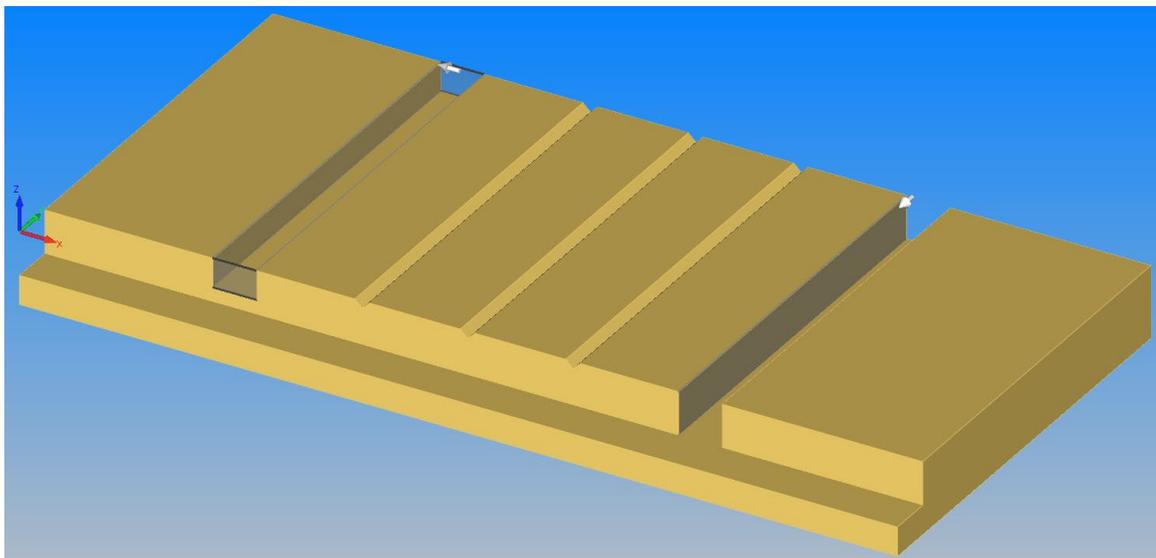


Figure 125 - Second slot detail extracted



The geometry for the left slot can also be created using this method if only one edge is required.



If the bottom edge of the slot is extracted, you can extract a U shape so the you can profile up and down the slot.



Remember to check the Tool Sides of any geometries that you extract using this method, they may or may not be correct for machining.

V Slots

The three V slots can all be extracted using the same technique as the second slot extraction method. Each slot must be selected and processed as an individual

SOLID MODEL EXTRACT > Contour from Picked Edges

The command line will prompt you to select the first edge (you can pick the top or bottom edge of the slot). The command line will prompt for further edges, as this is the only required edge, **<RClick>** to finish.

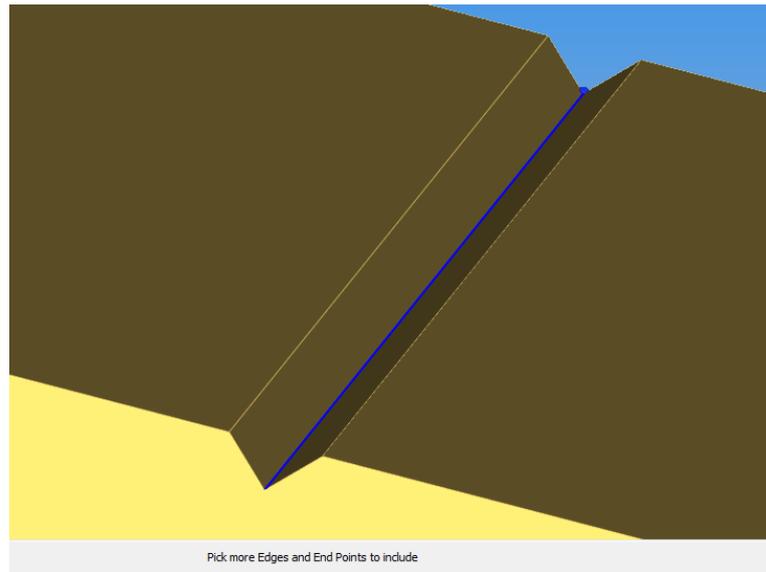


Figure 126 - Single edge selection for extraction

The geometry will be created, and the command line will prompt you to select an edge to mark the Bottom Z Level.



Figure 127 - Setting the Bottom Z Level

<LClick> on any edge that is on the same level as the bottom of the slot then **<RClick>** or click on **[Finish (ESC)]**.

The command line will prompt you to select an edge to mark the Top Z Level.

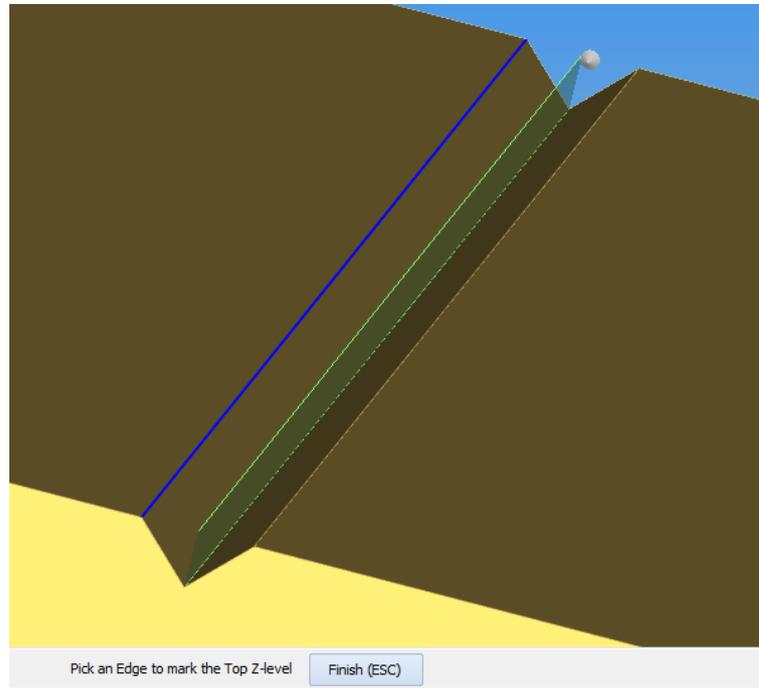


Figure 128 - Setting the Top Z Level

The Top level is where we need it to be, **<RClick>** or click on **[Finish (ESC)]**. The geometry is created on the **Open Areas** Layer.

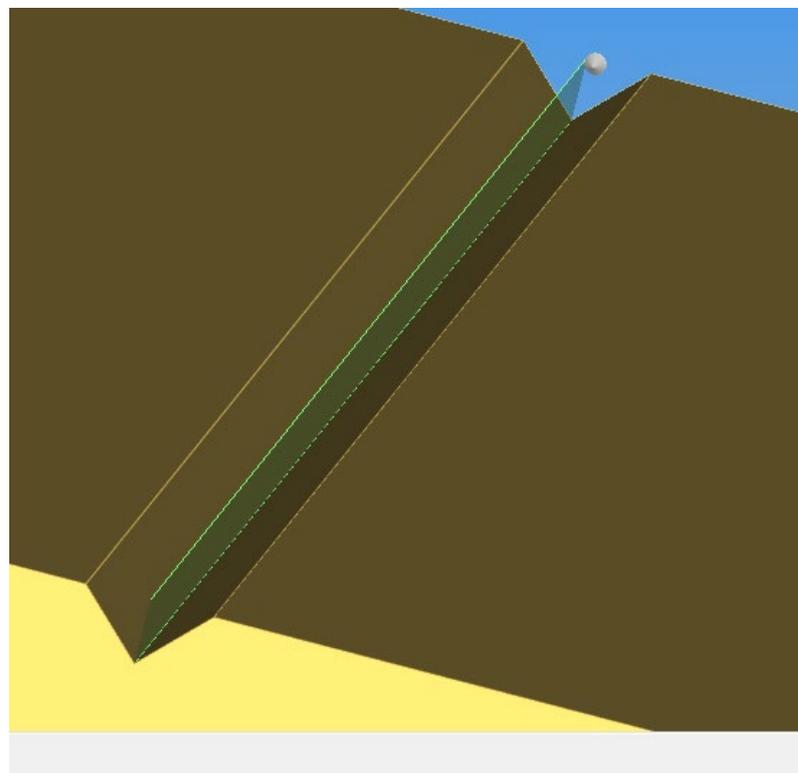


Figure 129 - Second slot detail extracted

The only consideration to give the slot details is which side of the line you will be machining on. Any of the extraction processes will place the ghost tool on one side of an extracted geometry, it is your responsibility to ensure that it is correct for manufacture.

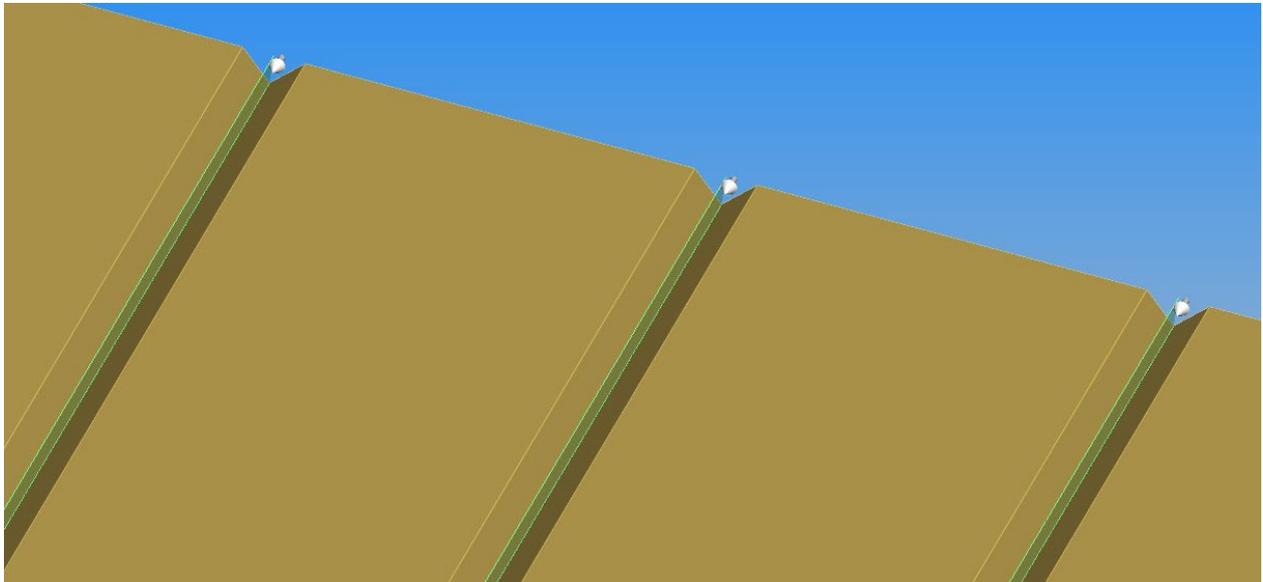


Figure 130 - V-slots as extracted

In this sample part, the tool will machine the slots along the centre of the extracted lines, so we must use the

Tool Directions  command to set the ghost tools on **Centre**.



Figure 131 - V-slots correct for manufacture

Front Rebate

The **Contour from Picked Edges** command cannot be used on the rebate because the right hand slot cuts the edge and there is no demarcation line across the lower face to select.

For this geometry, you need to manually draw the line required, snapping to end points **<F6>** of the model as required.

Auto snap will not function on the Solid Model corners, so use of the actual snap option, **<F6>**, is required.

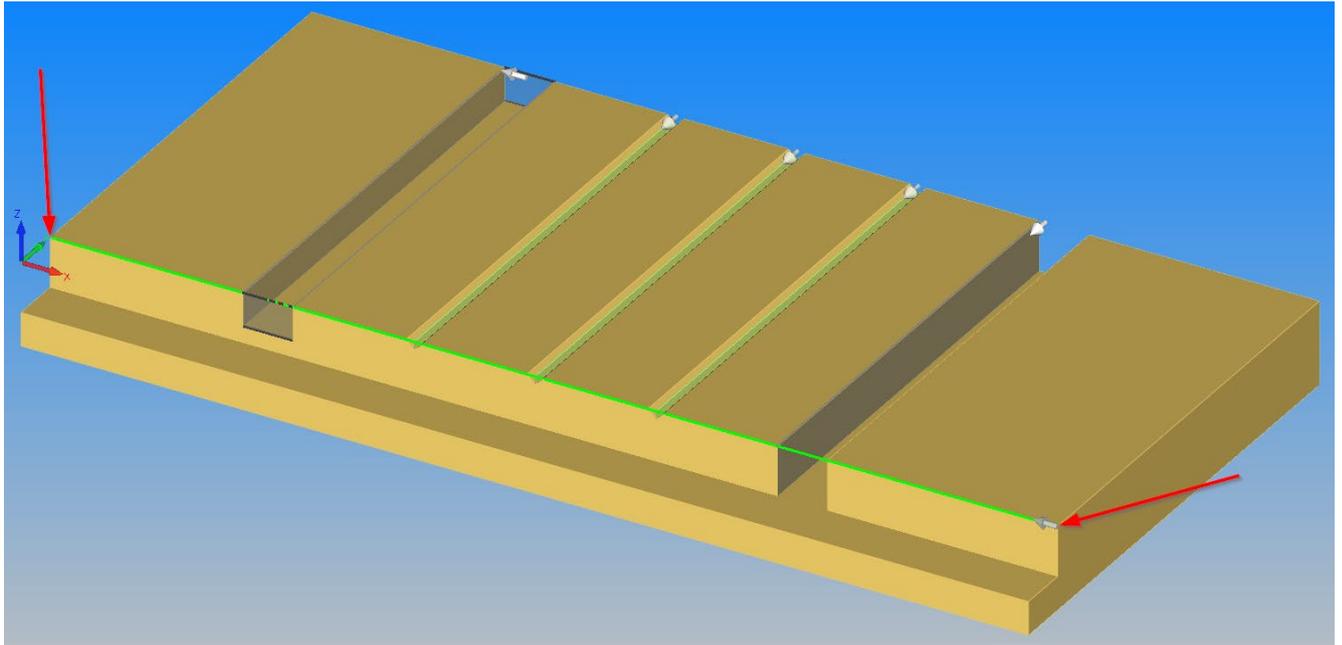


Figure 132 - Creating the manual geometry line

The geometry is drawn in Flat-Land and on the default **Geometry** layer.

It should also be noted that all of the geometries that you have extracted automatically have Z Levels assigned to them directly from the mode. Manually created geometries do not.

To set the Z levels of this geometry select

SOLID MODEL UTILITIES > Reset Geometry Z Levels 

1. **<LClick>** on the manually created geometry.
2. **<RClick>** to continue, then **<LClick>** to select an edge to mark the Bottom Geometry Z level.
3. **<RClick>** to continue.
4. **<LClick>** to select an edge to mark the Top Geometry Z level.
5. **<RClick>** to finish and the geometry Z levels will be set.



When setting the Geometry levels for manually created geometry, you **must** select both the **Bottom** and **Top** level options, if you use the **[Finish (ESC)]** option on either question, the levels will not be altered.

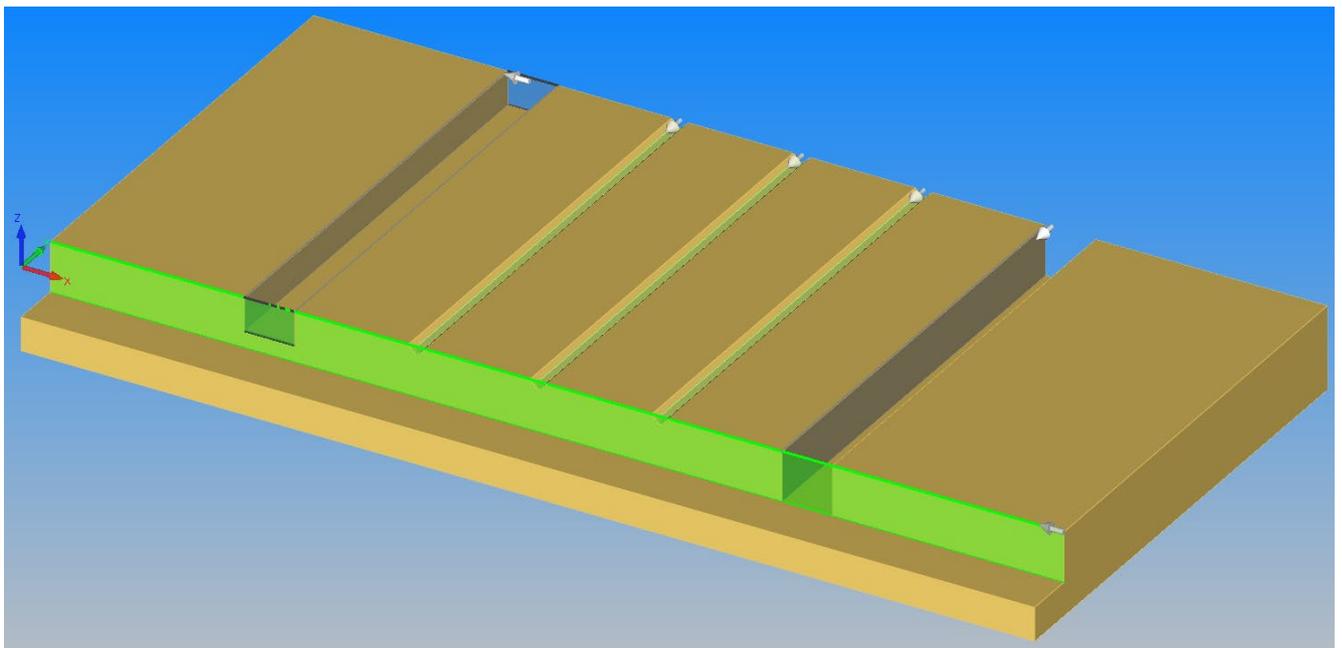


Figure 133 - Geometry Z Levels assigned to the manually added line



Note that the tool side is still set on **Centre** as this geometry was created manually and not extracted, adjust accordingly.

Outline

To extract the outline boundary select

SOLID MODEL EXTRACT > Extract Solid Body Outlines

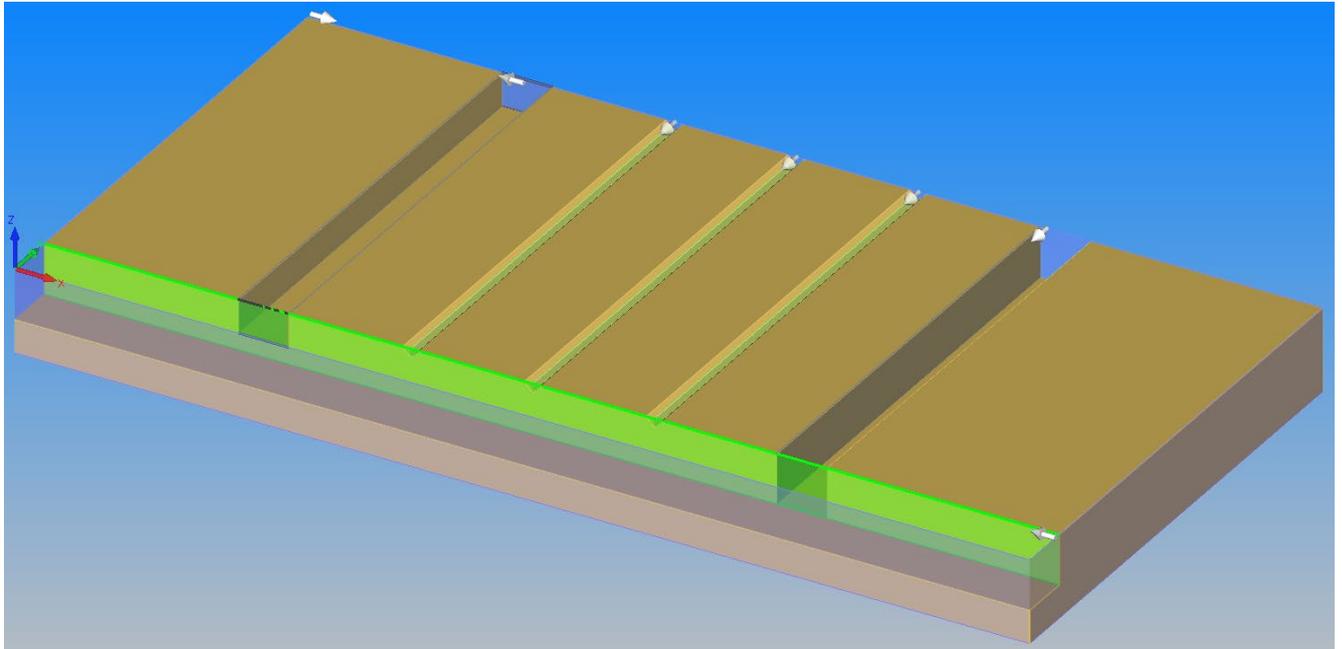


Figure 134 - Extracted Solid Body Outline from the previous example

This creates all geometries by looking down the Z axis of the current plane and creating geometry where it is possible to see the background, with tool side and direction set for correct machining.



Geometries will also be created for any through apertures and holes.



This process will include every solid body on a multiple body drawing, if you do not need to trace around some items, turn them off in the **Layers** project manager page.

Create Slot Geometries without extraction

Because the global datum is on the top of the part the two slots can be created by drawing two rectangles snapping **<F6>** to two diagonal corners, resetting the geometry Z levels from the model edges

Then using **MACHINE > Set/Unset Open Elements**  to open the end elements of the slots.
Alternatively, you could draw 3 lines to form a U shape.

If the datum is not at the top of the part, you must ensure you draw the geometries on a work plane or reset the geometry Z levels from the model to suit the machine datum.

#6 Wreathed Handrail

3D Minimal Bounding Box

Select **HOME > Input CAD**  and the following dialogue is displayed.

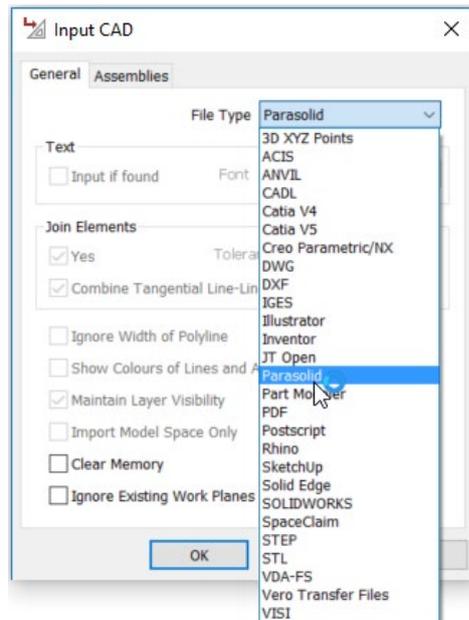


Figure 135 - Input CAD set for Parasolid

Set the file type to **Parasolid**, and then select **[OK]**.
The file selection dialogue is displayed.

Navigate to “...ALP TRG 114 Feature Extraction 2020\Examples\Parasolid Examples\” and select the “#6 Wreathed Handrail.x_t” then select **[Open]**.
The model is imported and displayed on screen.

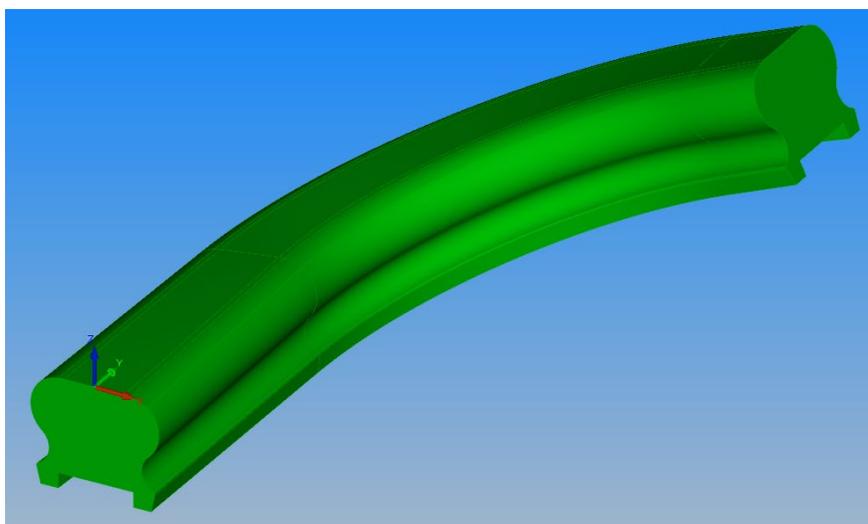


Figure 136 – Wreathed Handrail sample file.

With a part such as this handrail section, there is no easy way of aligning this to suit manufacture by any of the methods that we have used so far.

Auto Aligning will give this type of set up.

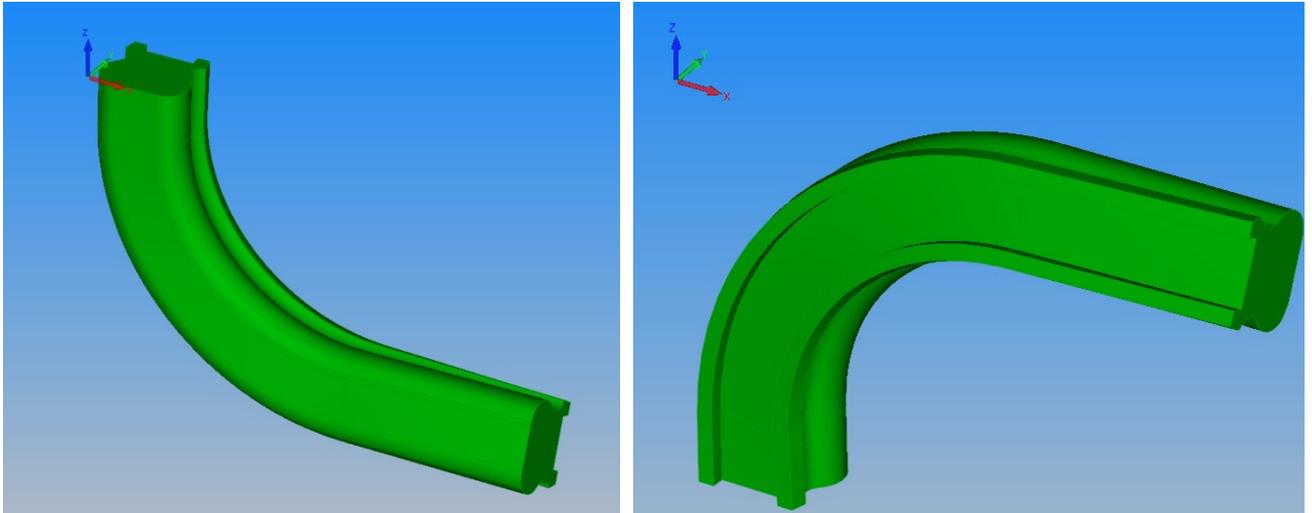


Figure 137 - Auto Alignment on Wreathed Handrail

Using the **Align Part command** will be even more difficult as there are only the ends that are the flat faces plus the edges of the rebate section are the only straight edges to align with the X axis.

The result would be exactly the same as using Auto Align, but more time consuming.

Method 1 accepting default alignment

To work around the restrictions on parts like this we use the command

GEOMETRY > Special Geometries > Minimal 3D Bounding Box 

This command works out the smallest 3D cuboid shape that will fit around the extremities of your part and offers the option to align this geometric shape and the item contained within it.

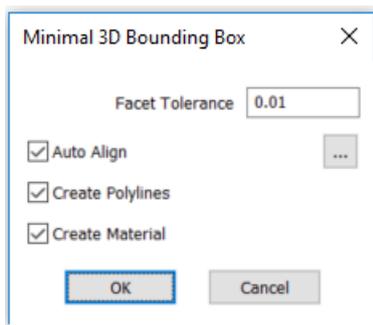


Figure 138 - Minimal 3D Bounding Box initial dialogue

- Auto Align** Uses the generated bounding cuboid, and the item within, to align according to your chosen setting from the Feature Extraction Settings dialogue.
- Create Polylines** Creates the twelve edges of the bounding box as polylines.
- Create Material** Uses the bounding box geometry as the basis for a material to enclose the part.

Once you **<LClick> [OK]** you are asked to select the item around which the bounding box will be created.

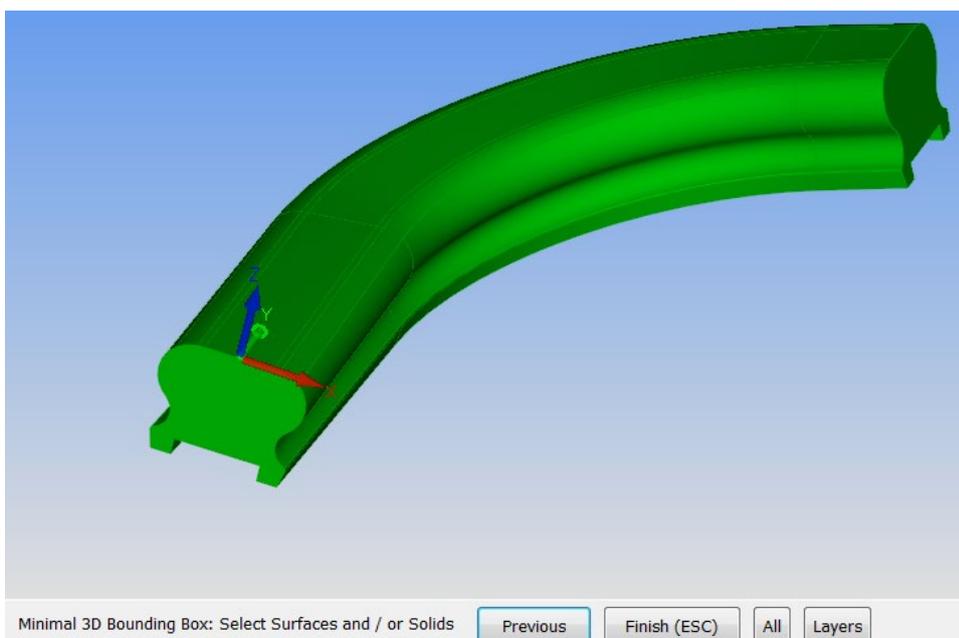


Figure 139 - Minimal 3D Bounding Box selection options

<LClick> the solid, then **<RClick>** to continue.

Once you continue you are asked for additional information based on your requirements.

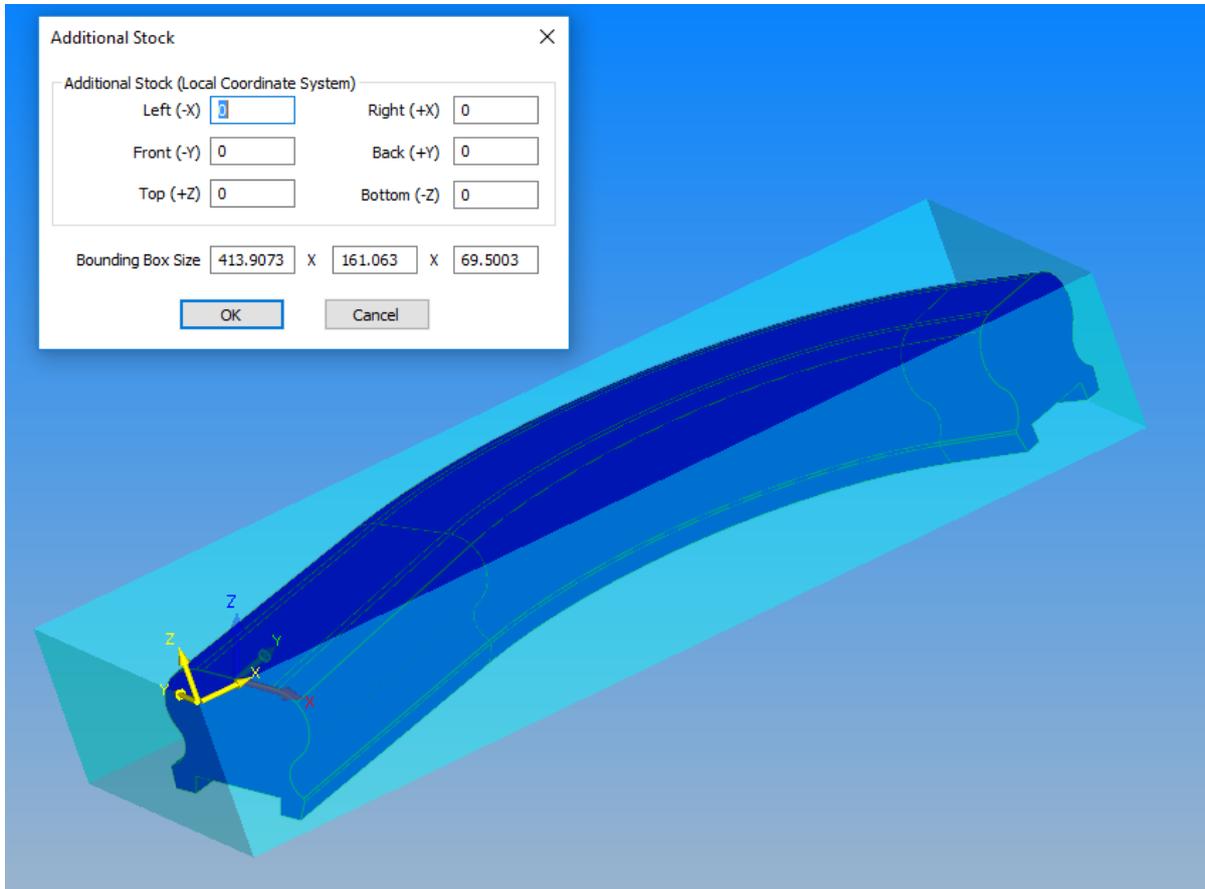


Figure 140 - Minimal 3D Bounding Box additional information dialogue

The four boxes for additional stock allow for a more adaptable stock set up than using the options **3D> Set Material** or **3D > Auto Set Material**.

Adding values to these six boxes will allow you to offset the faces on all six sides by similar or totally differing amounts.



Note that the naming convention of the boxes is directly related to the attitude of the generated **Work Plane** on the part.



The Work Plane indicated in the image above is set to the **Top** of the part, so **Right** is to the end of the X Axis indicator of the yellow Work Plane axis marker.

The Bounding Box Size gives a definite size value for the enclosed item, making deciding on material sizes simpler, or order material easier.

The values are **X x Y x Z** according to the attitude of the generated **Work Plane**.



Note that these values cannot be manually altered, only adjusted by entering the values in the six sides stock options.

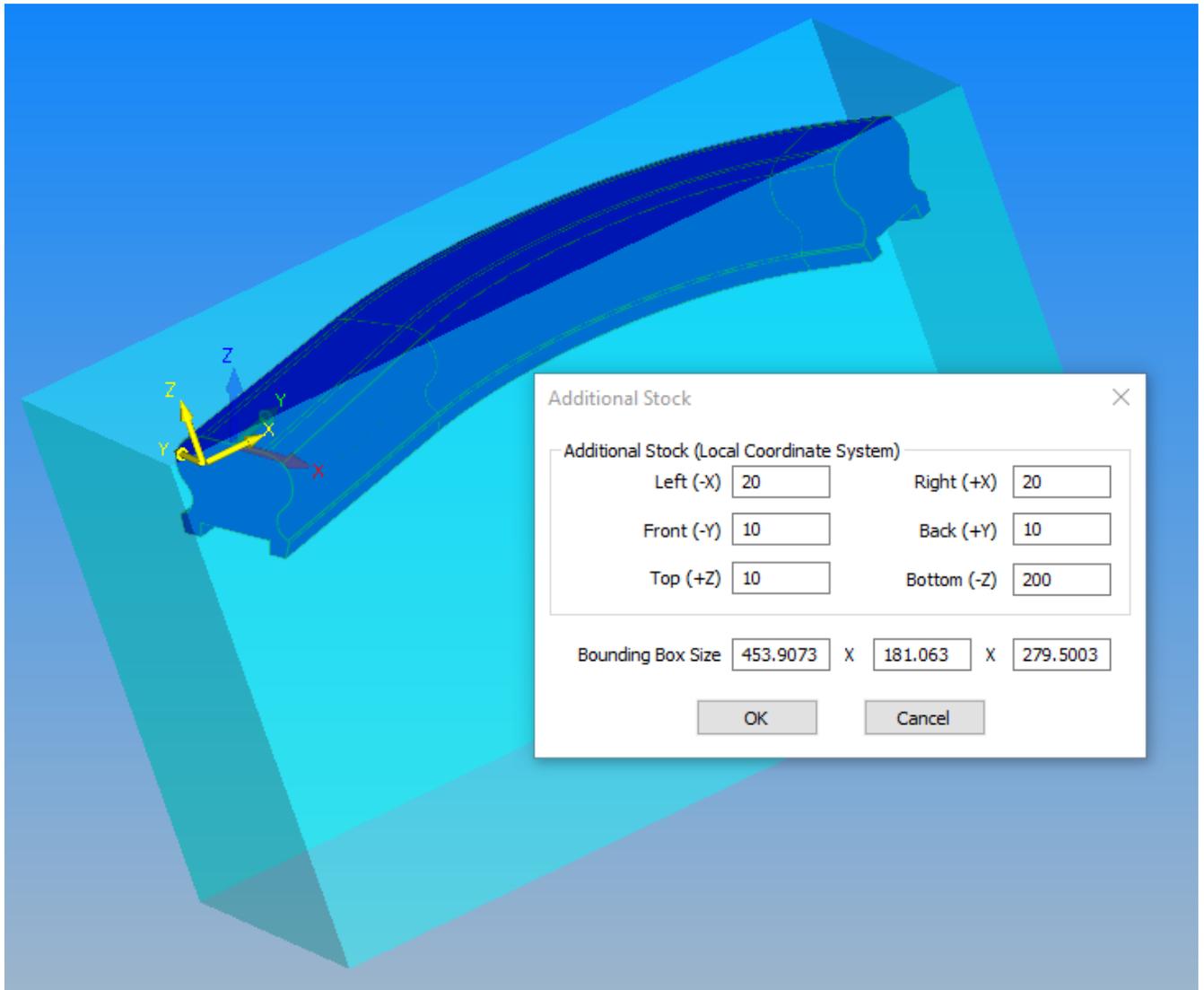


Figure 141 - Added values to increase the size

Notice how the values of the defined box alter according to the stock values.

Once you have the correct values entered, <LClick> [OK] to complete.

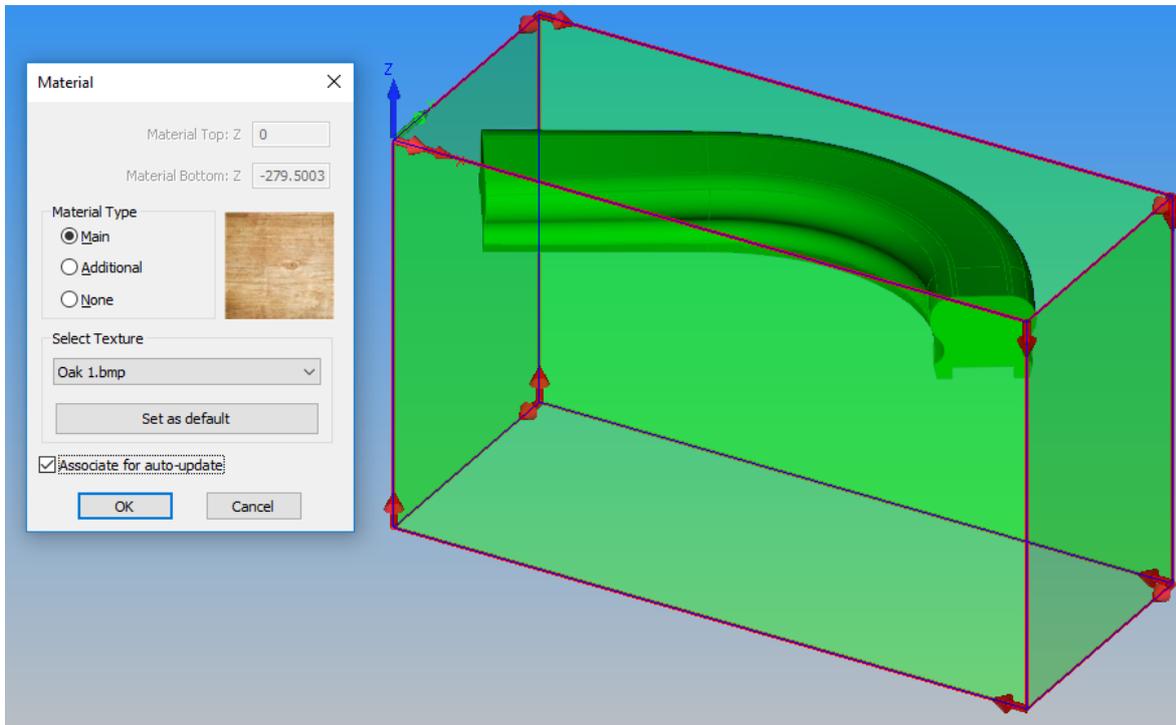


Figure 142 - Aligned part and Material dialogue

Once you have aligned the part from the previous dialogue, you are requested for information regarding the material and its set up.

- Main** Sets the geometry to be the main material used in the simulation process.
- Additional** Sets the geometry as an additional item, for example as a riser block.
- None** Removes the material option completely and leaves the bounding box as geometry only.

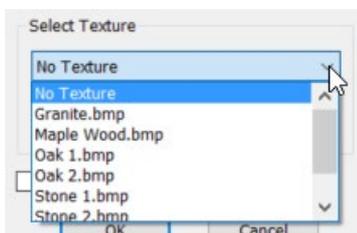


Figure 143 - Material Texture options

The Texture drop down offers you the opportunity to set a rendered image to the part to simulate actual materials.

A small selection is installed by default, there is nothing preventing you from adding more to your collection located in "...LICOMDAT\Textures.Alp"

Associate for Auto Update will allow ALPHACAM to store first operation data from the material and then apply this to any secondary operations as a new material to work with.

Options in the machining cycle must be set correctly to make use of this functionality.

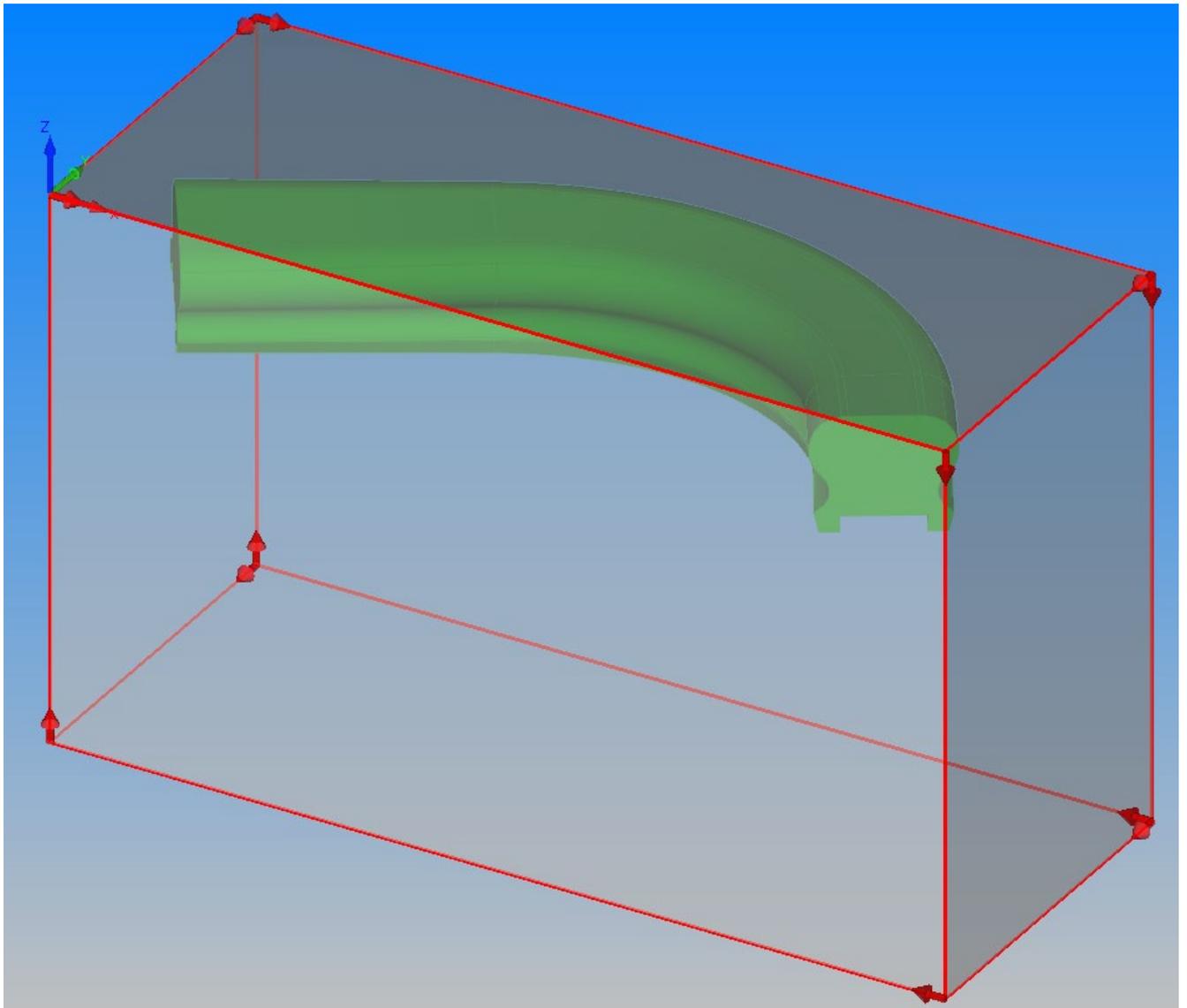


Figure 144 - Completed set up

The above image shows all of the options active,

- Auto Align,
 - Create Polylines
- and
- Create Material

It should be noted that as a simple exercise in the use of the command 3D Minimal Bounding Box, we have seen how it can be used to align inconveniently shaped parts easily.

You do have to take care that the results are what you need, if you wish to machine the rebate channel first, then the part in the image above is incorrectly set up.

Method 2 manual adjustment to the alignment

Select **HOME > Input CAD**  and the following dialogue is displayed.

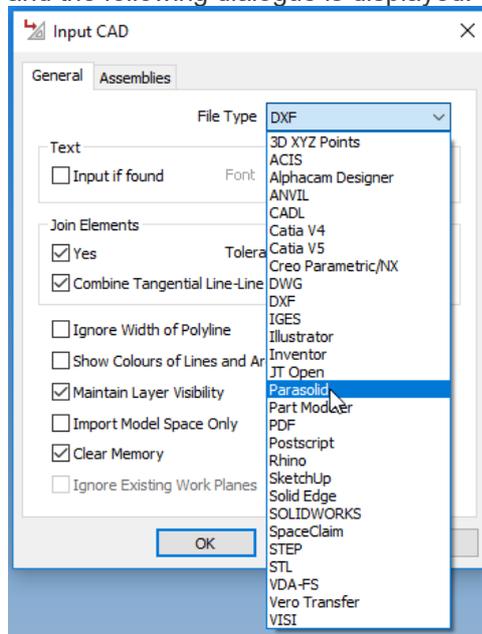


Figure 145 - Input CAD set for Parasolid

Set the file type to **Parasolid**, and then select **[OK]**.

The file selection dialogue is displayed.

Navigate to “...ALP TRG 114 Feature Extraction 2020\Examples\Parasolid Examples\” and select the “#6 Wreathed Handrail.x_t” then select **[Open]**.

The model is imported and displayed on screen.

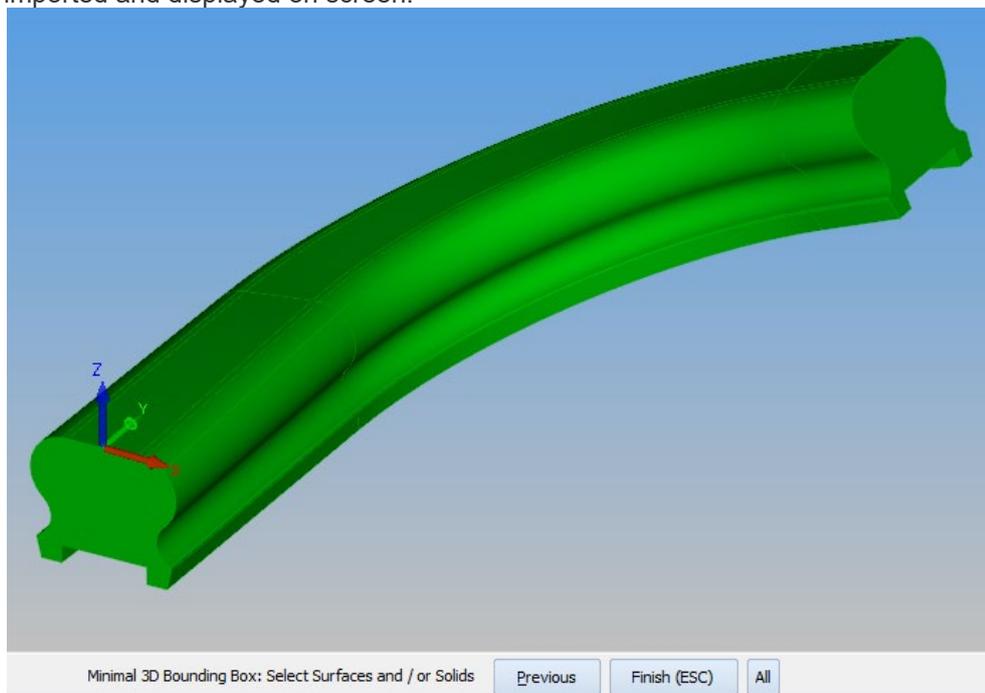


Figure 146 - Wreathed Handrail sample file

Use the command

GEOMETRY > Special Geometries > Minimal 3D Bounding Box 

This time, only activate the **Auto Align** option.

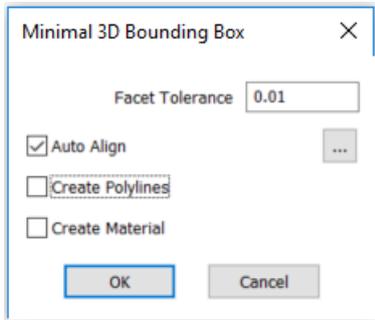


Figure 147 - Minimal 3D Bounding Box initial dialogue

Once you **<LClick> [OK]** you are asked to select the item around which the bounding box will be created.

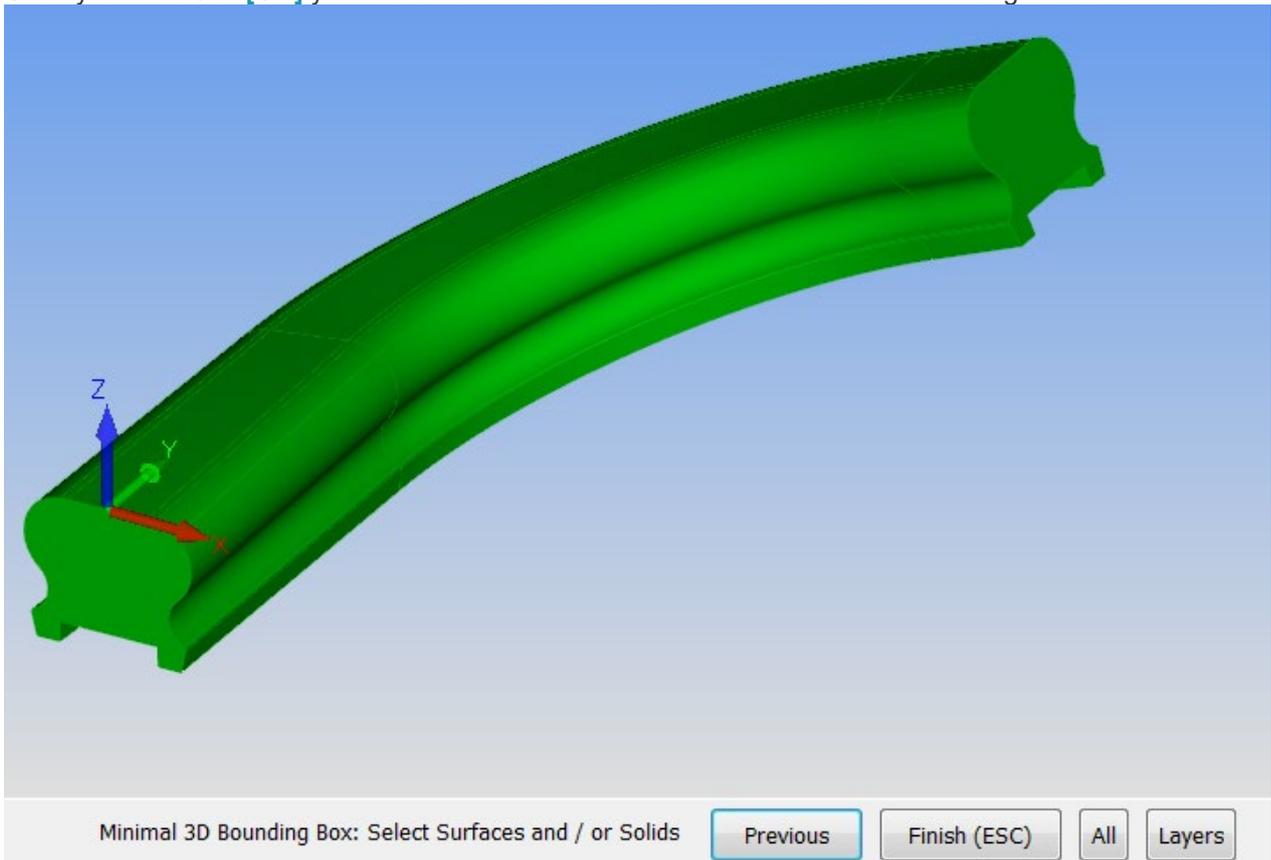


Figure 148 - Minimal 3D Bounding Box selection options

<LClick> the solid, then **<RClick>** to continue.

Once you continue you are asked for additional information based on your requirements.

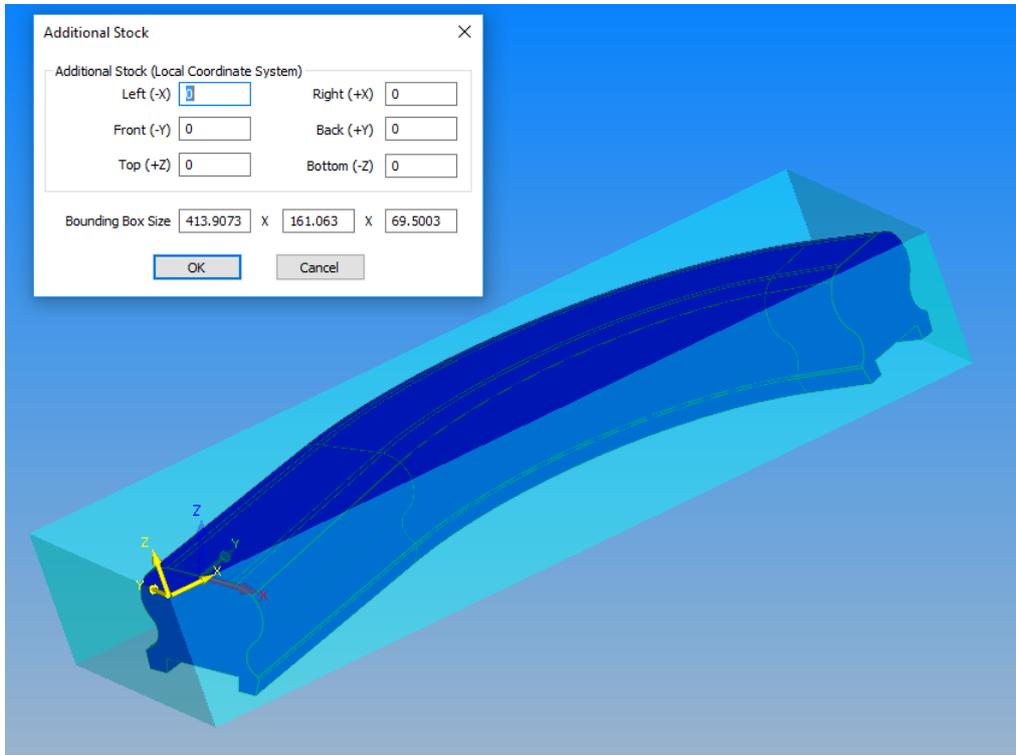


Figure 149 - Minimal 3D Bounding Box additional information dialogue

For this example, leave all the **Additional Stock** options set to zero, we will be using the bounding box as a guide to manipulate the part, after which we can then add a material if we require. **<LClick>**. **[OK]** to continue.

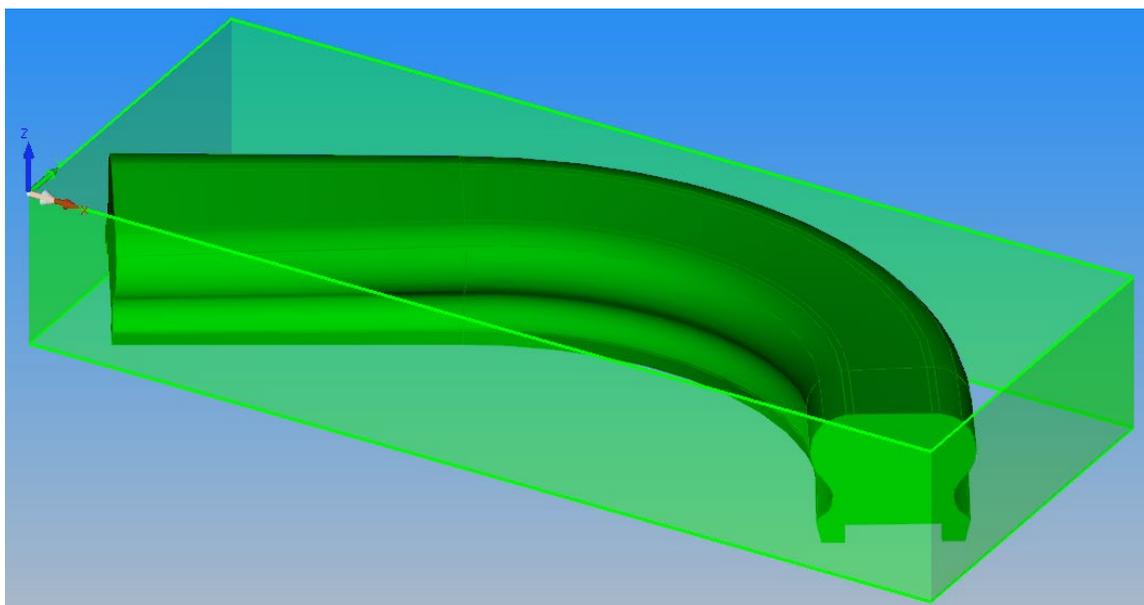


Figure 150 - Aligned part with 3D Minimal Bounding Box geometry

Use the command **EDIT > Rotate** .

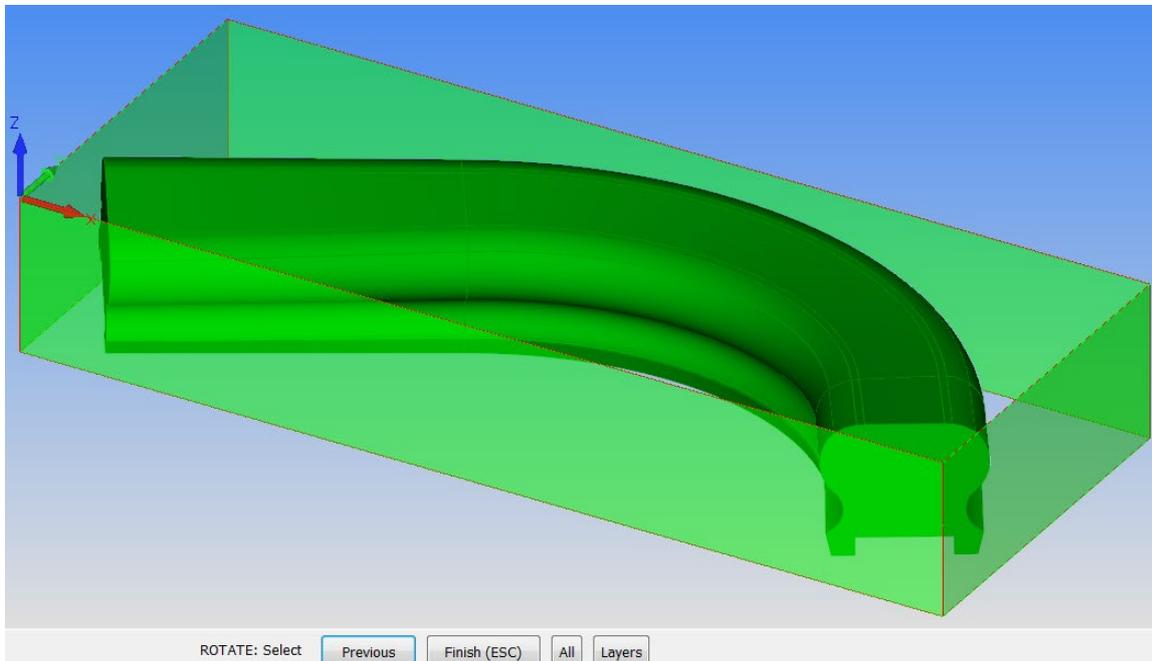


Figure 151 - Items to rotate

When prompted, **<LClick> [All]**, then **[Finish (ESC)]**.

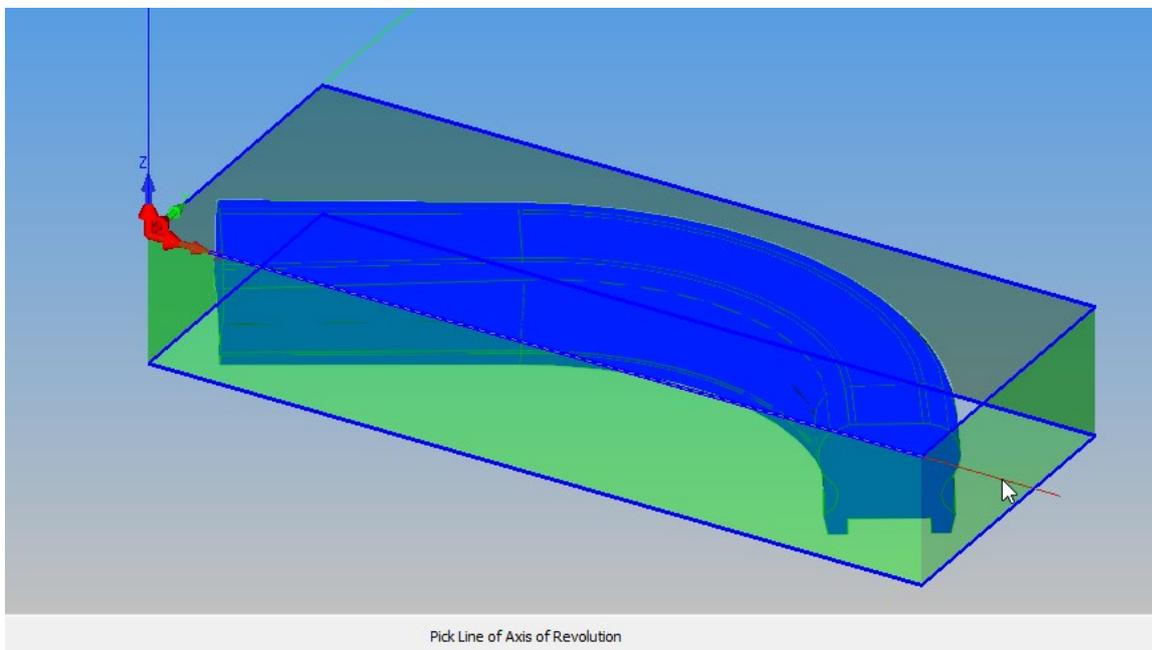


Figure 152 - Select that axis to rotate around

As this is a 3D part, you are asked for an **Axis of Rotation**.
We wish the rebate face to be uppermost, so **<LClick>** on the **X axis** polyline.

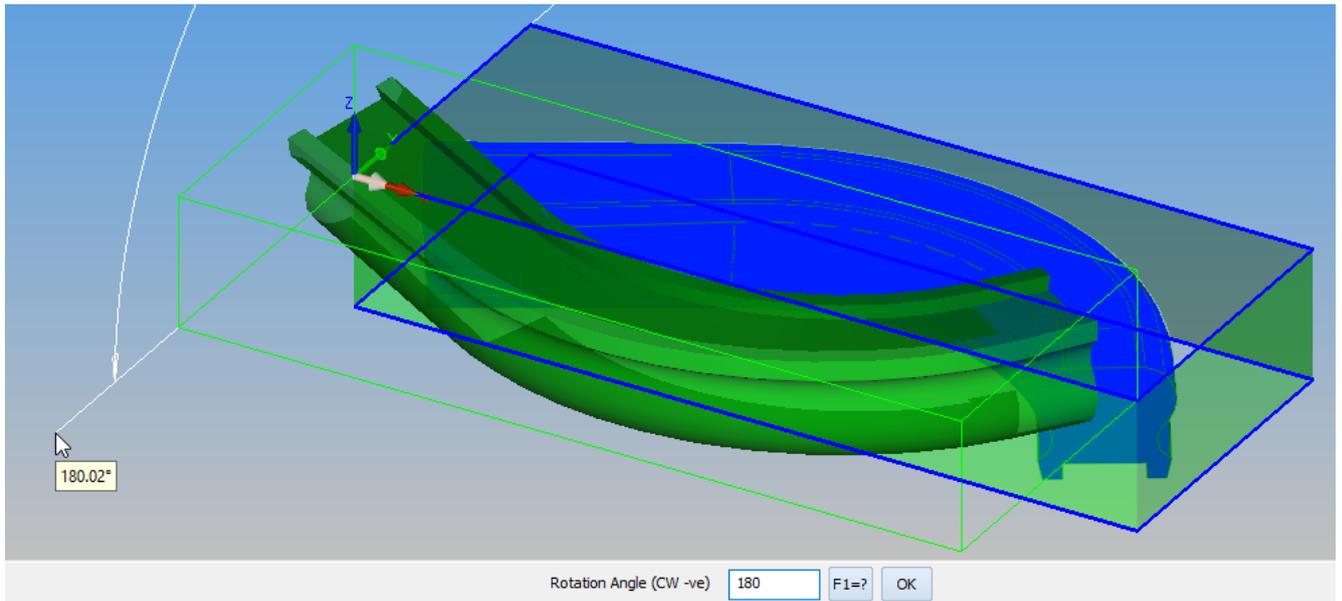


Figure 153 - Degrees of arc to rotate by

Enter **180** as the value for the angle. **<LClick> [OK]** to continue.

When prompted, enter **1** as the number of repeats, **<LClick> [OK]** to continue, then **<RClick>** to finish.

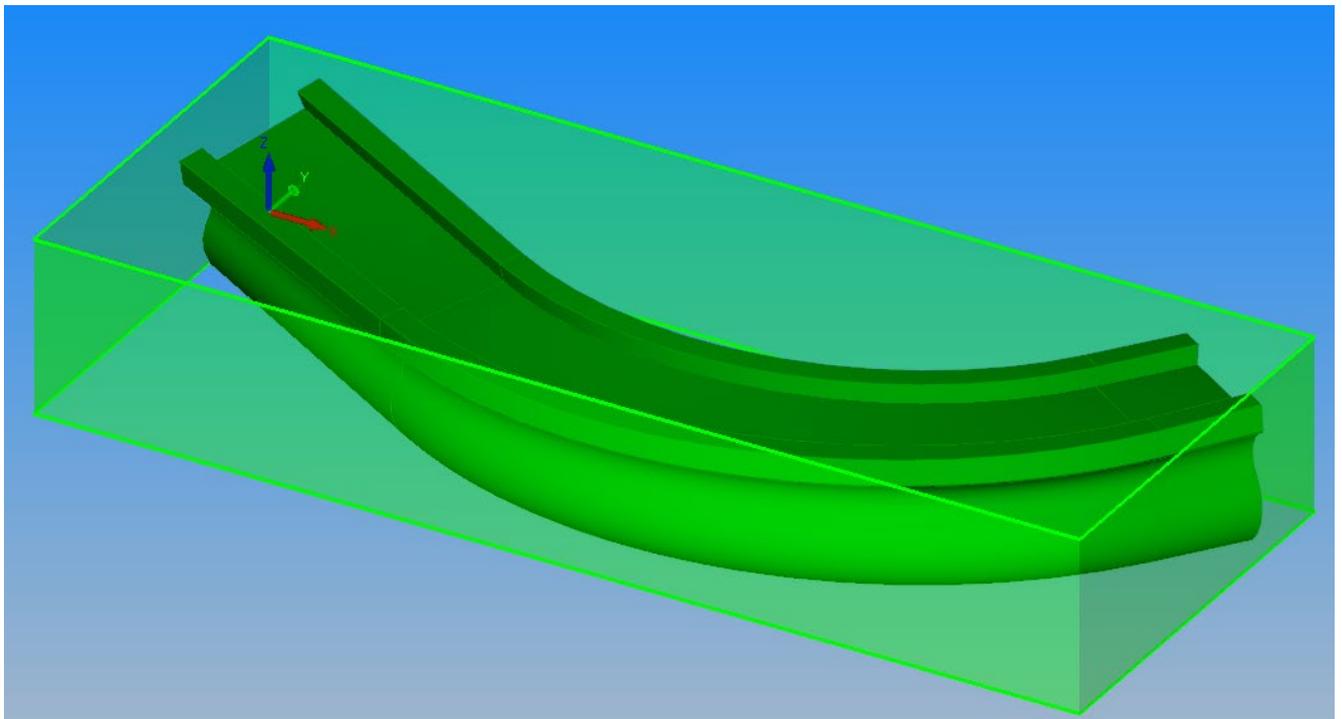


Figure 154 - Rotated part.



On some parts, free rotation maybe useful by using the mouse pointer to rotate the part to a suitable angle.

Use the command **EDIT > Move** 

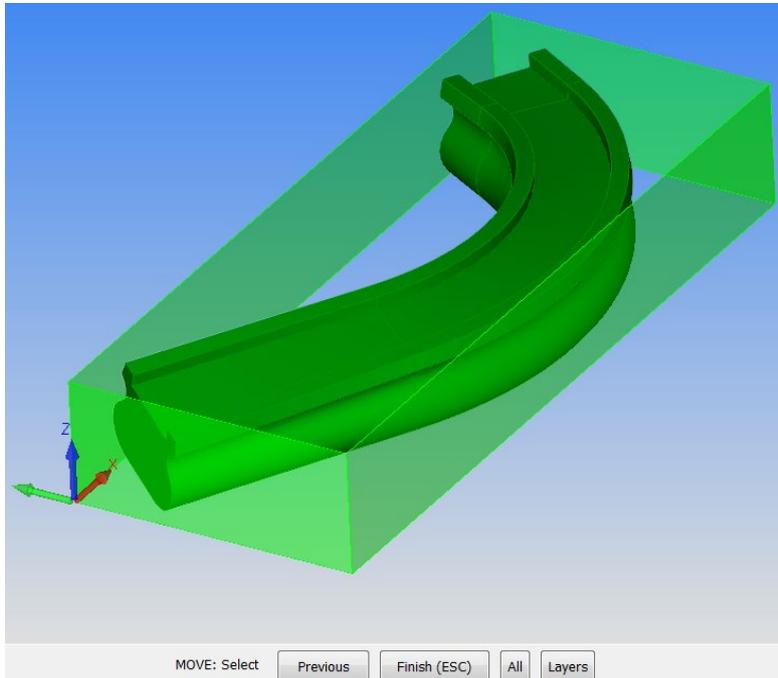


Figure 155 - Items to Move

When prompted, **<LClick> [All]**, then **[Finish (ESC)]**.

When you are asked for the Base Point, this will be the **Top, Front Left** corner of the bounding geometry. As this is a 3D transformation, Auto Snap **<F2>** cannot locate the Top Front Left corner of the geometry, so you will need to use **<F6>** End of to pick the point.

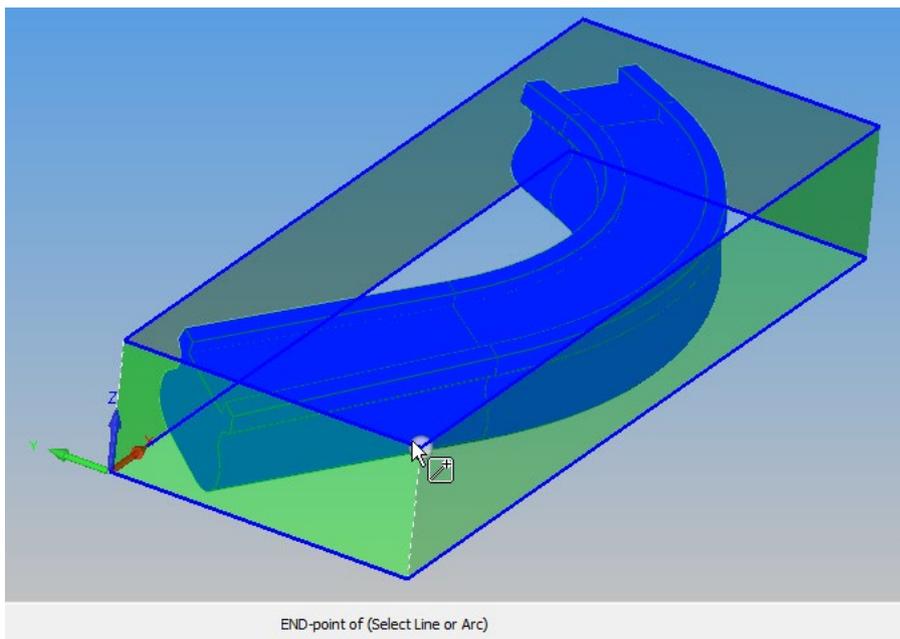


Figure 156 - Selecting the correct corner as the base point

Finally enter the required location for this chosen point.
In the case of this example we will enter zero into all the boxes.

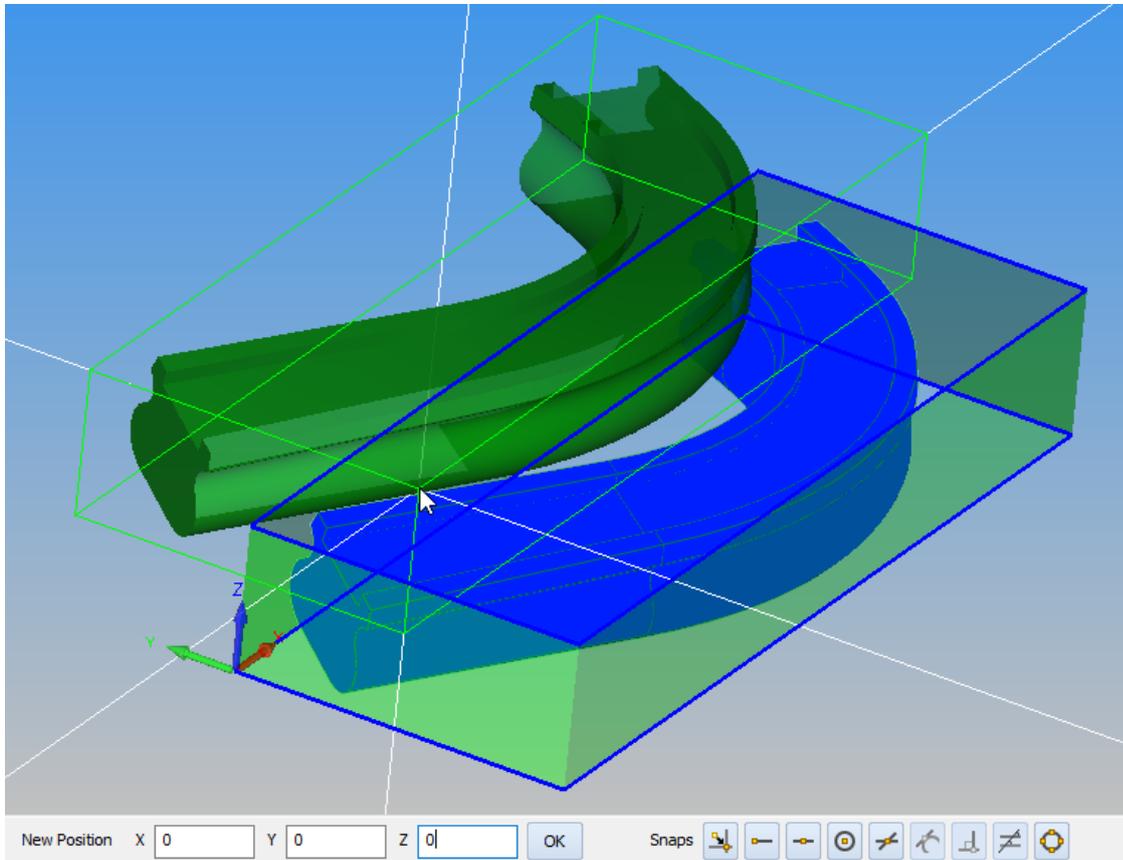


Figure 157 - Entering the new location

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

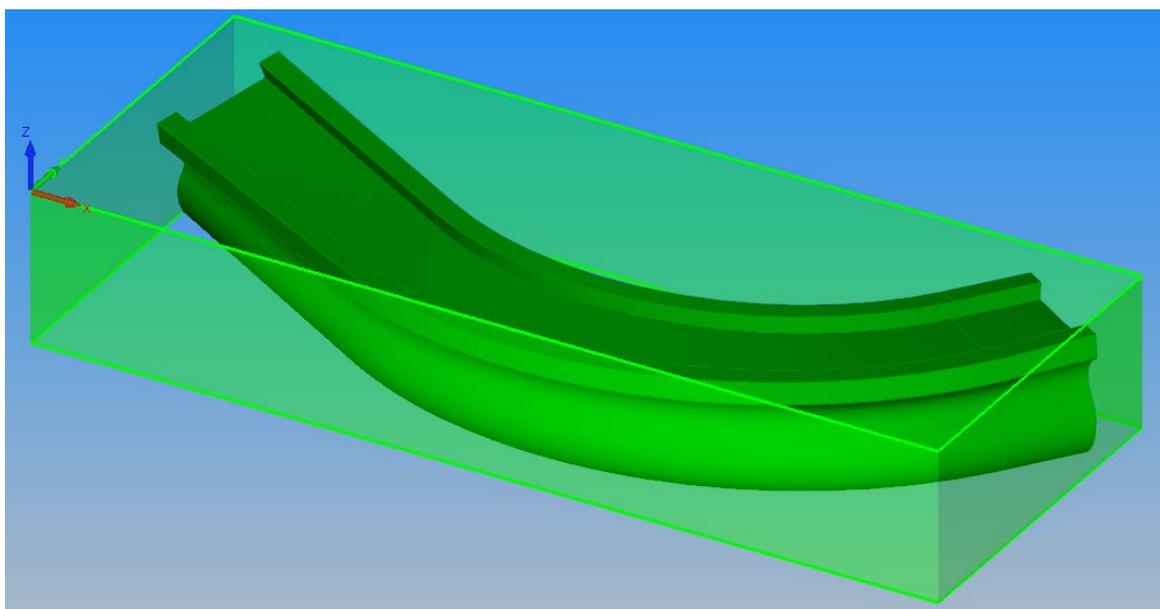


Figure 158 - Correctly set up part

#7 Open Pocket Block

Import the model

Select **HOME > Input CAD**  and the following dialogue is displayed.

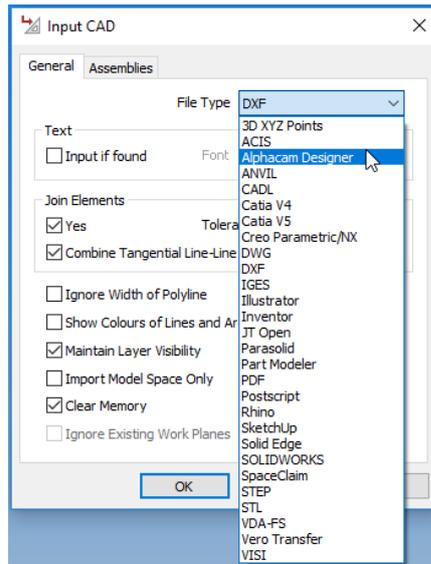


Figure 159 - Input CAD set for ALPHACAM Designer

Set the file type to **Alphacam Designer**, and then select **[OK]**.

The file selection dialogue is displayed.

Navigate to “...ALP TRG 114 Feature Extraction 2020\Examples\Alphacam Designer Examples\” and select the

“**#7 Open Pocket Block.vdf**” then select **[Open]**.

The model is imported and displayed on screen.

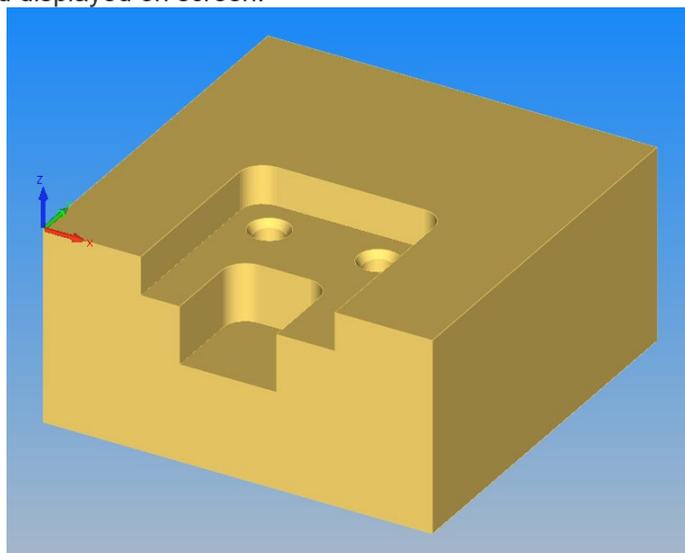


Figure 160 - Open Pocket Block sample file

Align the component if required.

Automatic Extraction of Geometries

The following section of the manual provides an insight into the methods used when the **Use Open Air Pocket Method** is switched off.

The geometries are extracted Ok if the **Use Open Air Pocket Method** is switched on. Extract the features from the model using

SOLID MODEL EXTRACT > Automatic Extraction  ensuring that the **Use Open Air Pocket Method** is not active.

Turn off the model and ghost tools.

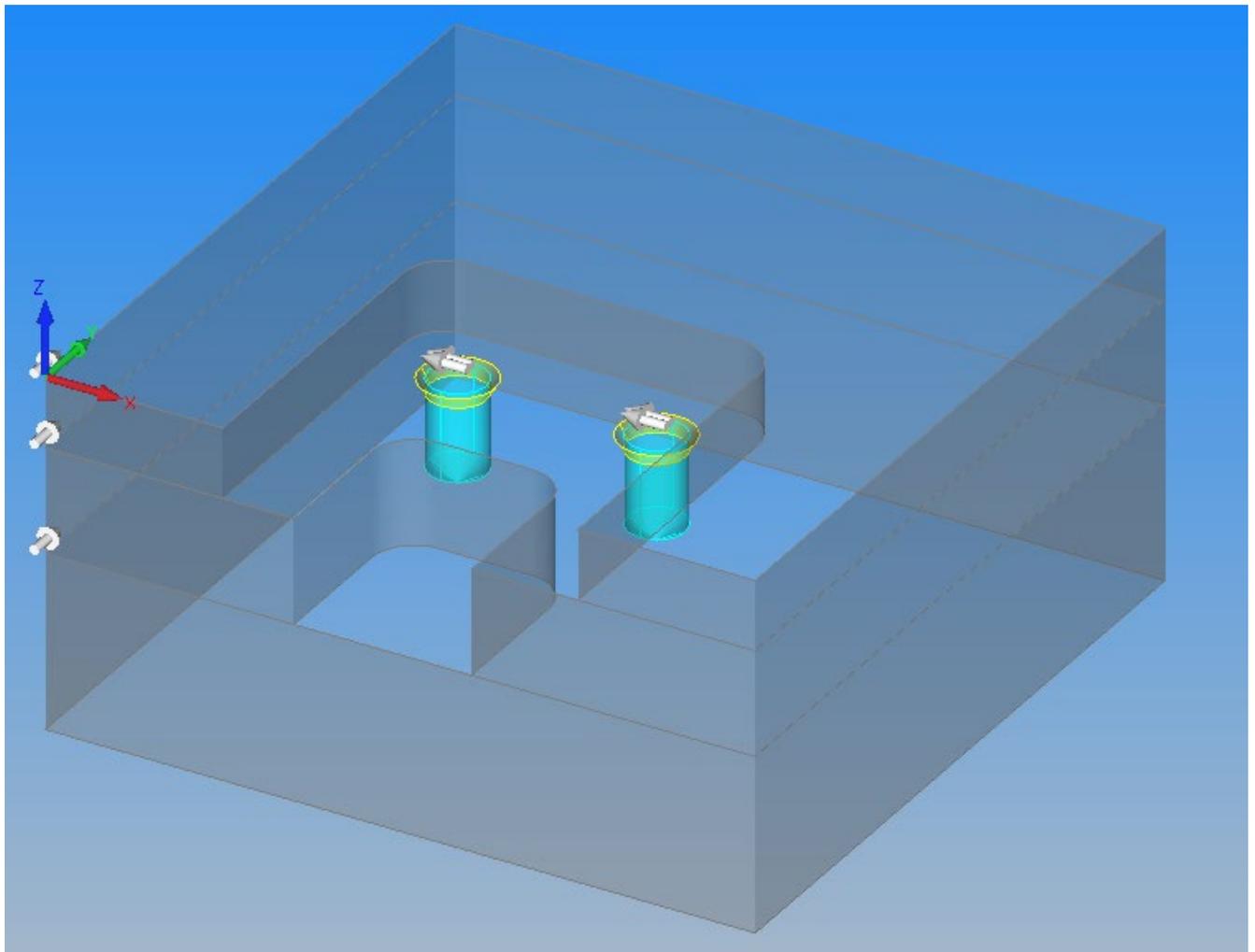


Figure 161 - Automatically extracted geometries without the Open Air method

You will see that the Automatic Extraction process has extracted the pocket geometry and a part of the outer profile.

Whilst this set of geometries could be machined, as in the previous example some of the faces will be machined more than once and this could lead to unacceptable finish or may damage the part depending on the type of tooling used or the material to be machined.

Manual Adjustment to Automatic Extracted items

Adding extra Geometries

To make this practical, machinable geometry we need manual intervention.

Draw a Geometry Line between points 1 and 2 as well as points 3 & 4 shown below.

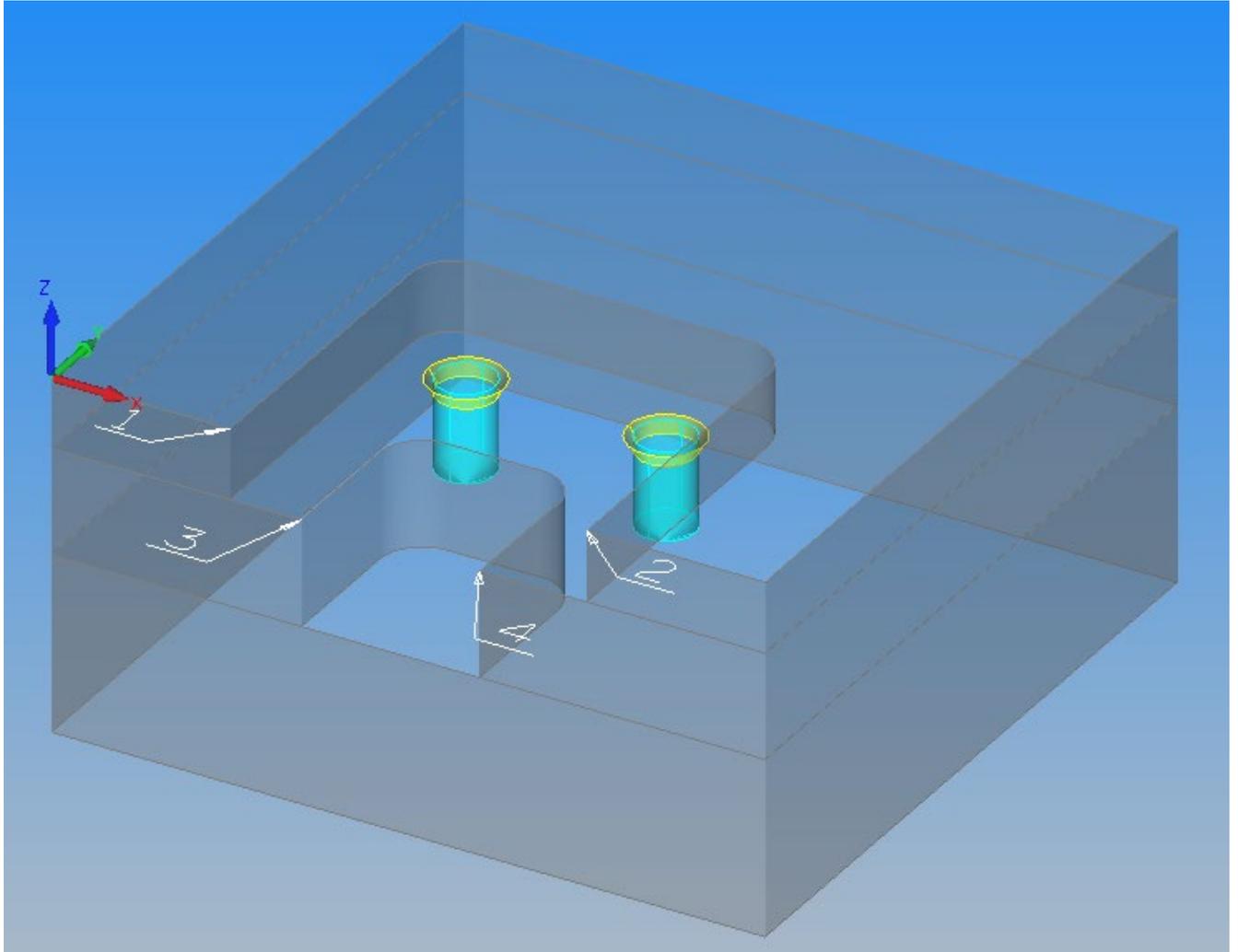


Figure 162 - Adding the first manual geometry line

It may prove easier to view the drawing in the **X Y view**  to make selecting the numbered points easier.

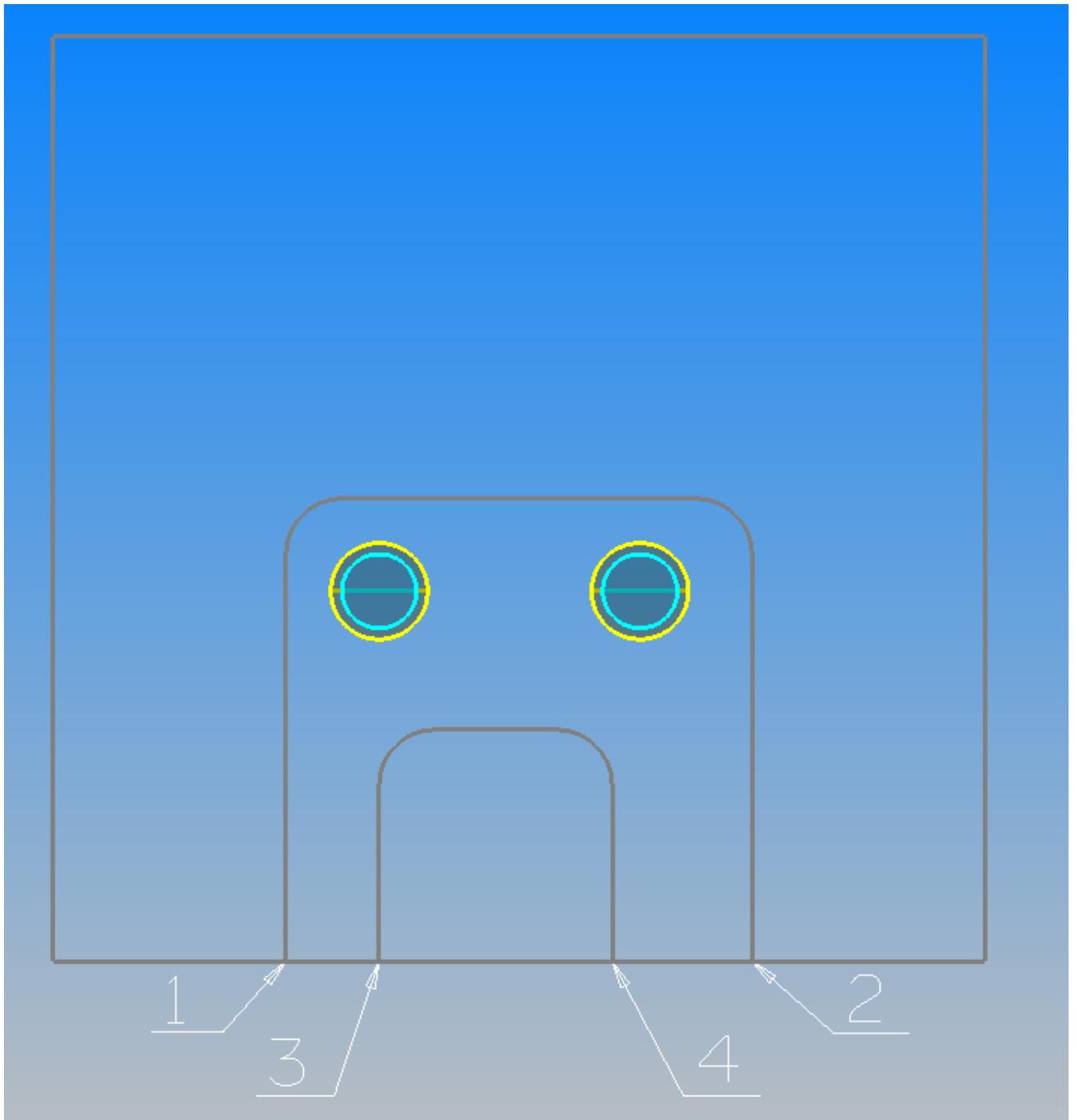


Figure 163 - View in XY to make selection easier

Once you have created the lines you will notice that they are not between the marked numbers on the images.

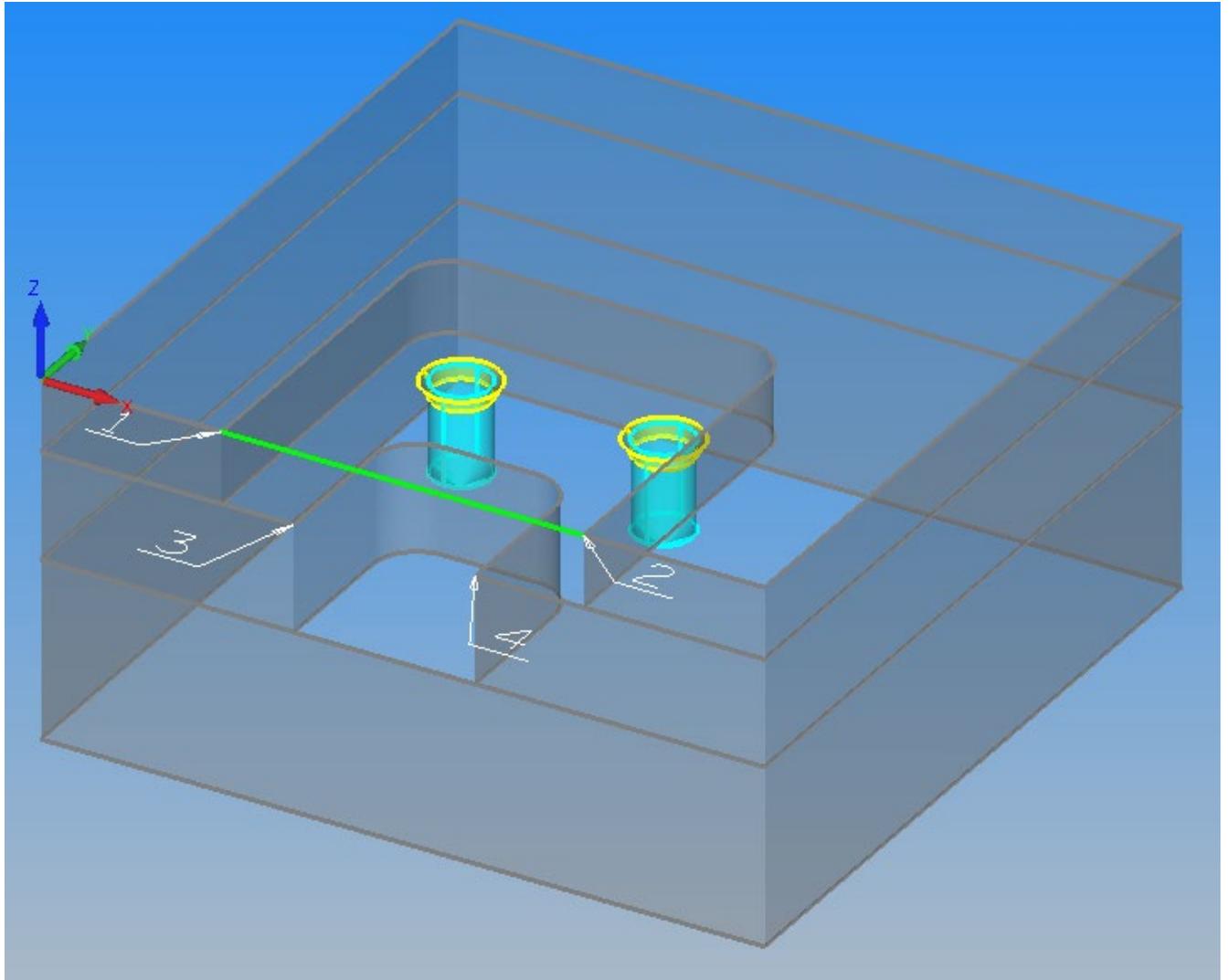


Figure 164 - Manually added geometries

This is because there is only a single Work Plane and the manually added geometries have been added accordingly.

All other geometries have Z Top and Z Bottom values and so appear to be in 3D.

Trimming the existing extracted geometries

We now need to adjust the form of the geometries to suit a better method of manufacture.

Using **EDIT > Trim**, , **<LClick>** the longer of the two lines, the one between points 1 & 2, as the cutting geometry.

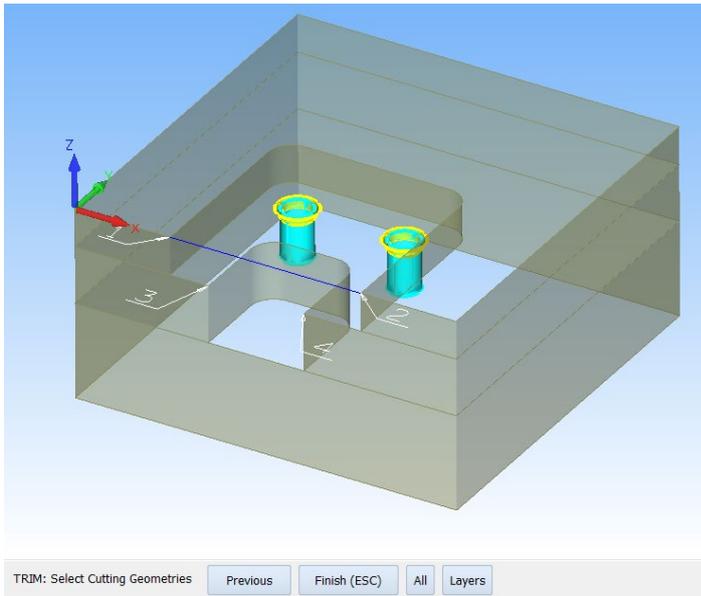


Figure 165 - Select the longer line

<RClick> to continue.

Now from the back of the drawing, **<LClick>** on the upper most Feature Extracted geometry as the item to trim.

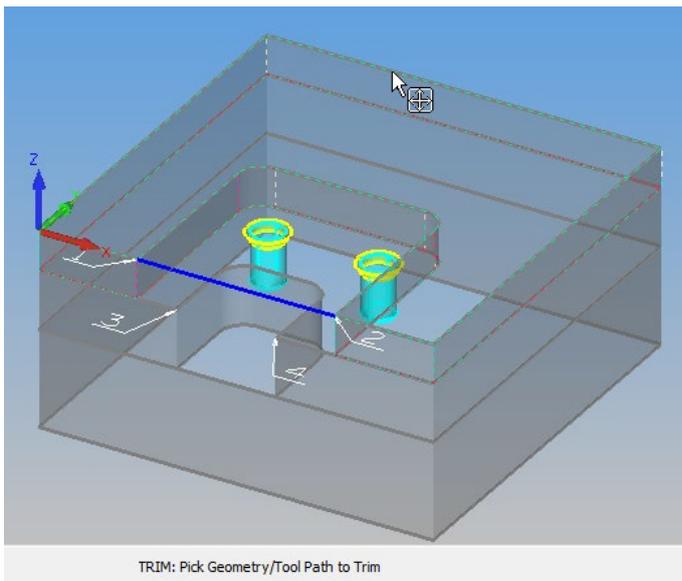


Figure 166 - Selecting the item to trim

<RClick> to finish.

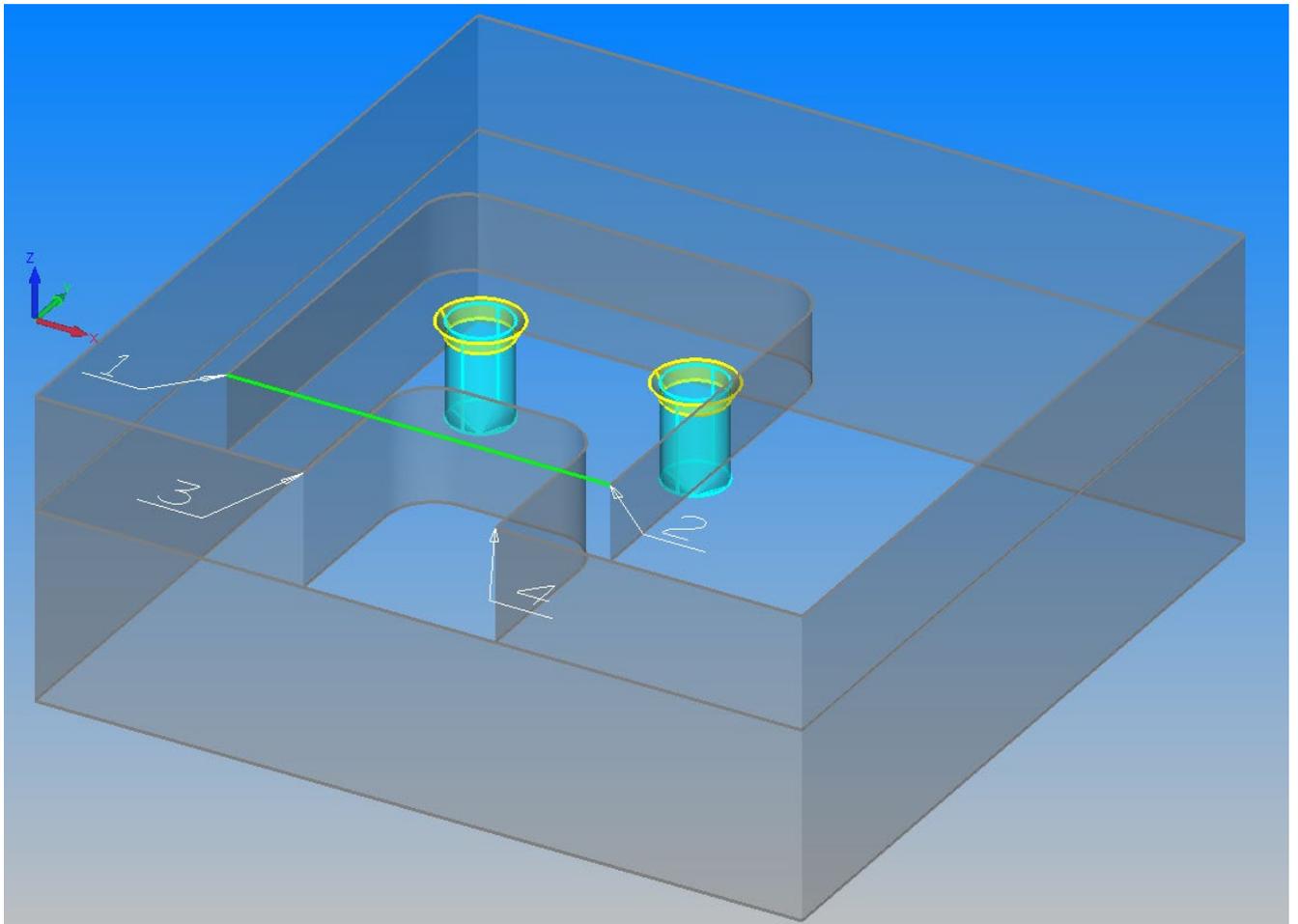


Figure 167 - Trimmed upper section



When trimming in this manner, it is advisable to select the upper Z geometry line of the extracted profile because the lower level is two lines.
The Bottom Z Level of the upper slice and the Top Z Level of the middle slice.



You will trim off the wrong section when trying to use a common line section as the Top geometry of any slice will always take precedence.

Using **EDIT > Trim**, , **<LClick>** the shorter of the two lines, the one between points 3 & 4, as the cutting geometry.

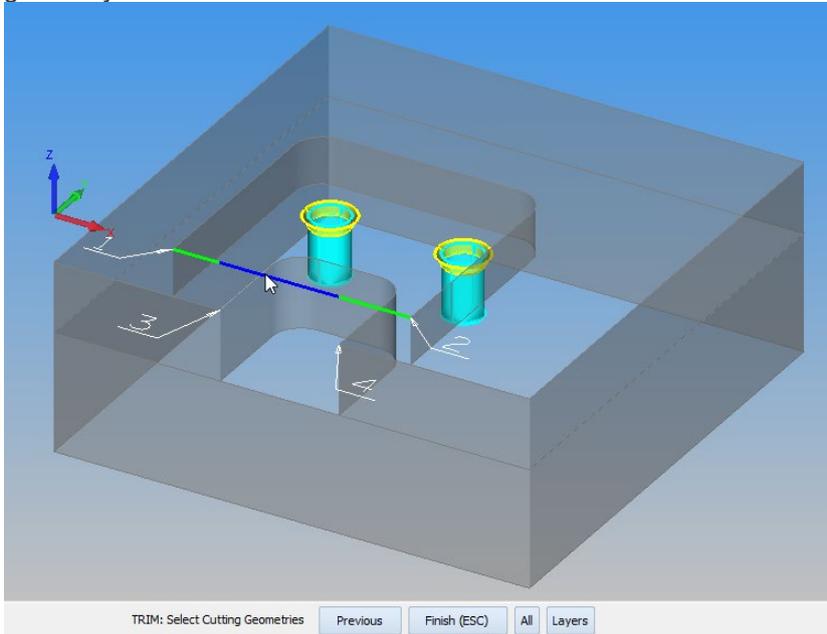


Figure 168 - Select the shorter line

<RClick> to continue.

Now from the back of the drawing, **<LClick>** on the upper most Feature Extracted geometry as the item to trim.

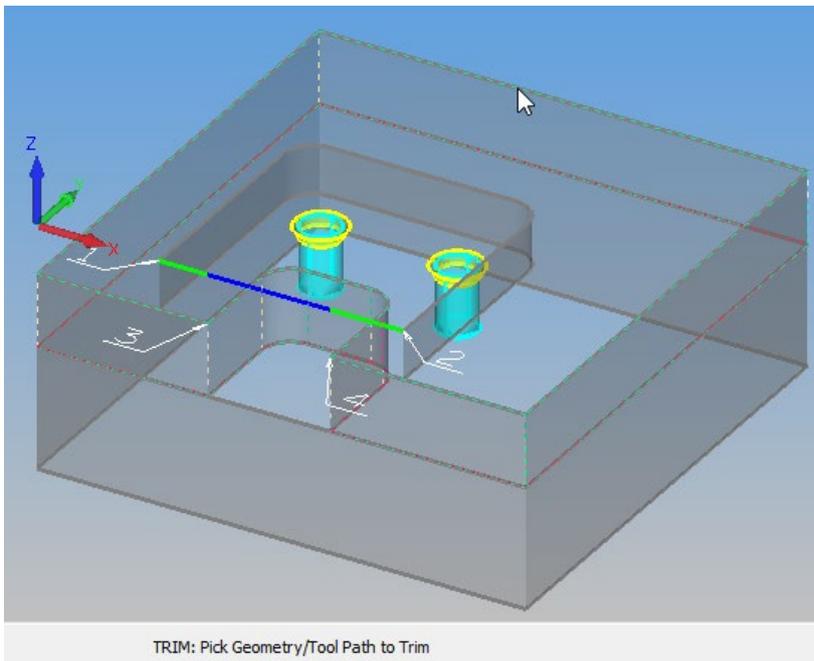


Figure 169 - Selecting the item to trim

<RClick> to finish.

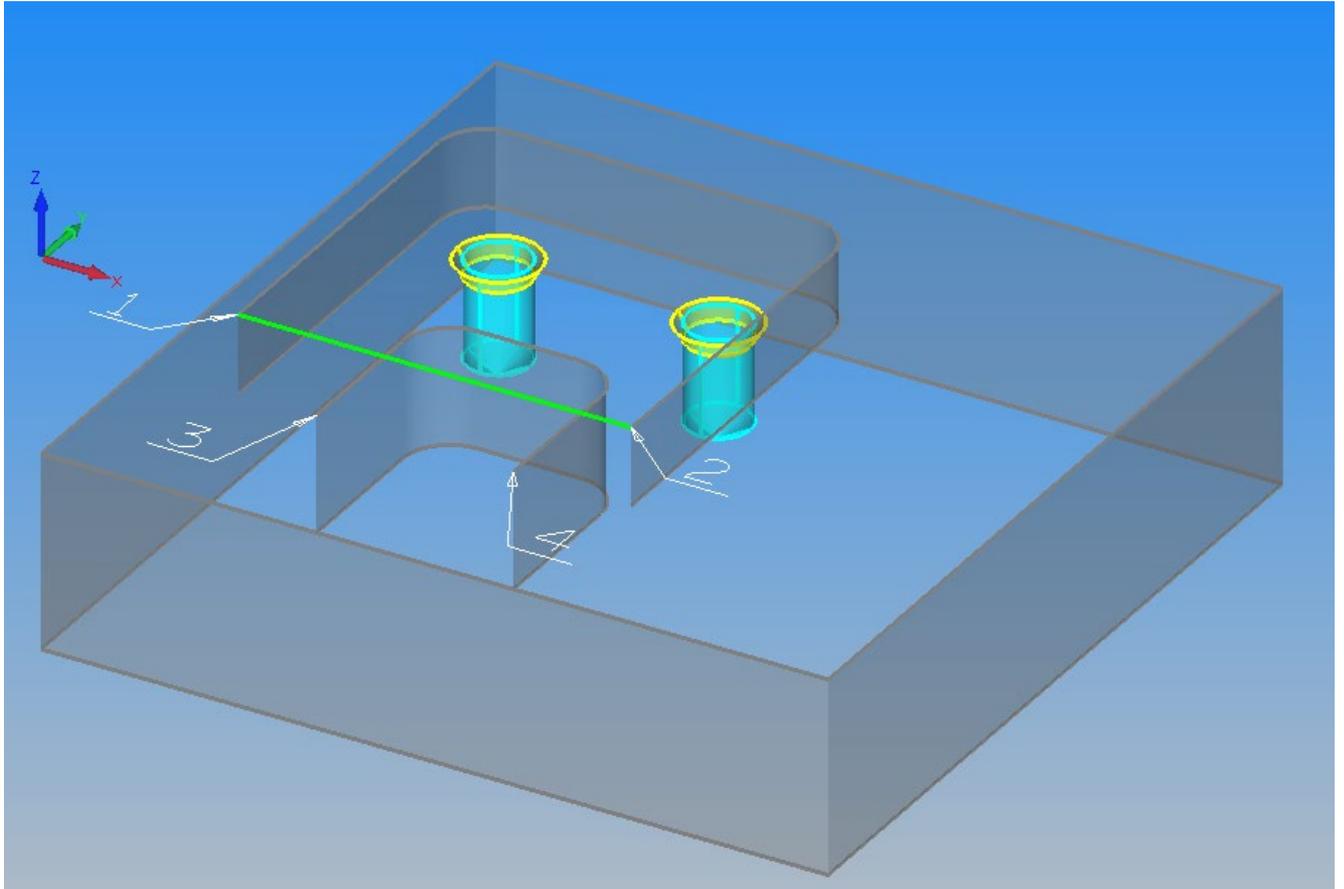


Figure 170 - Trimmed lower section

Creating the pockets

Using **EDIT > Join**  we will now create the geometries for two pockets.

The long line will be joined to the upper extracted geometry, the shorter line to the lower geometry. During this process it may prove beneficial to hide some of the working geometries as they may occupy the same locations as each other and prevent you from selecting the correct item.

Select **EDIT > Join** .

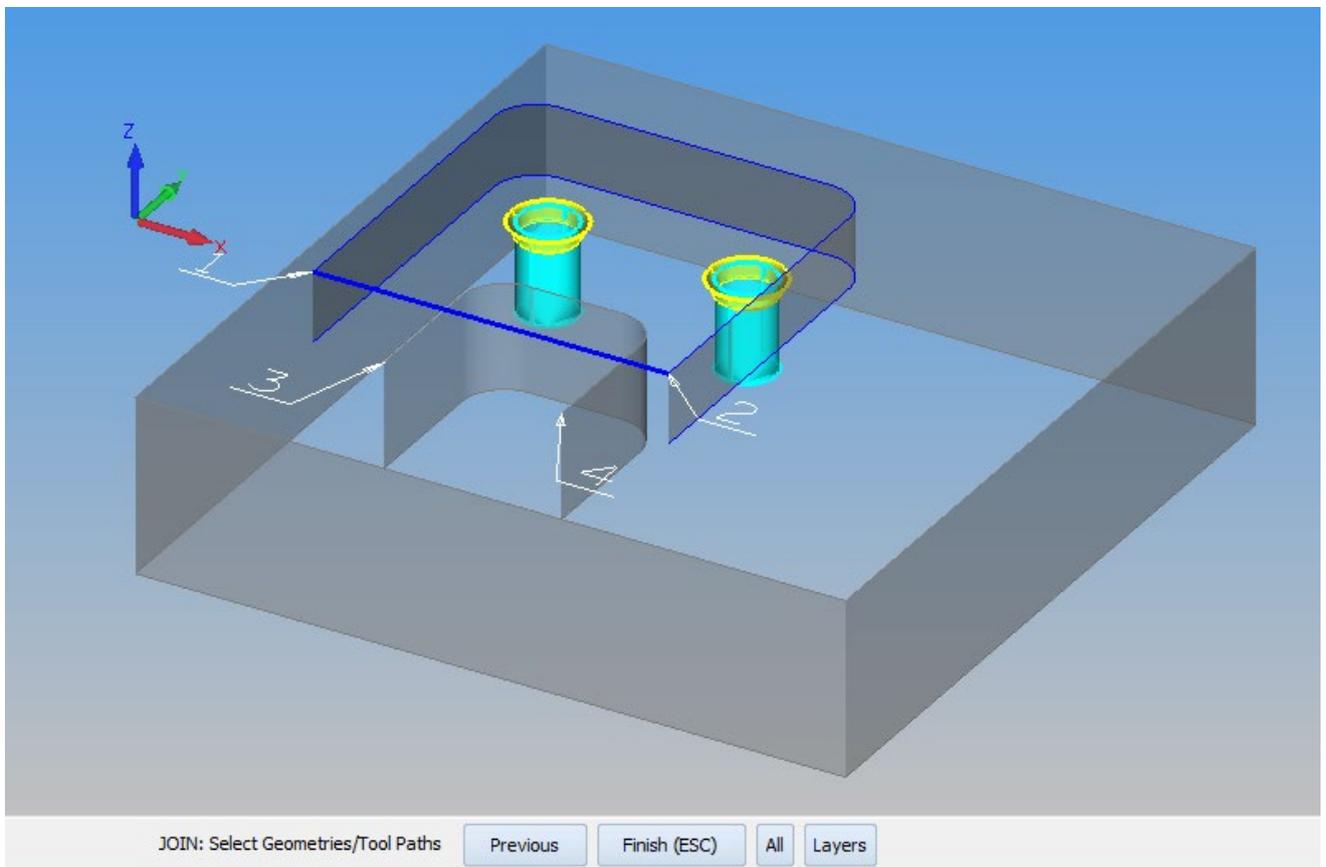


Figure 171 - Items to join for the upper pocket

<LClick> the items shown above, then **<RClick>** to finish.

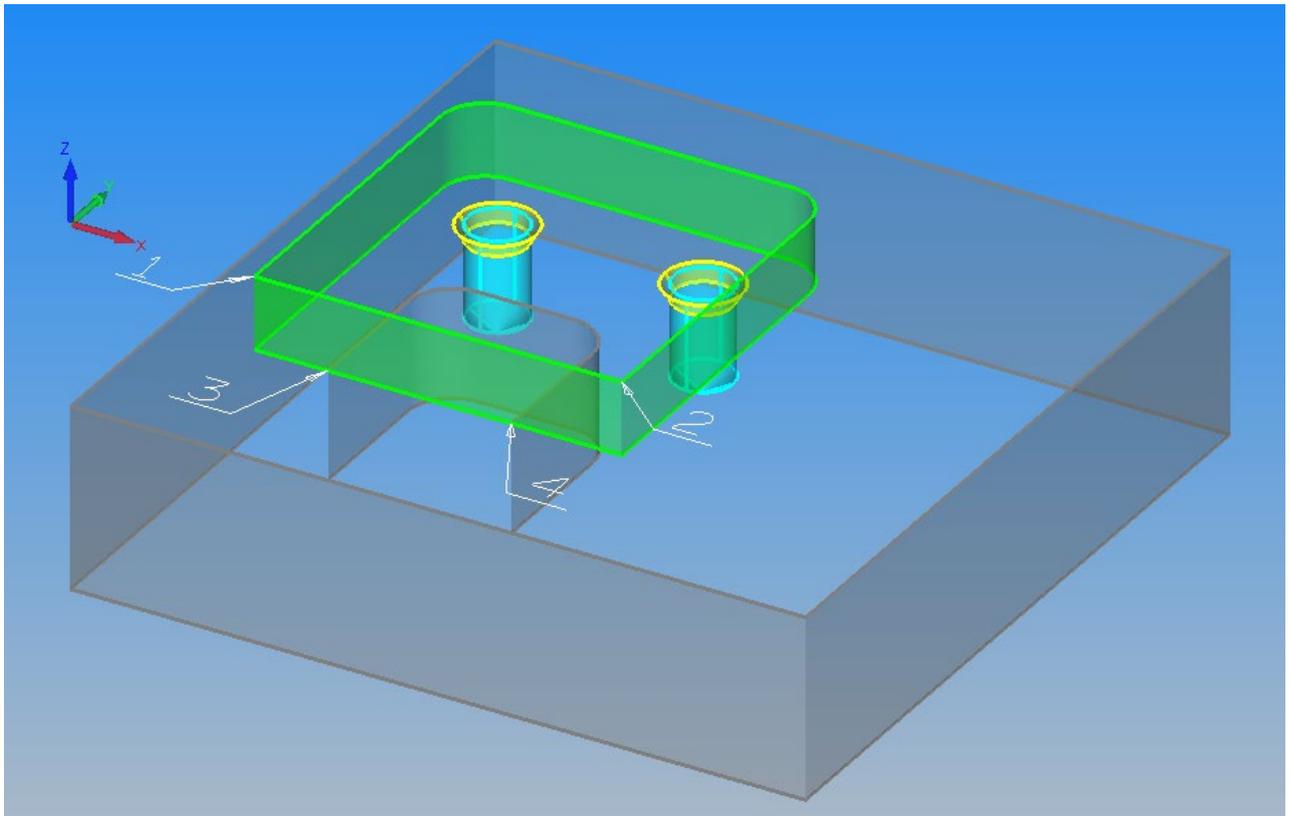


Figure 172 - Completed upper pocket

Notice that when you join manually created geometry to Feature Extracted geometries, the manual item gains the attributes of the extracted items.

Notice also, that the geometries are now place on the primary Geometry layer.
Manual geometry layer placement takes priority over Feature Extracted layer placement.

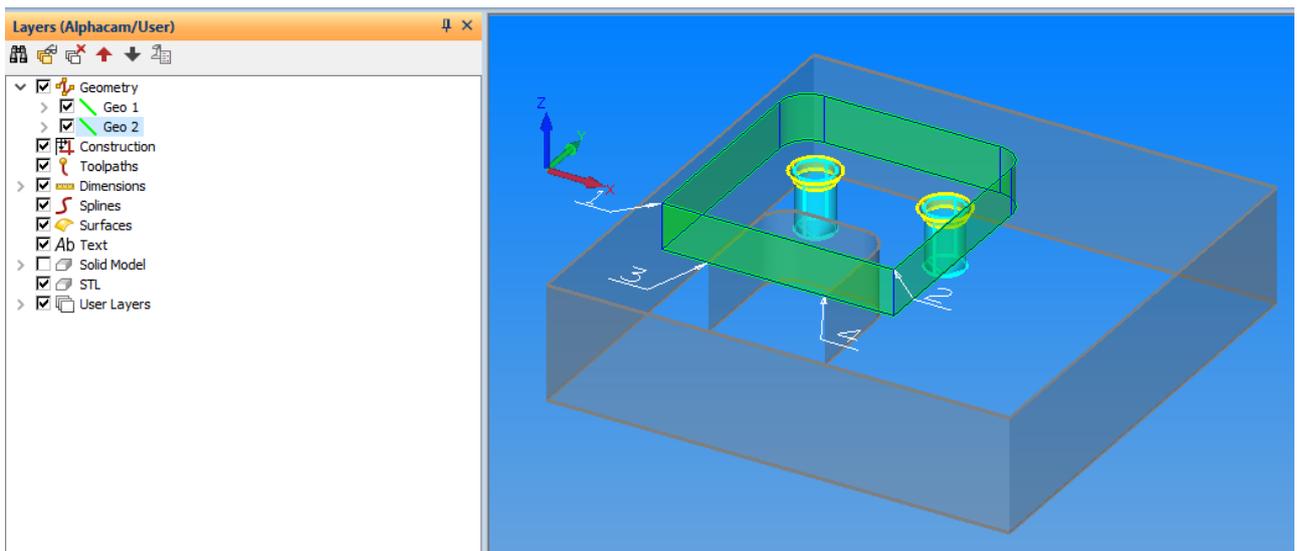


Figure 173 - Layer location of the new pocket

You will need to untick the new geometry to gain access to the smaller line we drew earlier.

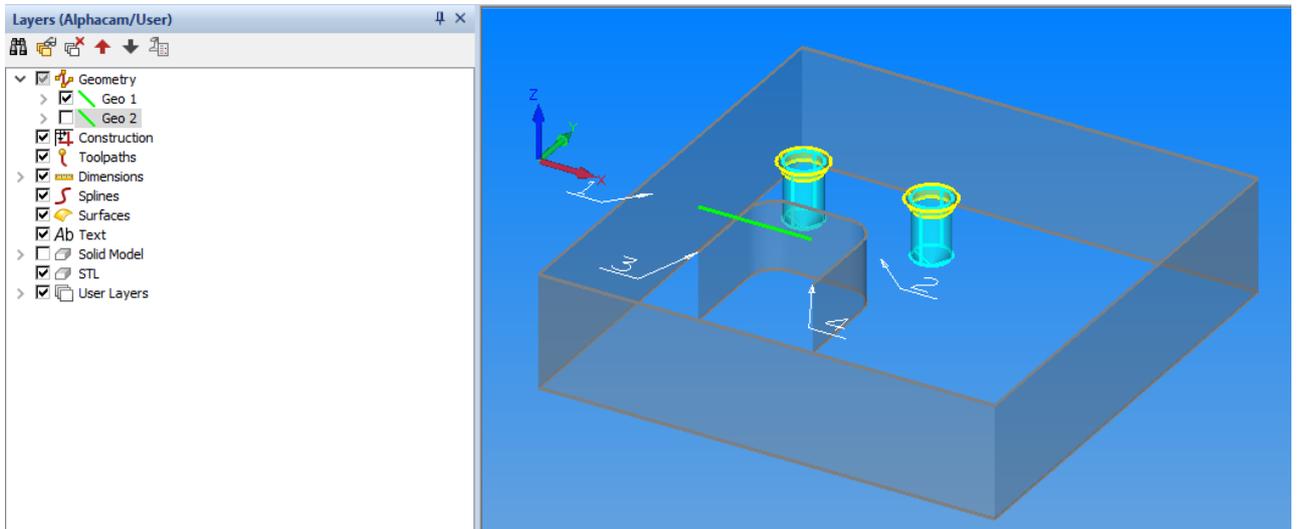


Figure 174 - Hide the new pocket

Select **EDIT > Join** 

Repeat the process with the small line to the smaller pocket detail.

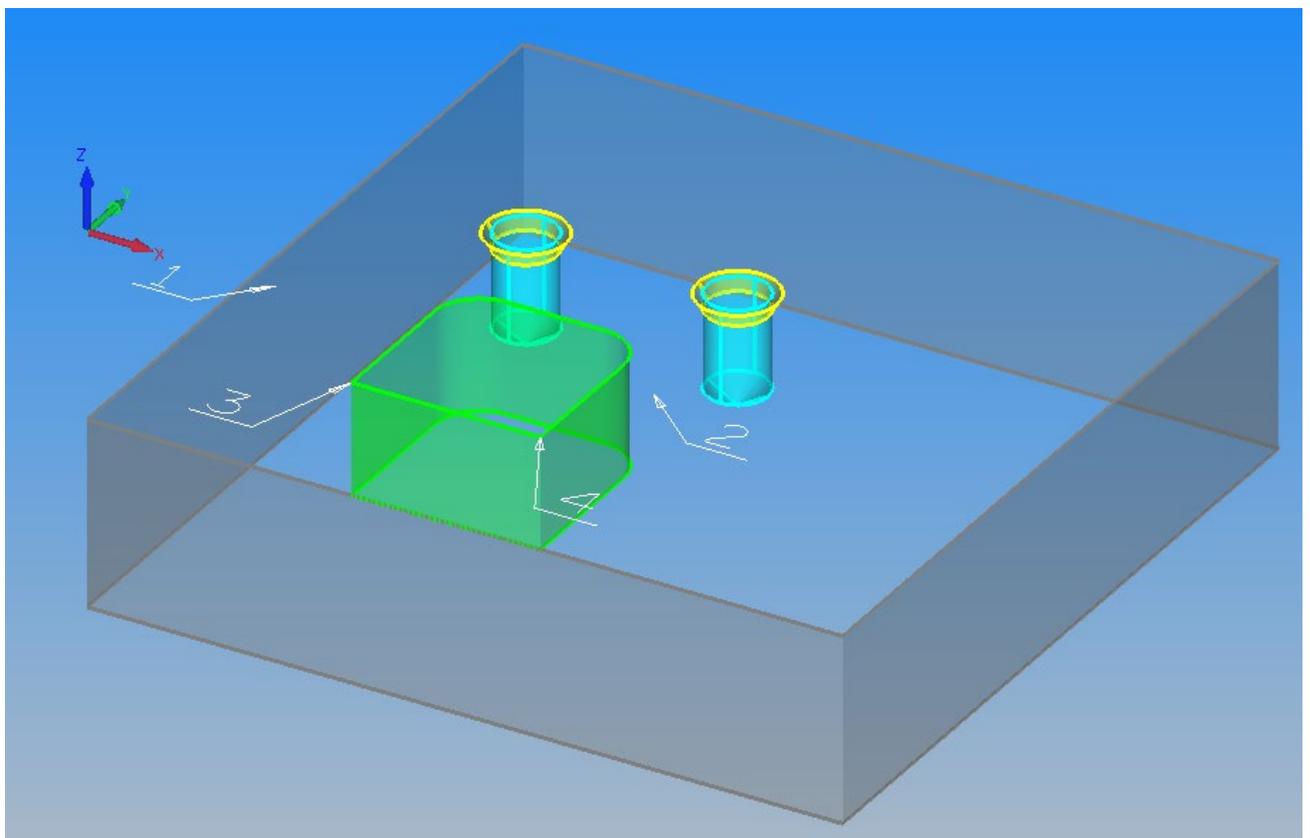


Figure 175 - Completed smaller pocket

Setting the pockets ready for manufacture

If we were to add a pocket cycle to these two new geometries, the pocketing would be contained inside the boundary and not flow out as described in a previous example with the open ended slot detail.

Turning on the ghost tools will also indicate that the start point for machining is not in a good place.

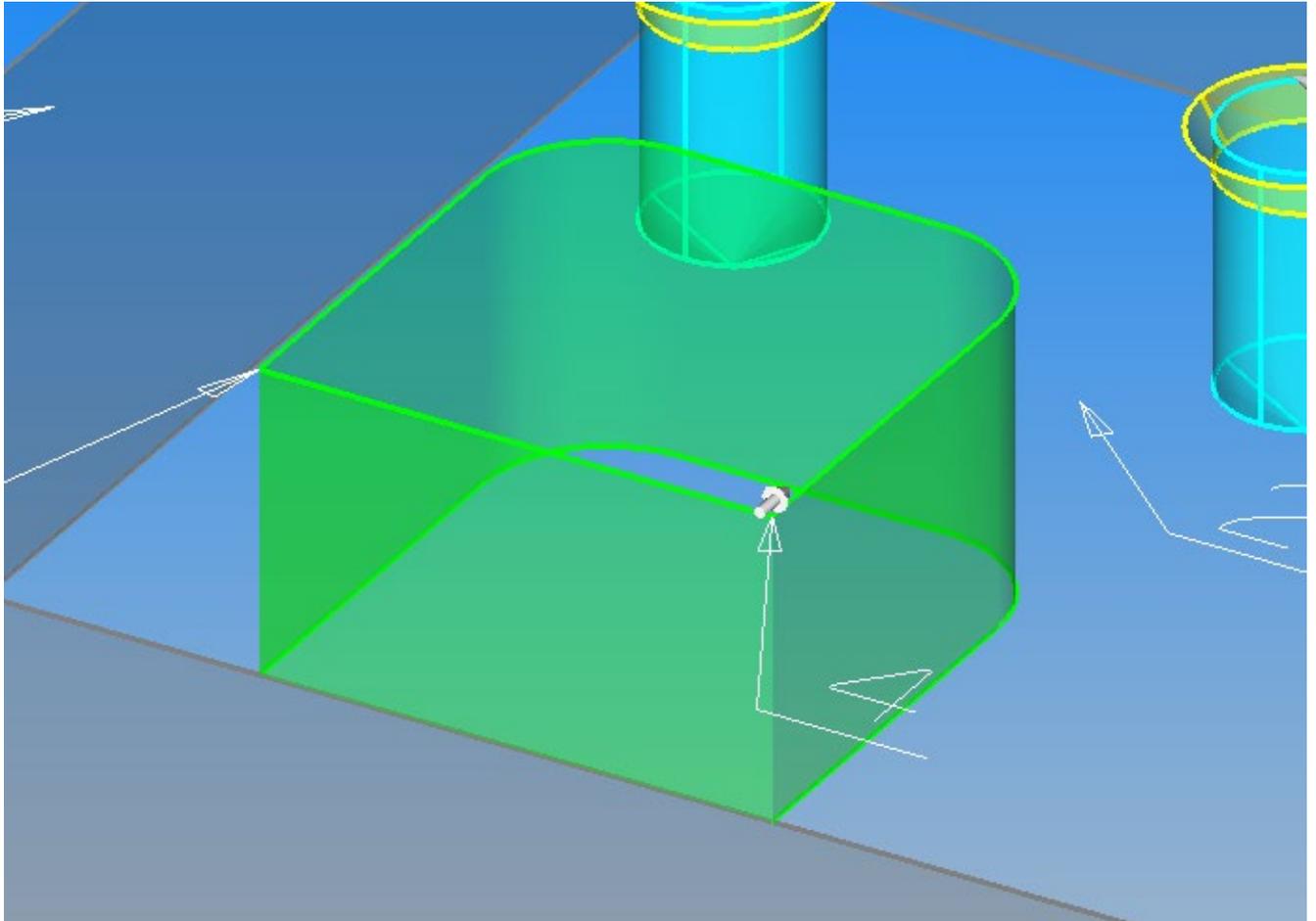


Figure 176 - Bad Start Point location

The **Start Point** needs to be in the middle of the open side of the pocket and that side also needs marking as an **Open Element**.

When setting the Open Elements of split areas, care was needed to choose the correct First and Last items to mark the correct section as open.

Marking a single element as open is much simpler, **<LClick>** on it twice.

If we open the side first, then alter the Start Point, it will make the job much simpler.



Use the **MACHINE > Set/Unset Open Elements** command.
<LClick> the side between the numbered markers, once to say it is the **First** and again to say it is the **Last**.
<RClick> to finish.

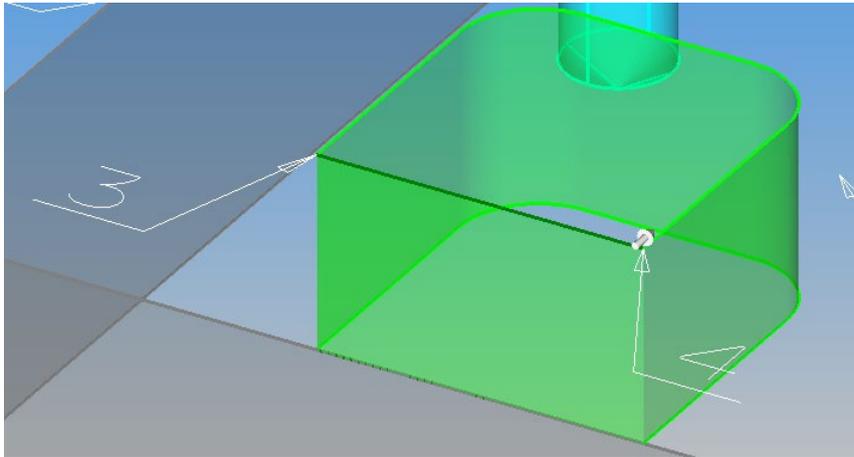


Figure 177 - Front side set Open



Use **EDIT > Start Point** command.
 Set the start location to the mid-Point **<F7>** of the open element.

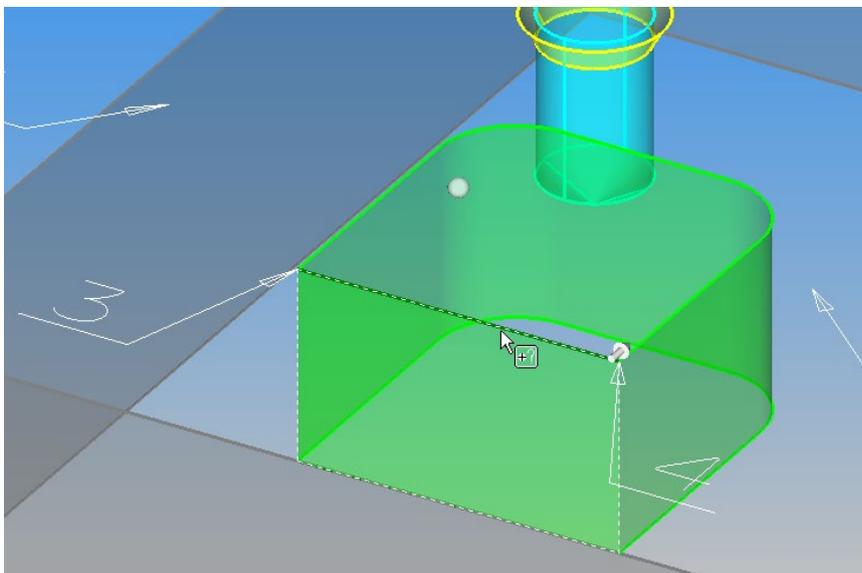


Figure 178 - Resetting the Start Point



Notice that the Snap bubble will appear above the line because the line actually exists on **Flat Land**.

<RClick> to finish.

We will repeat the process on the upper pocket.

Hide the lower pocket and make the upper one visible.

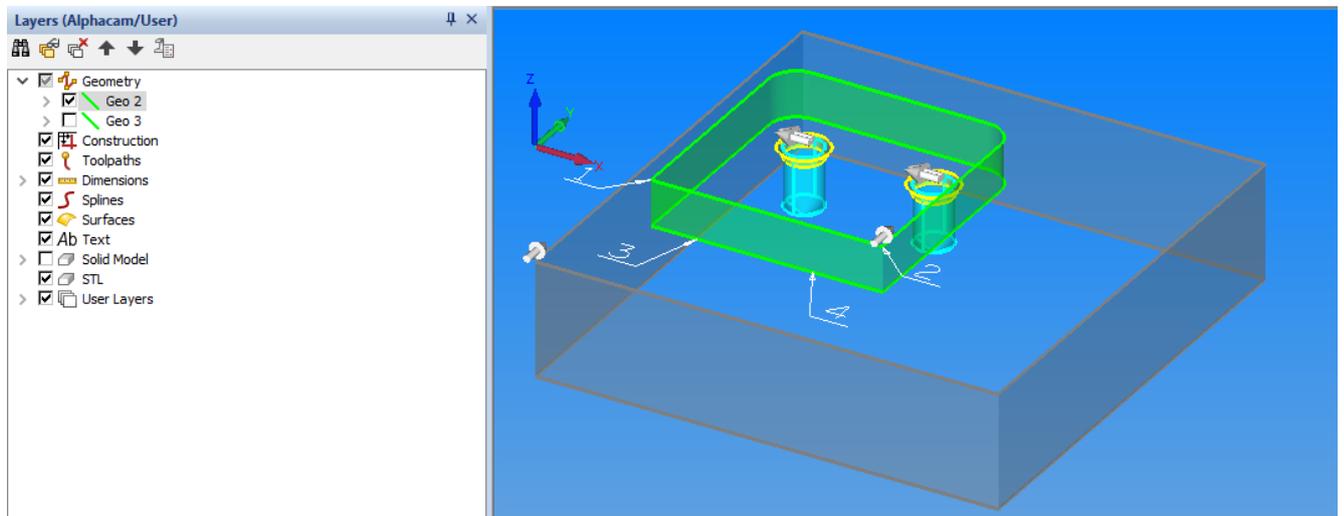


Figure 179 - Swap pocket visibility



Use the **MACHINE > Set/Unset Open Elements** command.
<LClick> the side between the numbered markers, once to say it is the **First** and again to say it is the **Last**.
<RClick> to finish.

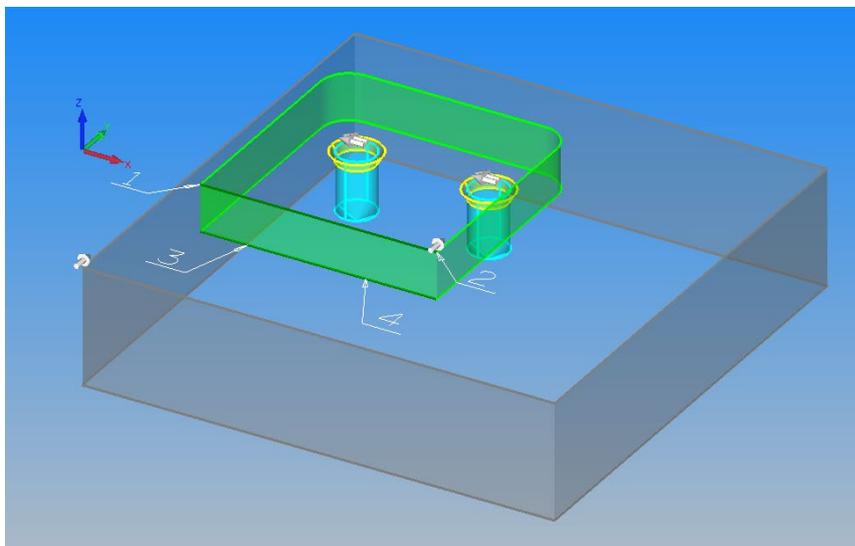


Figure 180 - Front side set Open

Use **EDIT > Start Point** 

Set the start location to the mid-Point **<F7>** of the open element.

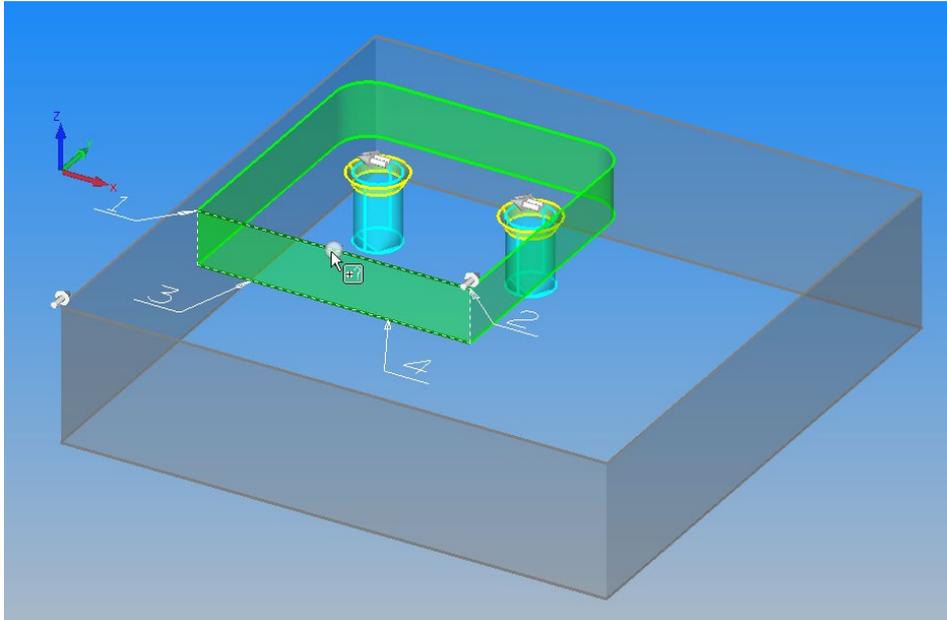


Figure 181 - Resetting the Start Point

<RClick> to finish.

Turn both pockets visible once more.

Make the Solid Model visible too.

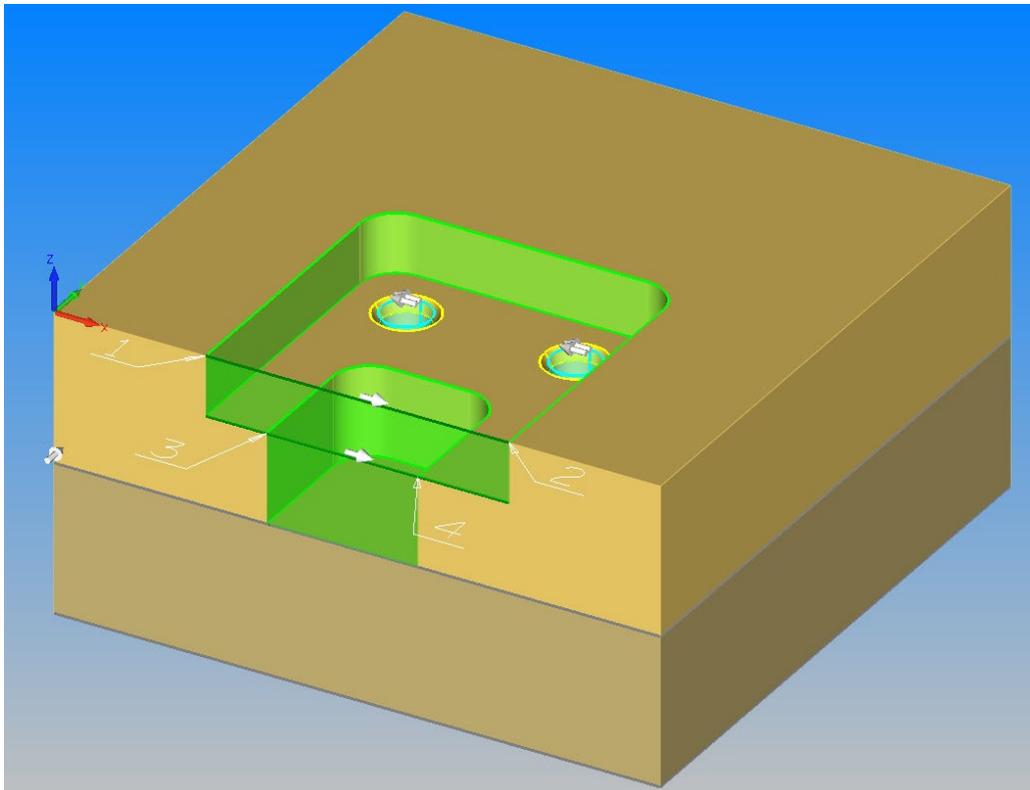


Figure 182 - Correct pocket set up

Reset the Outer Profile

The last part of this process is to make the Outer Profile the same height as the actual model.

Use **SOLID MODEL UTILITIES > Reset Geometry Z Levels**
When prompted, **<LClick>** the outer profile geometry.

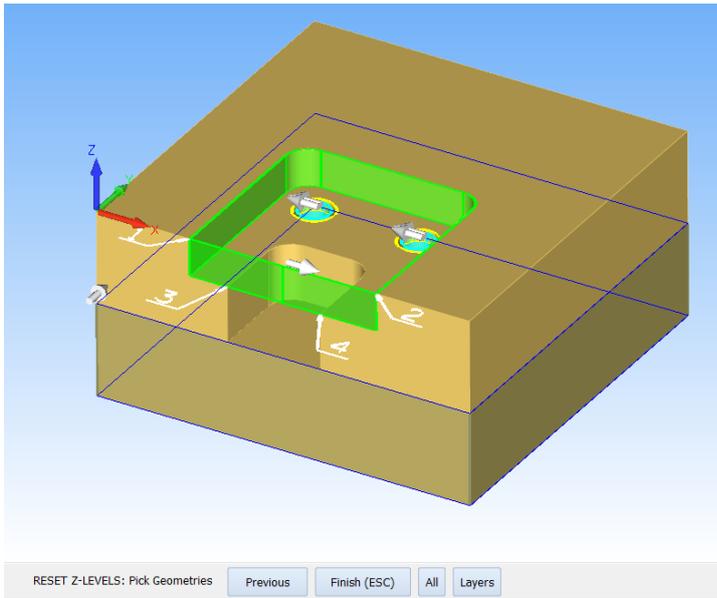


Figure 183 - Selecting the outer profile

<RClick> to continue.

As the bottom of the geometry is where it needs to be located.

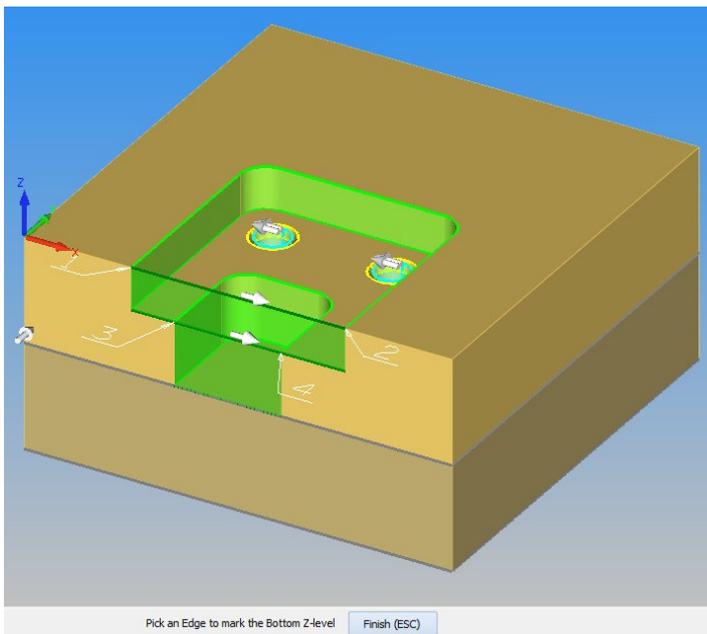


Figure 184 - **<RClick>** to accept the current correct location

Using **<RClick>** to accept its location.

<LClick> on any edge on the top face of the model for the new Top location of the geometry.

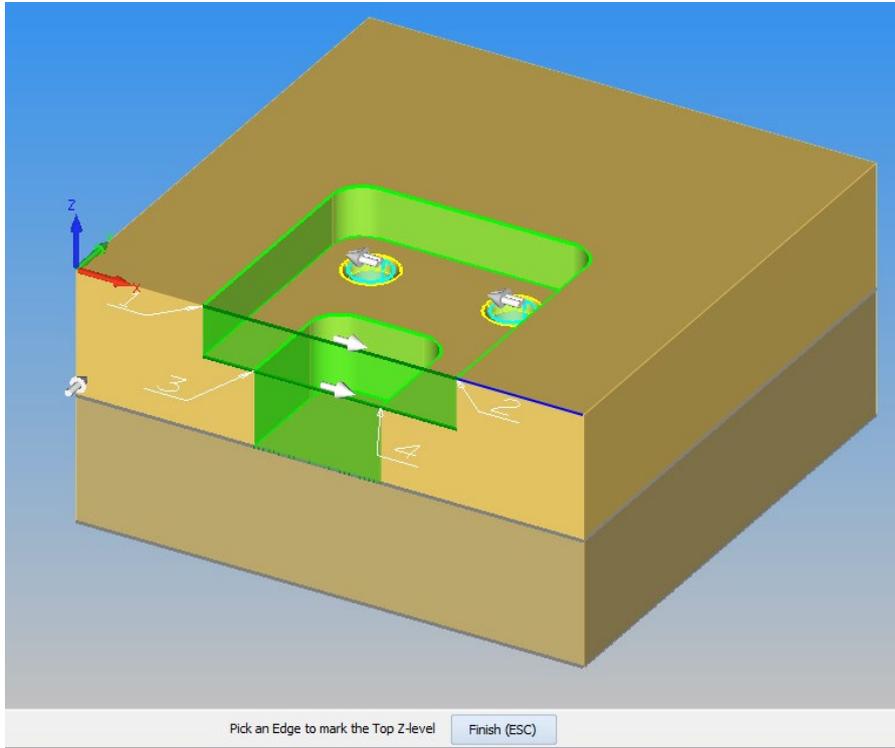


Figure 185 - Any edge on the top face is a correct selection

<RClick> to finish.

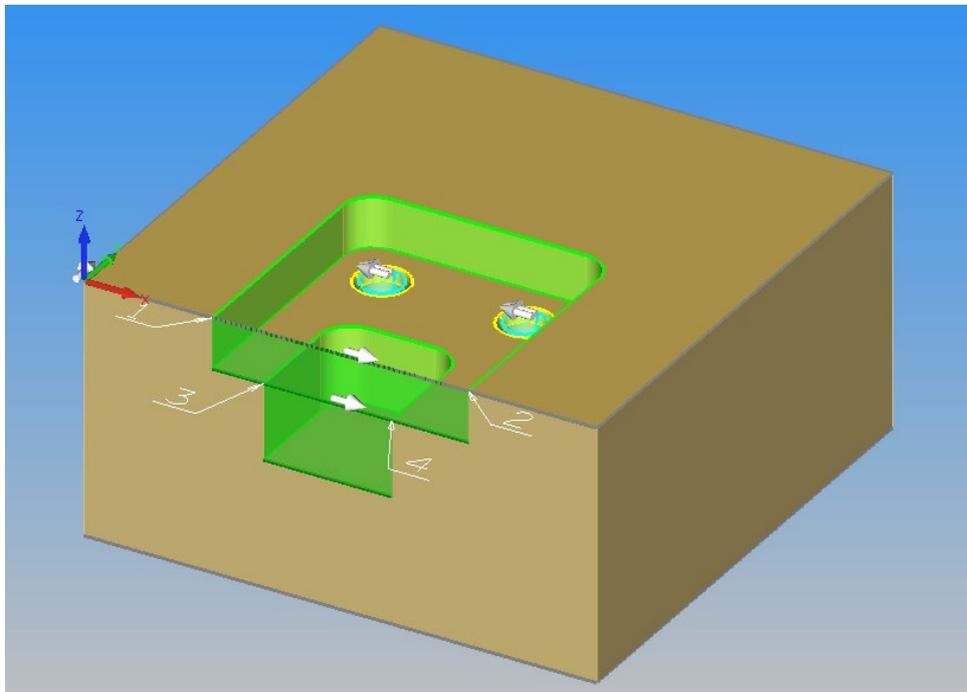


Figure 186 - Completed part

Save your drawing with a suitable name.

Using Open Air Pocket Method

Instead of drawing out the parts by hand, we can also use the **Use Open Air Pocket Method** on the

SOLID MODEL EXTRACT > Automatic Extraction



Contours tab.

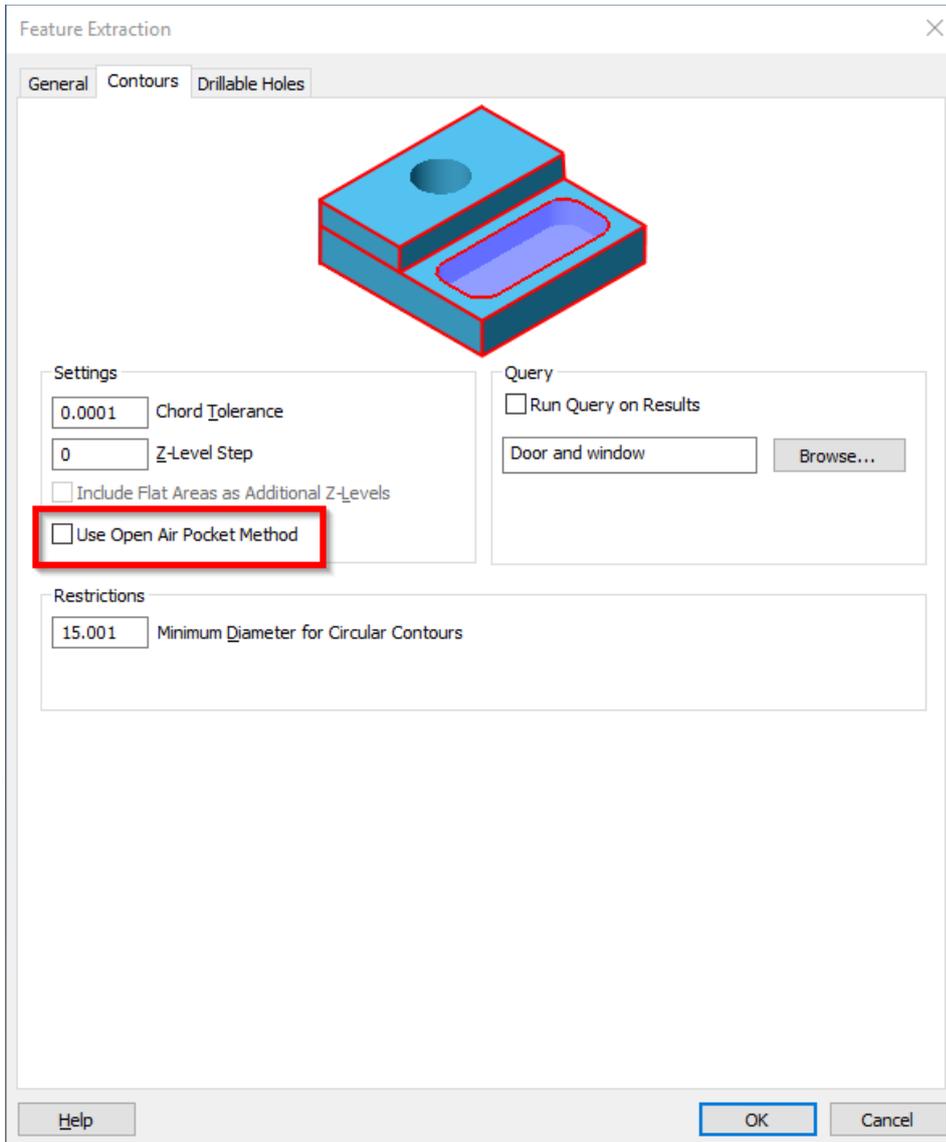


Figure 187 - Use Open Air Pocket Method option

Reset the part

Use **HOME > Clear Memory** 

Make the options as shown to remove the previous work.

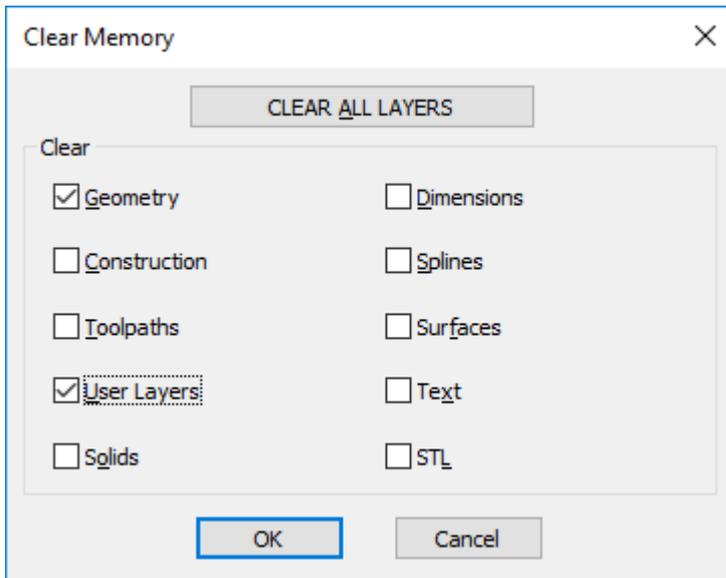


Figure 188 - Clear Geometries and User Layers

<LClick> [OK] to continue.

Use **SOLID MODEL EXTRACT > Automatic Extraction** 

All settings are remembered from the last use of the command, just go to the Contours tab and make the **Use Open Air Pocket Method** active.

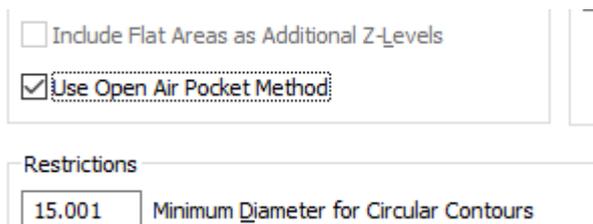


Figure 189 - Use Open Air Pocket Method set to active

<LClick> [OK] to continue.

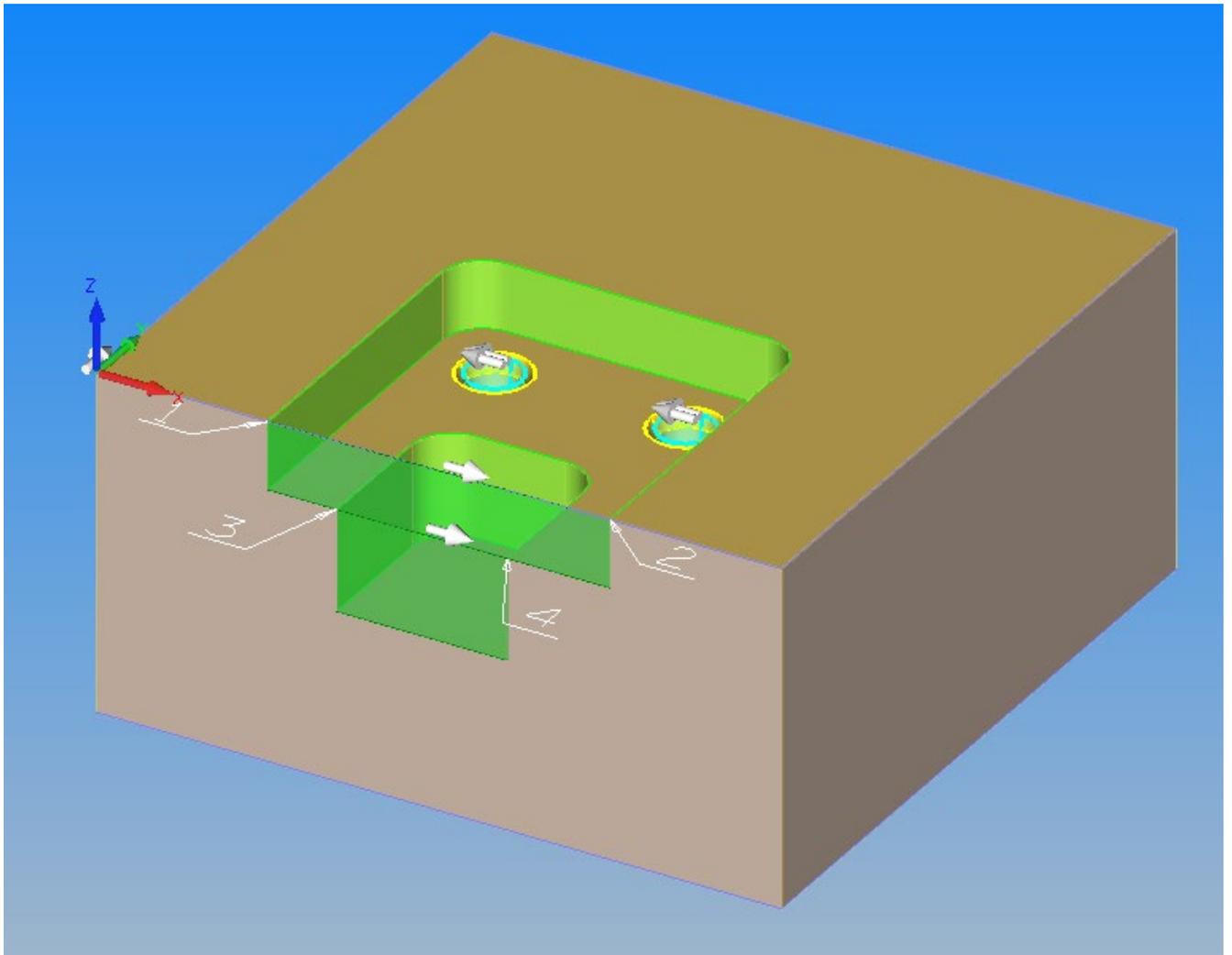


Figure 190 - Completed part

Important Note

The reason for the first section of this Open Pocket Block tutorial was to instruct you for when parts are not extracted as you would expect.

Use of the Open Air Pocket Method has restrictions, the most important one is that the Open Elements **MUST** exit the actual model.

Pocket sections similar to this sample that would be open sided but still within the confines of the absolute outer profile of the model are not considered for the Open Air Pocket Method.

#8 Good Trim Part

Trimming Part

Select **HOME > Input CAD**  and the following dialogue is displayed.

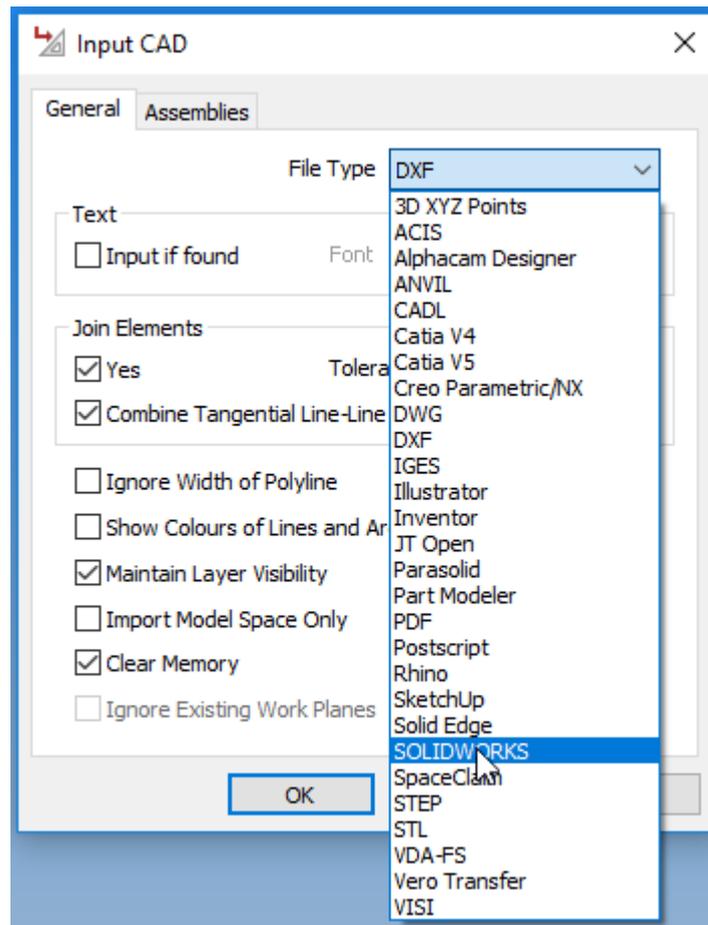


Figure 191 - Input CAD set to Solidworks

Set the file type to **SolidWorks**, and then select **[OK]**

The file selection dialogue is displayed.

Navigate to “...ALP TRG 114 Feature Extraction 2020\Examples\SolidWorks Parts\” and select the “#8 good trim part.SLDPR” then select **[Open]**.

Importing SolidWork Configurations

With SolidWorks files we have the ability to import any of the Configurations held within the model.

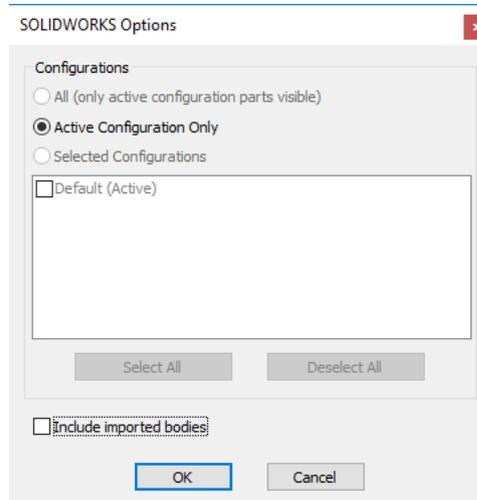


Figure 192 - SolidWorks Configurations options

At this point you would select the correct one if there are more than the single Default option we see in the image above.

<LClick> to continue.

The model is imported and displayed on screen.

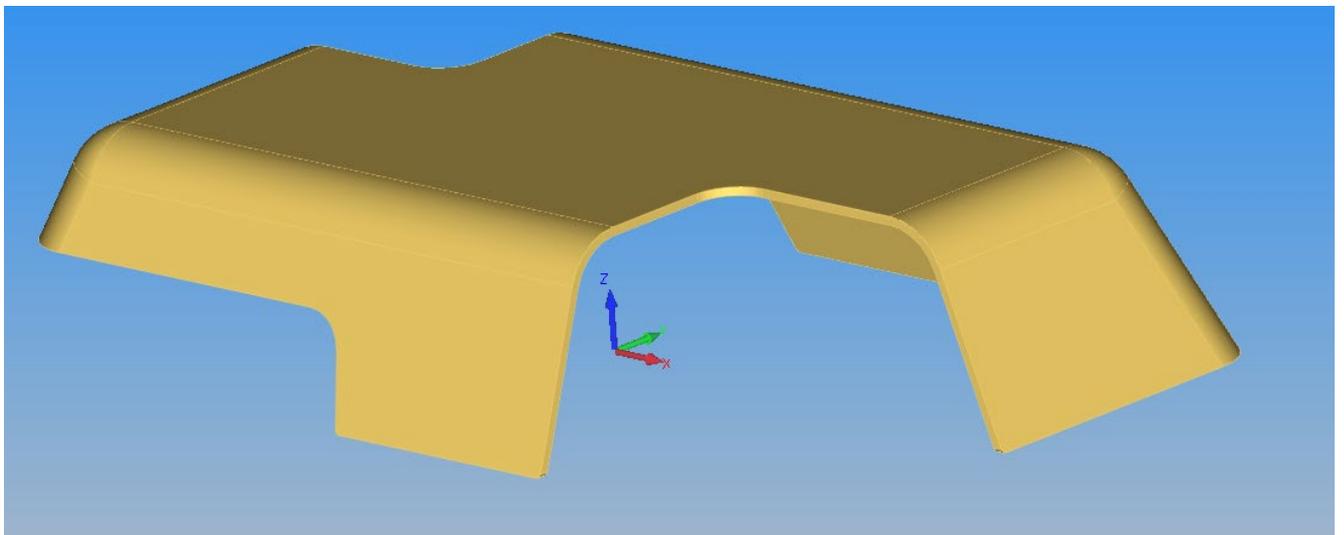


Figure 193 - Good Trim example part

The model is in the correct orientation however the datum position would be better at the lower left corner of the component.

This can be done in several ways.

Position Model Using 3D Rotate

Select **SOLID MODEL UTILITIES > Auto Align Part** 

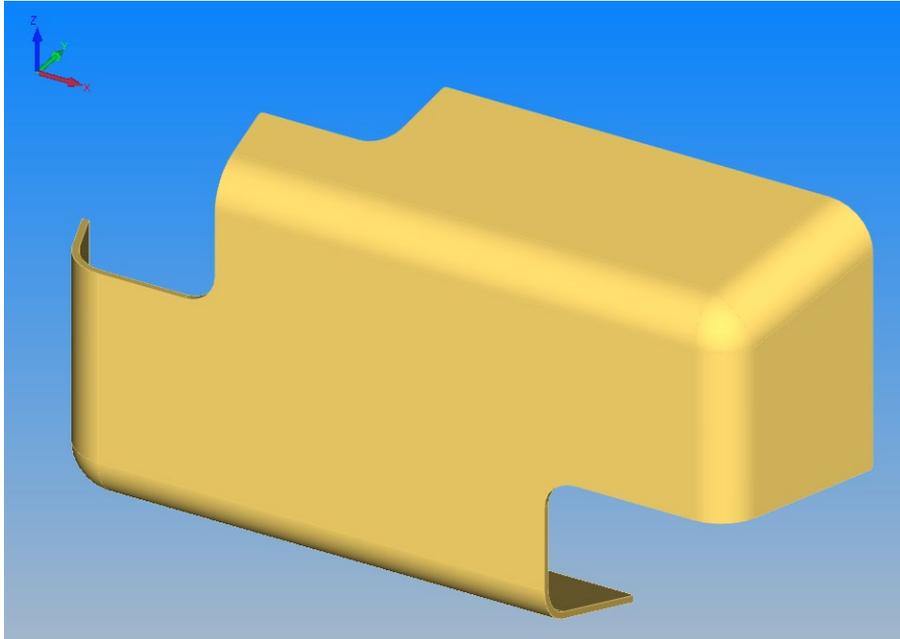


Figure 194 - Auto-Align once

then Select Auto Align again.

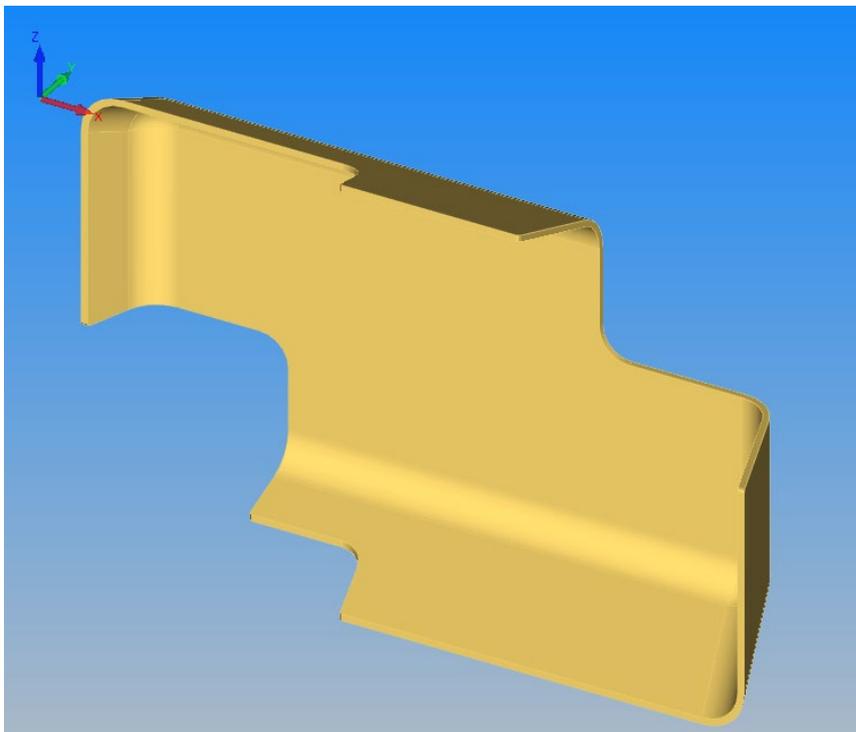


Figure 195 - Auto-Align twice

3D Part Rotation options

Use **3D > 3D Part Rotation** 

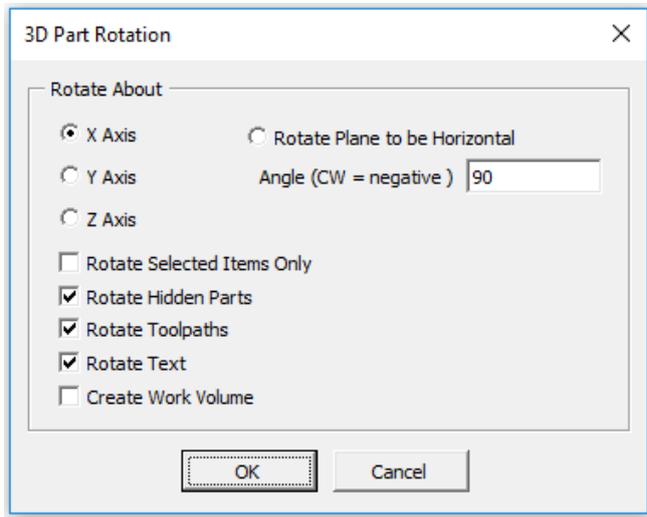


Figure 196 - 3D Part Rotation dialogue

This command allows for a more flexible manner of rotating parts than the **EDIT > Rotate** command will.

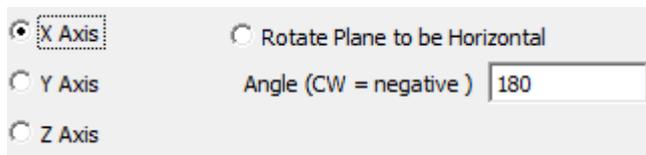


Figure 197 - Rotation Axis options

Using 3D Part Rotation allows for the use of the primary X, Y & Z global axis using one of the three options.

⊙ Rotate Plane to be Horizontal will prompt you to select three points on a solid model which will then be aligned flat to the global X & Y axis. This method allows you to rotate items when you do not have a specific angle to work with.

Angle (CW = negative) allows for the direct input of a dedicated rotation angle.

Rotate Selected Items Only

Allows you to choose which items to rotate

Rotate Hidden Parts

If you have items hidden, User Layers for example, with this option you do not need to make them visible to perform the rotation on them.

Rotate Toolpaths

Allows for the rotation of any applied tool paths.

Rotate Text

Allows for the rotation for any text.

Create Work Volume

Creates a Work Volume around the part once the rotation command is actioned.

Make the options as follows.

Rotate about the X axis with an angle value of 90°. <LClick> [OK].

Set the rotation point to X0, Y0, Z0, then <LClick> [OK].

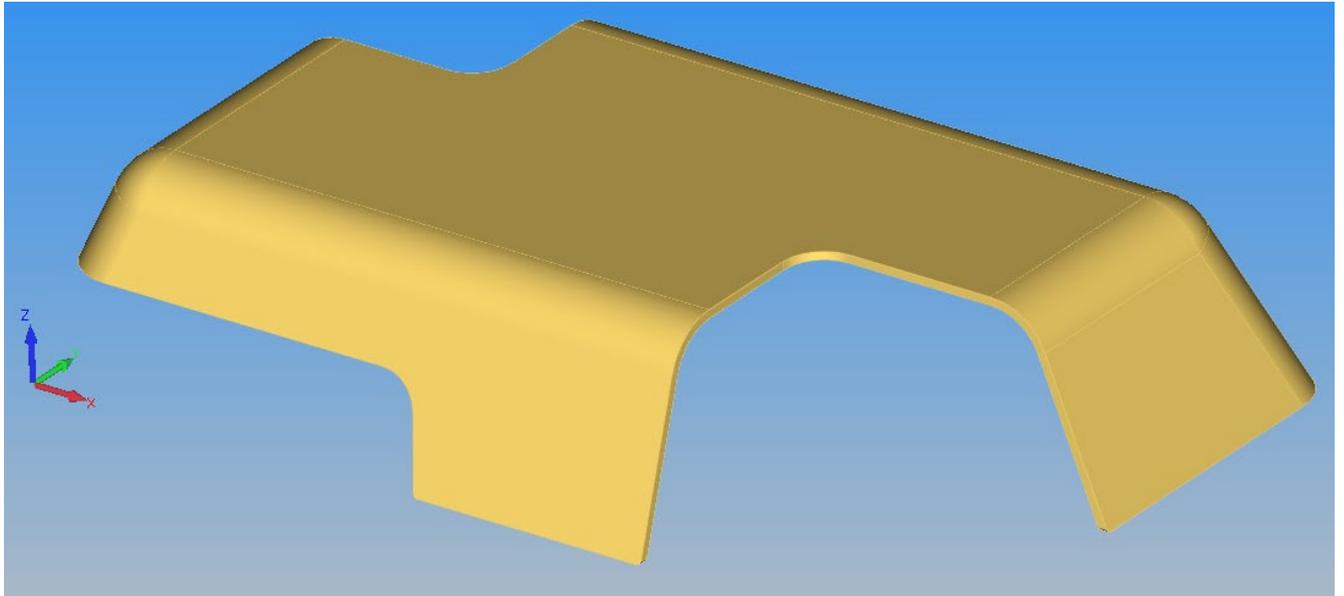


Figure 198 - Part in correct attitude

Position Model Using added geometry and Move command

Using **Geometry > Line** , create a line along the front lower edge of the model as shown below.

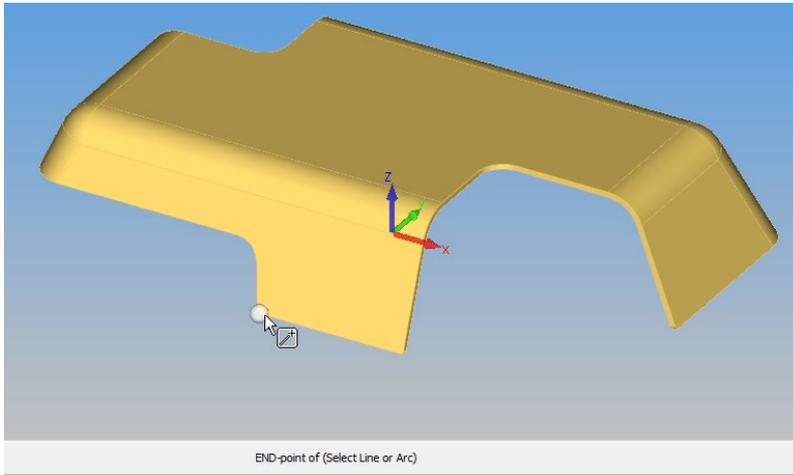


Figure 199 - Front line start point

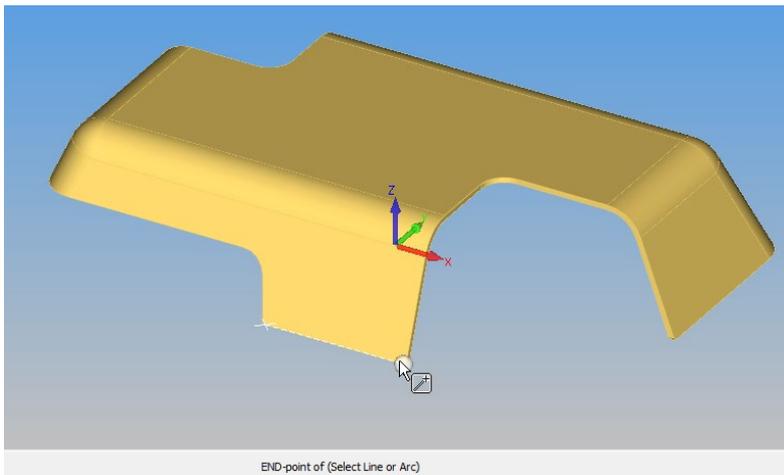


Figure 200 - Front line end point

<RClick> to finish.

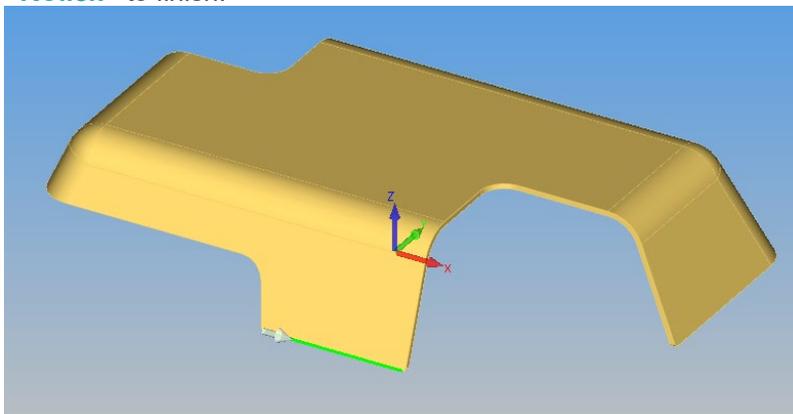


Figure 201 - Applied front line

Using **Geometry > Line** , create a line along the left hand lower edge of the model as shown below.

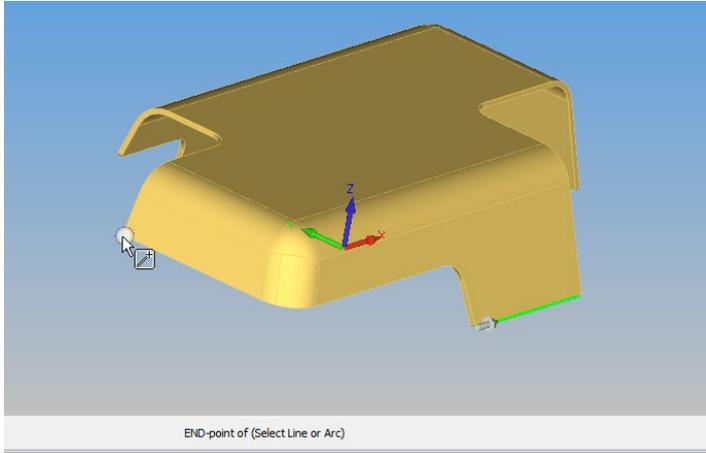


Figure 202 - Left line start point

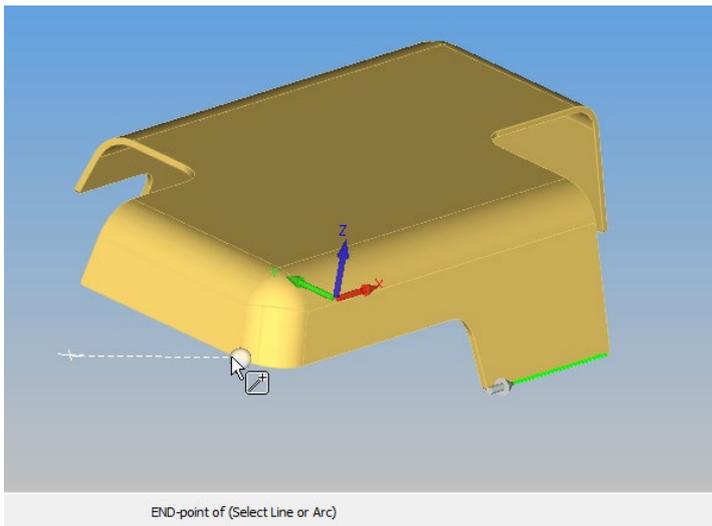


Figure 203 - Left Line end point

<RClick> to finish.

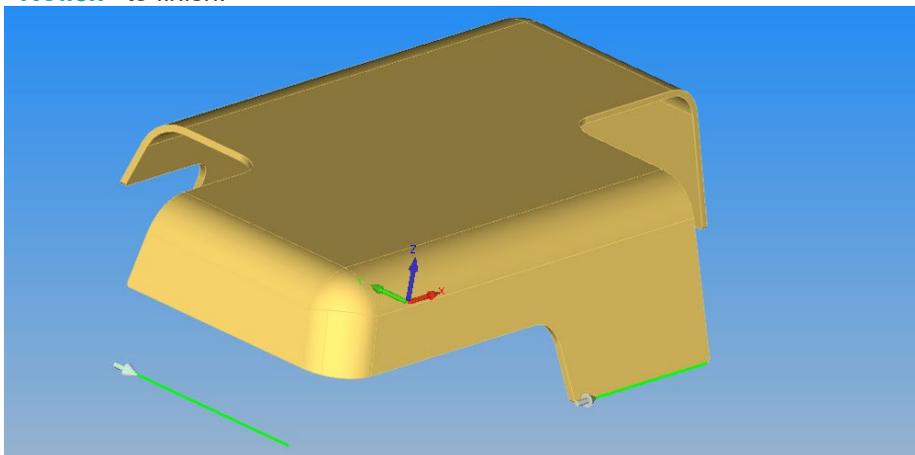


Figure 204 - Applied left line

Using **Edit > Move** , select **[All]** then **[Finish (ESC)]**.

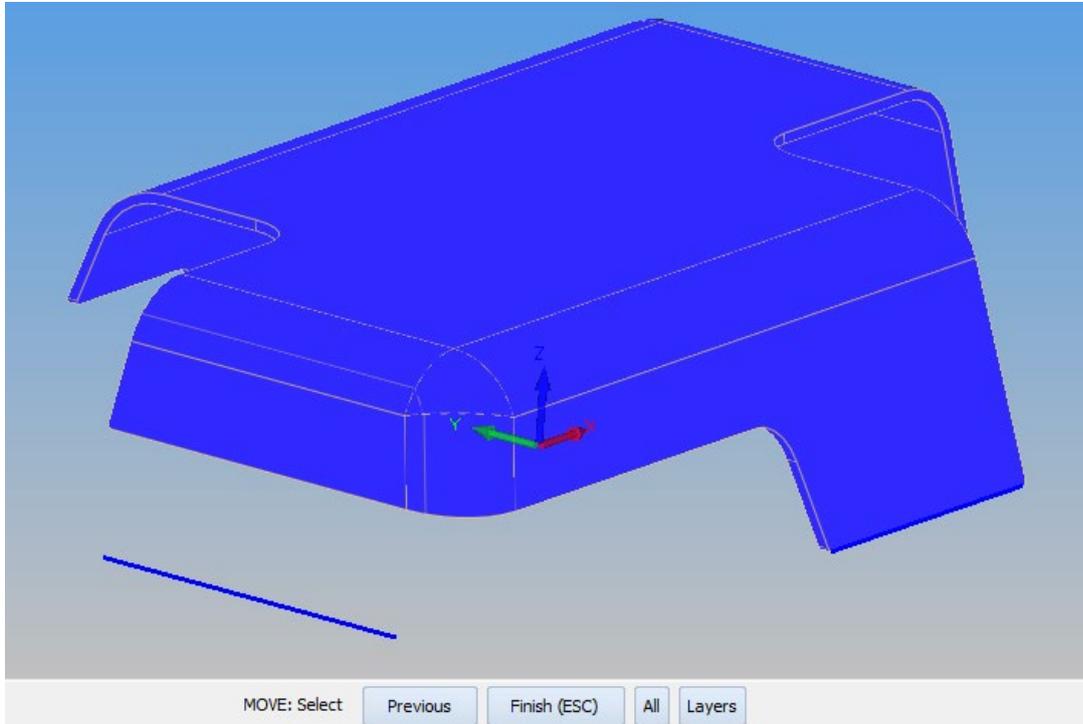


Figure 205 - Items selected to Move

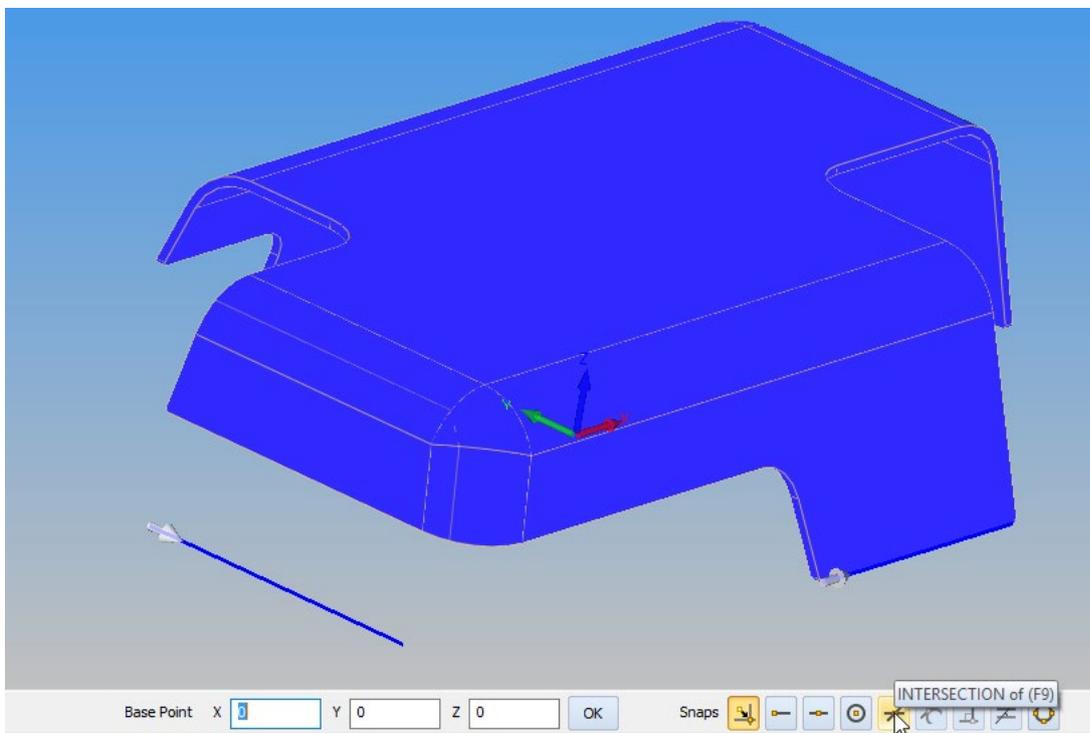


Figure 206 - Base point selection

For the Base Point, use the **<F9>**. Key or the **Intersection of (F9)** button, then select the two line you created.

When prompted, set the new location as X0, Y0 & Z0

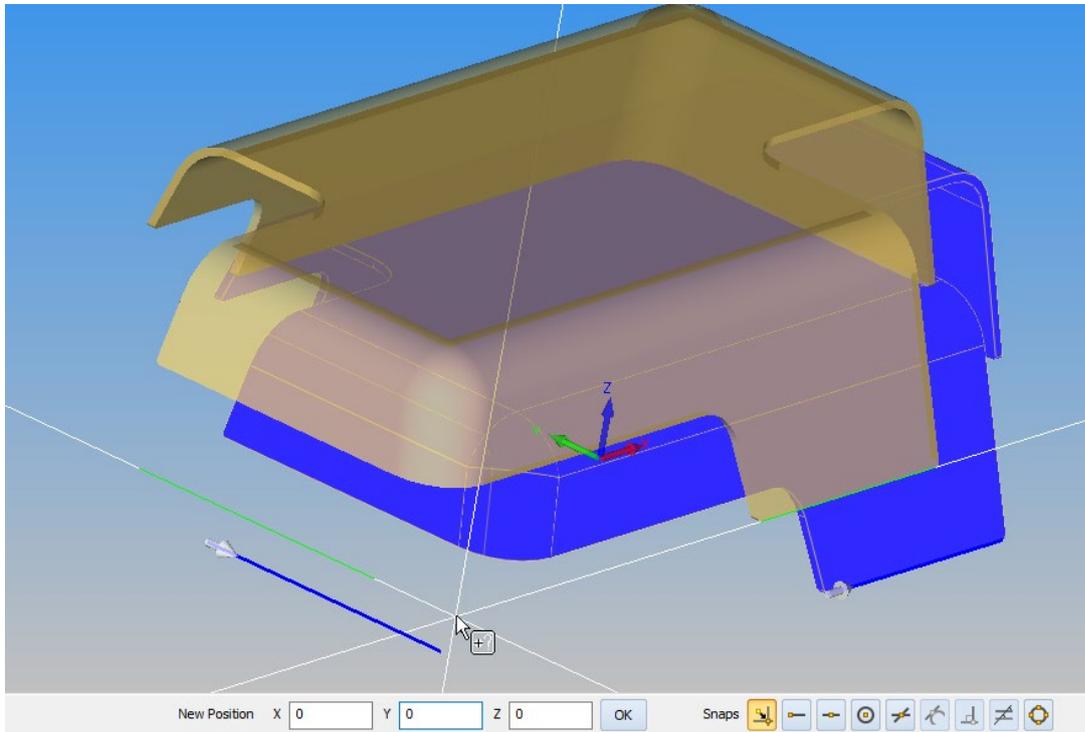


Figure 207 - New location for the part

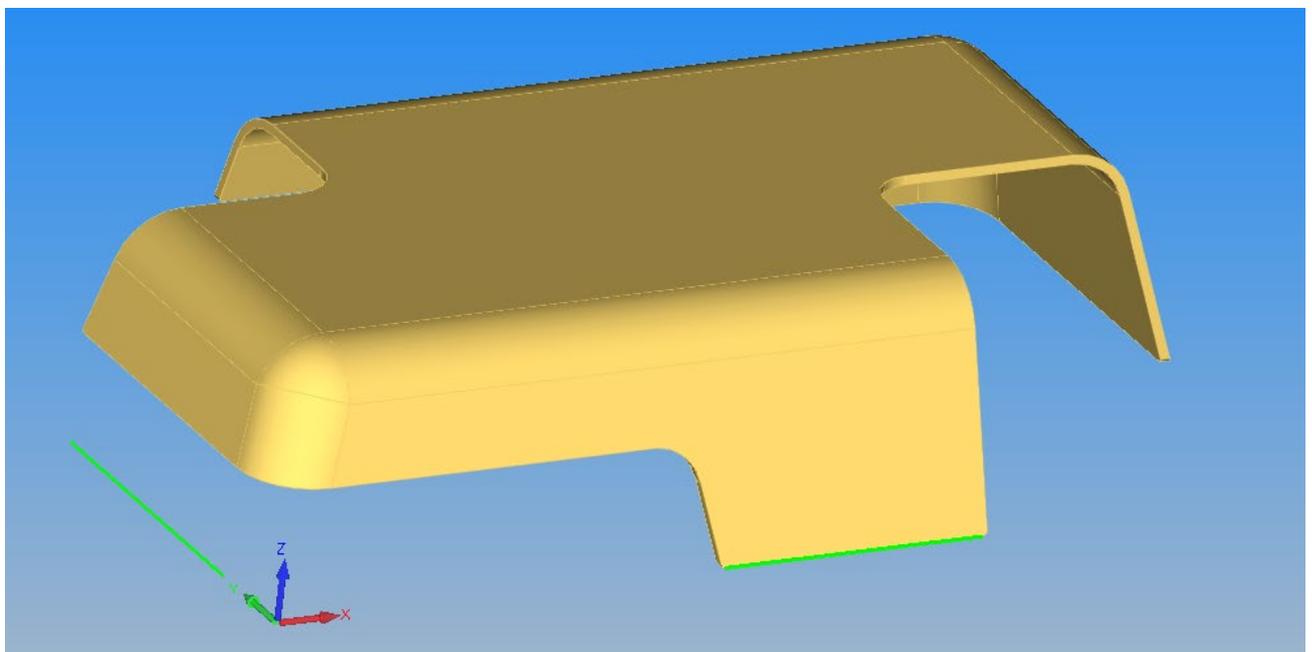


Figure 208 - Correctly relocated part

Extract the 3D Edge

Select **SOLID MODEL EXTRACT > 3D Edge Extraction** 

The first dialogue allows you to specify the chord tolerance used when the polyline is created. When you point at the inside edge the system will automatically track along tangential edges, in this case all the way around.



If the tracking stops when using this command, it is usually because there is a choice of direction. In cases such as those, you may need to **<RClick>** to finish selecting one section, then **<LClick>** again to continue the process.

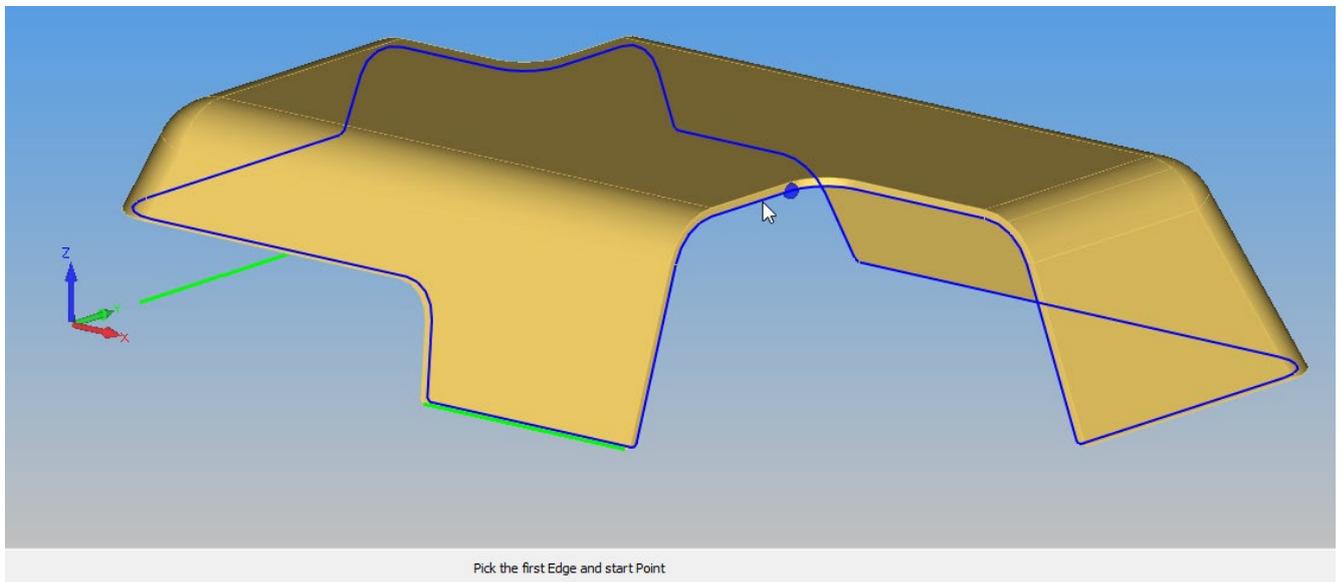


Figure 209 - 3D Edge Extraction creating a complete path

<LClick> to select this edge then **<RClick>** to create the poly line.



Figure 210 - 3D Edge Extraction seen in unshaded mode

Identifying the faces to machine

To use the polyline for trim machining it may prove beneficial to identify the edge faces first.

Select **SOLID MODEL UTILITIES > Set Face Colour** 

When you point at the inside edge face this is highlighted and the Face Selector dialogue is displayed.

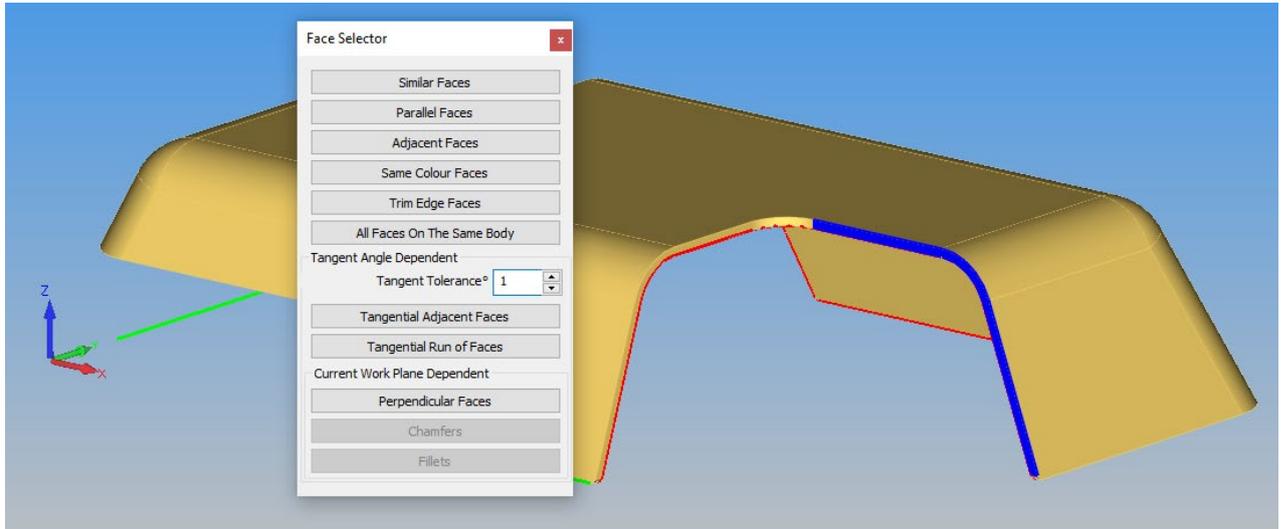


Figure 211 - First face selected and the Face Selector dialogue

The Face Selector is an aide to selecting multiple faces automatically. Select **Tangential Run of Faces** and the system will automatically track along tangential faces, in this case all the way around.

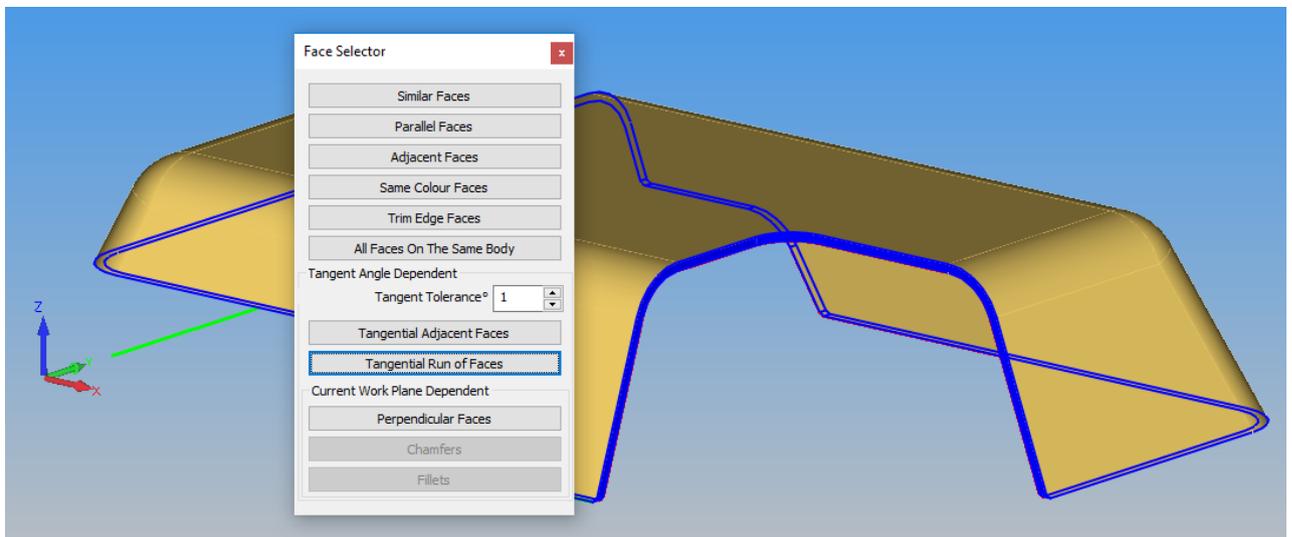


Figure 212 - Selecting all edge faces using Tangential Run of Faces



If this method does not select the required faces, each face can be selected individually.

<RClick> to display the colour selector dialogue, pick a colour and the faces will then be displayed in the selected colour.

For machining, it may be necessary to reverse the direction of the polyline using the option under

3D > Edit 3D Polyline  and set the starting position using **EDIT > Start Order > Start Pt** 

Setting the start point on a polyline can only be done by using either **END Point of (F6)** or **MID-point of (F7)** Snapping functions.

Select a suitable tool and using **MACHINE > Cut Spline or Polyline**  cut the extracted polyline making using the options **Parallel to Solid Model Faces**. Select one face and when the face selector is displayed select **[Same Colour Faces]**

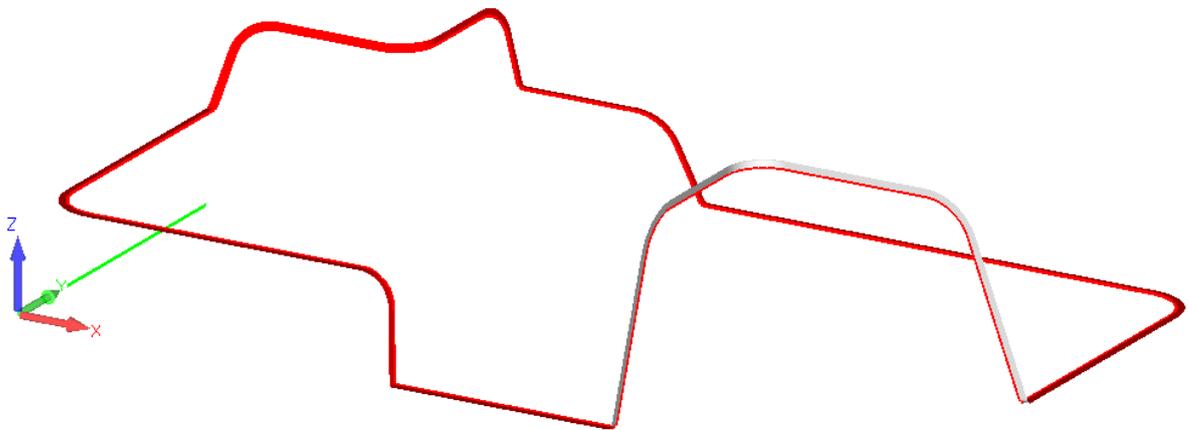


Figure 213 - Solid Model hidden to show faces to machine

Have a go using “#9 poor trim part.SLDPRT”, you will notice that when using **3D Edge Extraction** it does not automatically trace the complete edge as this is not a good model however it does provide a realistic scenario.

The edge has to be extracted in sections then connected together in a suitable manor as problems occur around these non-tangential positions.

#9 Boundary Edges

Input the solid

Select **HOME > Input CAD**  and the following dialogue is displayed.

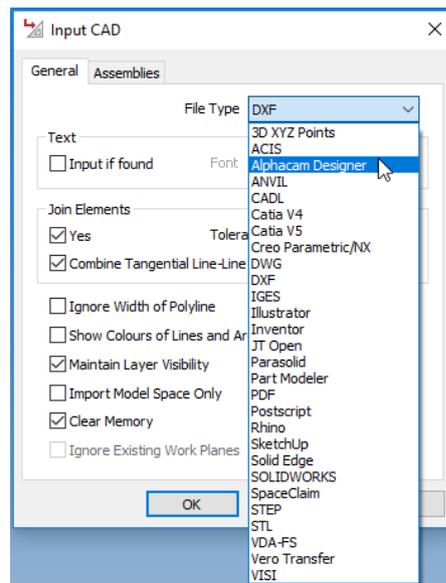


Figure 214 - Input CAD set to ALPHACAM Designer

Set the file type to **Alphacam Designer**, and then select **[OK]**.

The file selection dialogue is displayed.

Navigate to the “...\\ALP TRG 114 Feature Extraction 2020\\Examples\\Alphacam Designer Examples\\” and select the “#9 Boundary Edges.vdf” then select **[Open]**.

The model is imported and displayed on screen.

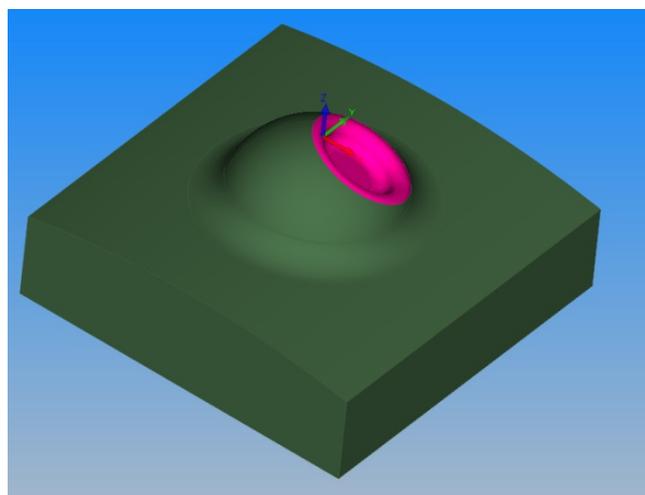


Figure 215 - Boundary Edges sample file

Align the part

Align the part so that the datum is in a suitable position for machining.

To do this you can just use the move command to move the top of the part to 150 in Z to represent the part on a trunnion table for example.

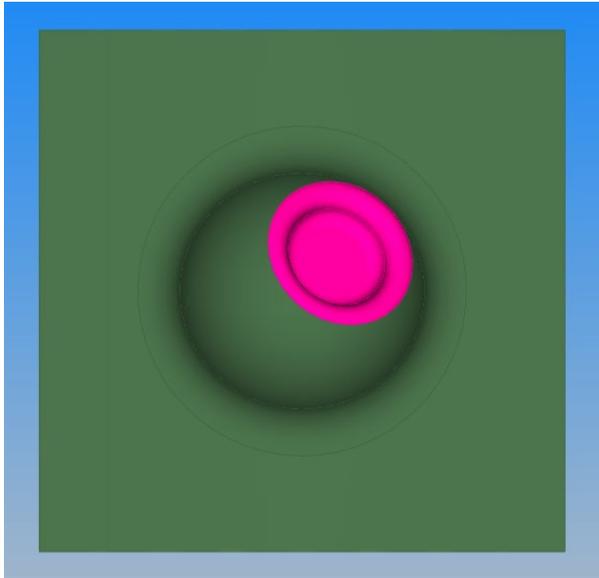
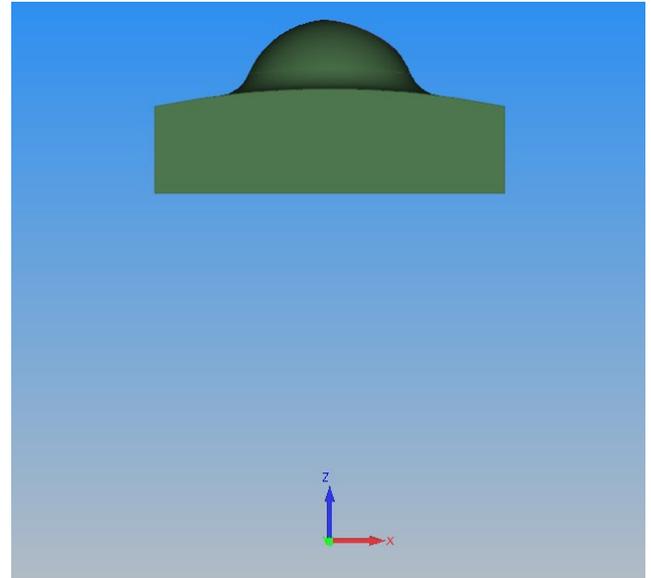


Figure 216 - Datum at a machine specific location



Select the **WORK PLANES > From Solid Model Face**  and select the face at the bottom of the pink recess, then **<RClick>** to create the plane.

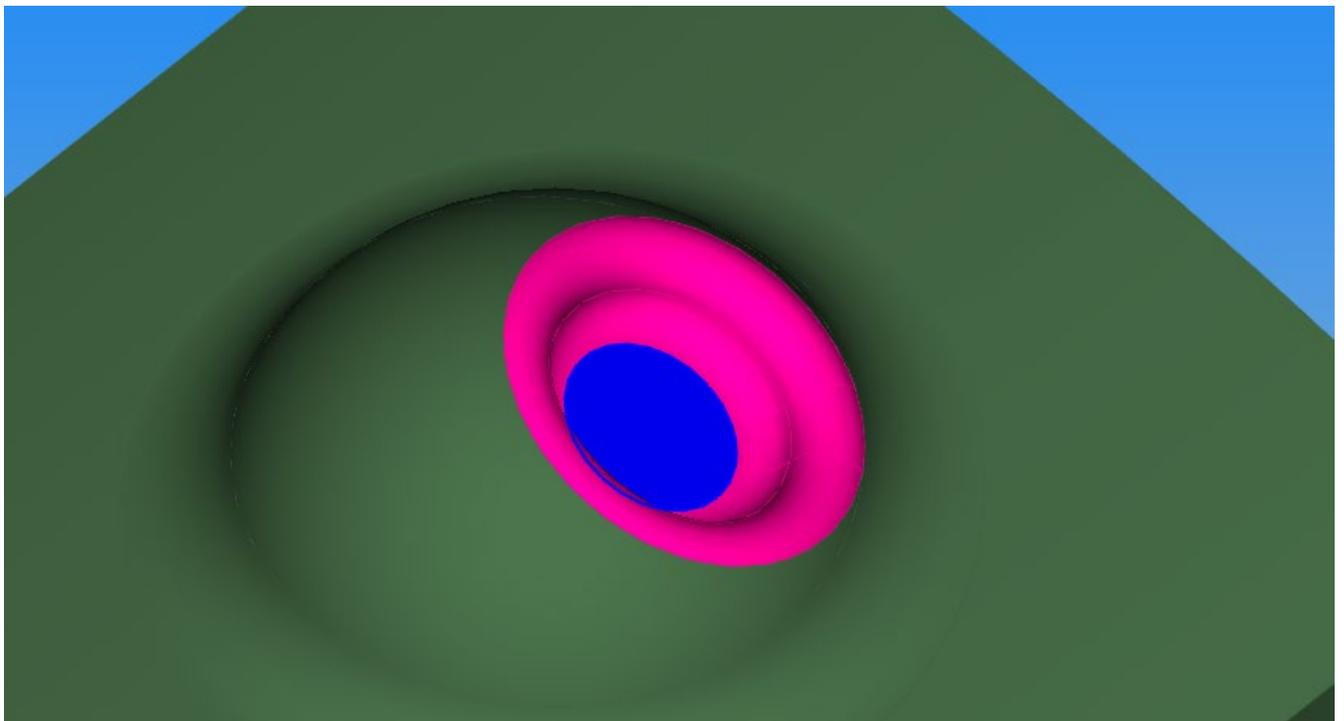


Figure 217 - Work Plane creation for angled face

The work plane is created on the face with the work plane origin normal to the global origin.

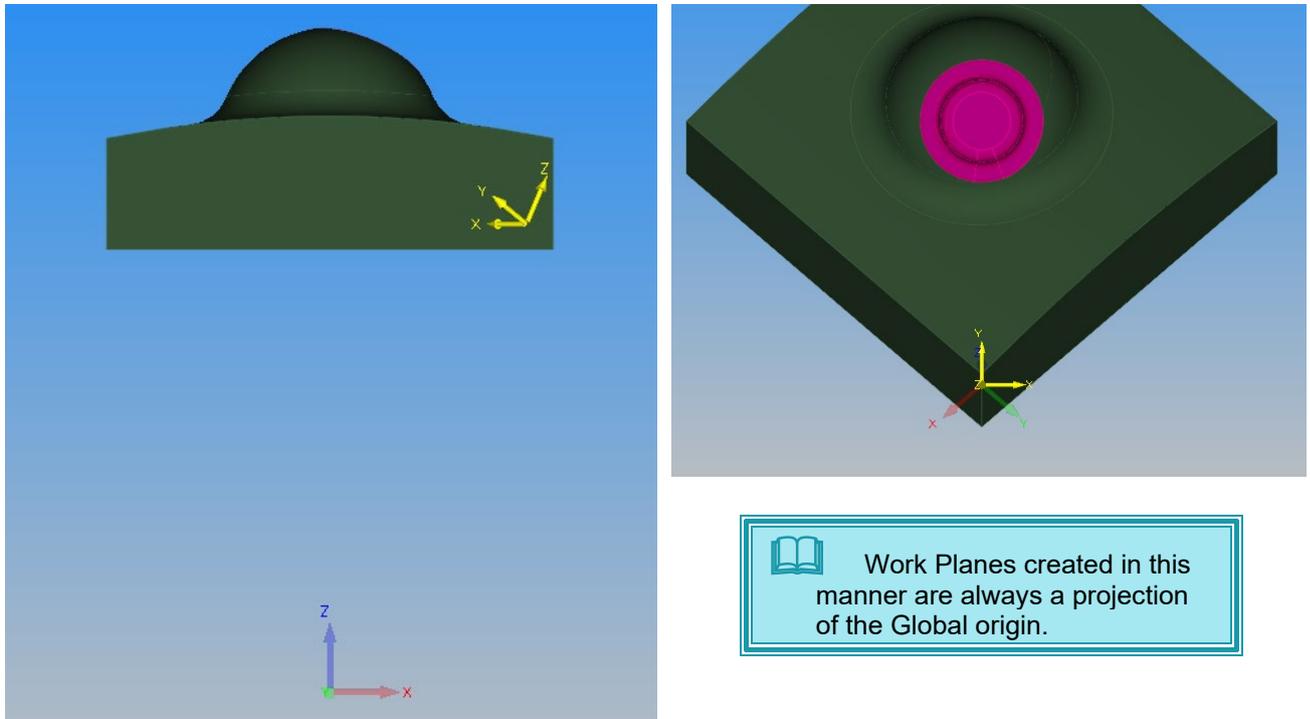


Figure 218 - Work Plane Z origin in line with original selected face

The work plane needs to be lifted to a suitable level to make the geometry easily selectable.

Select **WORK PLANES > Parallel to Current Plane**  and set the parallel value to **20**.

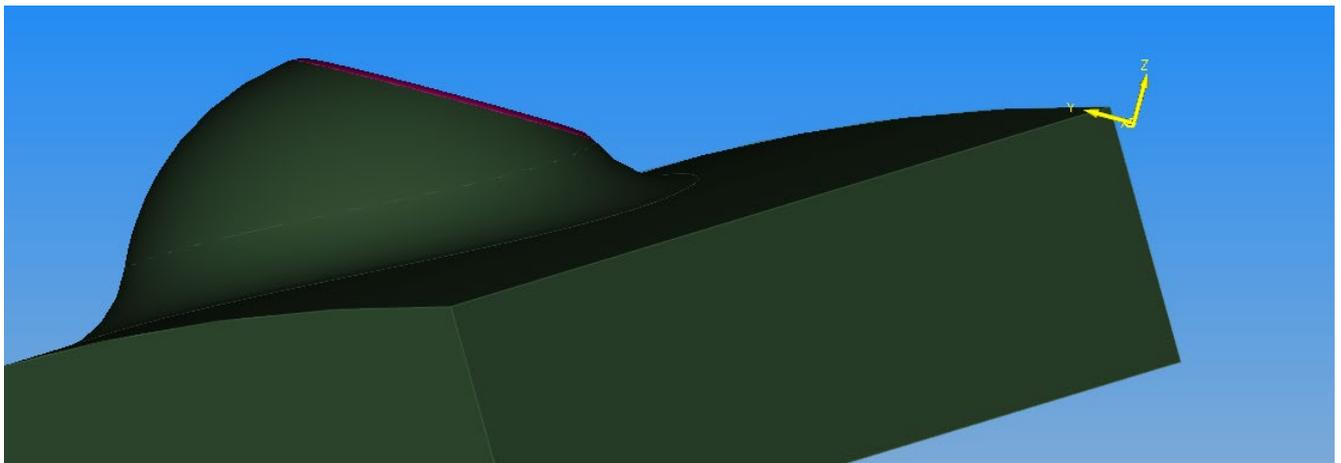


Figure 219 - Work Plane Z origin relocated in line with outer faces of recess

Extracting working geometries

To extract the outer pink edge geometry, you can either:

Use **SOLID MODEL EXTRACT > Contour from Picked Edges**  and **<LClick>** the outer pink edge. When requested to select the bottom and top Z position just **<RClick>** to accept the default extracted locations.

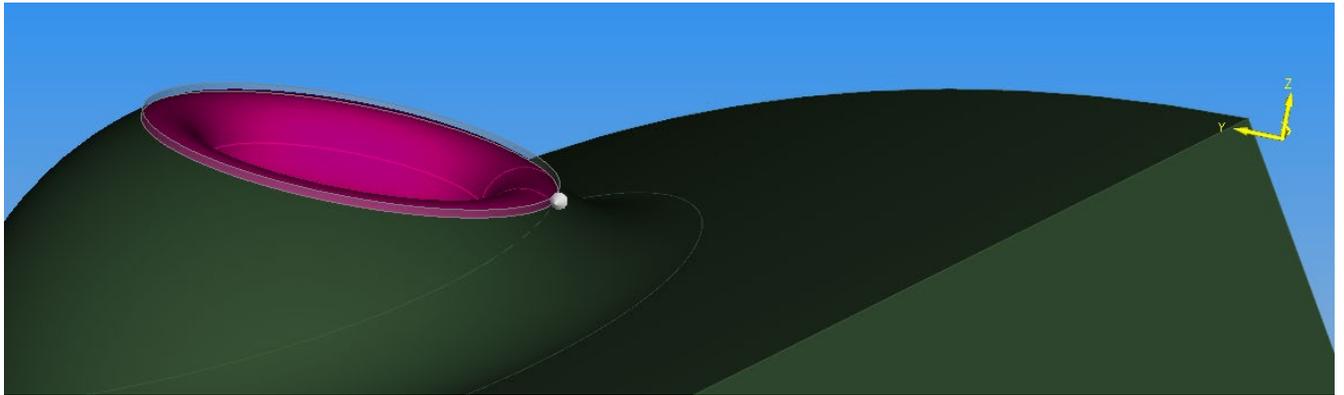


Figure 220 - Extracting edges using Picked Edges

The Extracted edge geometry values can be reset using **3D > Set Geometry Z Levels**  and setting the top and bottom values to **0**.



Figure 221 - Resetting Z levels to a common zero level

Or use **SOLID MODEL EXTRACT > Projected Face Outlines to Work Plane**



select the outer rim faces (2 of them) and **<RClick>** to finish.

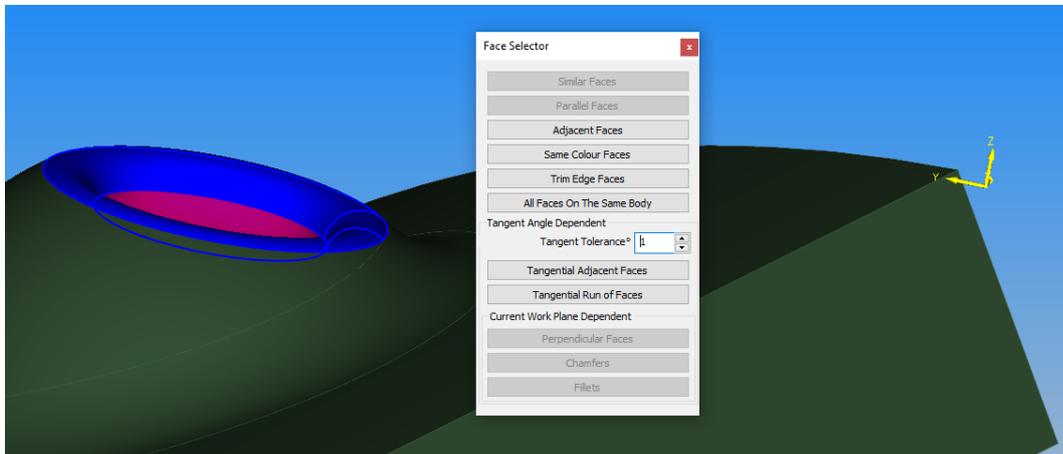


Figure 222 - Extracting using Picked Faces

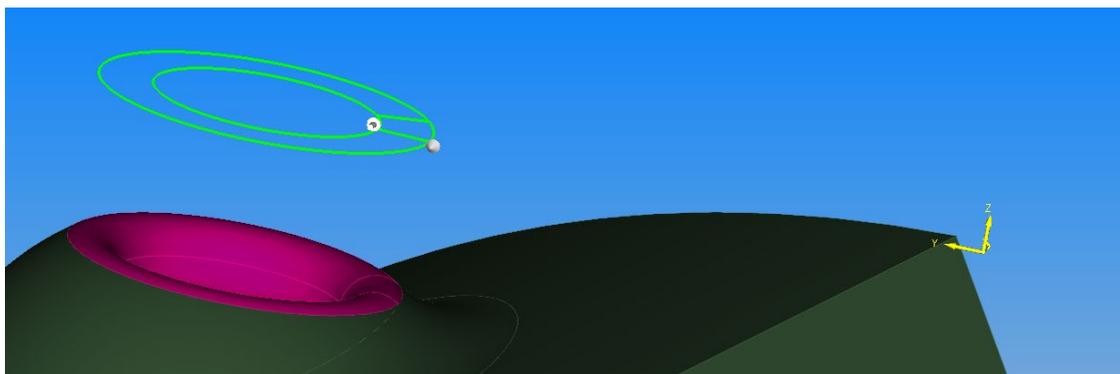


Figure 223 - Extracted face outlines

In this case the geometries will require editing to create the boundary.

This is easily achieved using **EDIT > Fabricate > Unite**



then selecting all the extracted face outlines to create two single geometries.

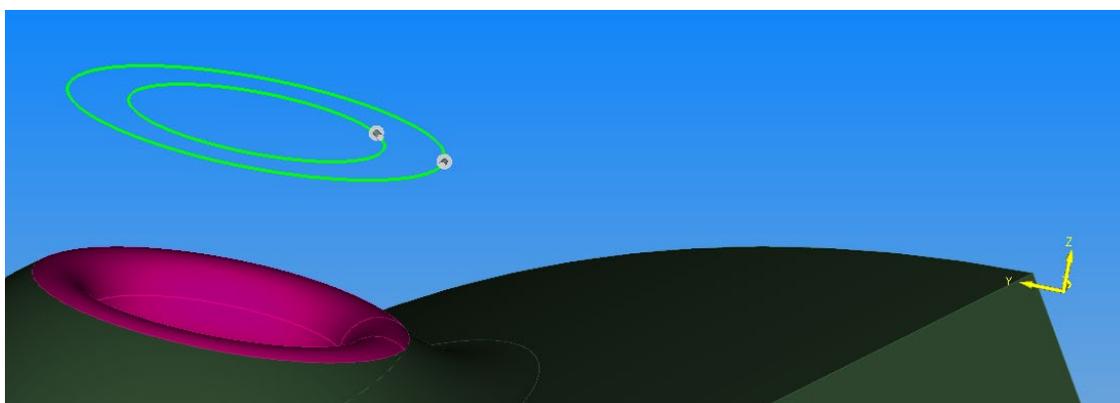


Figure 224 - United geometries

You can then delete the unrequired items.

For other Face edges for Flat land boundaries for Vertical 3D Machining the

SOLID MODEL EXTRACT > Projected Face Outlines to Work Plane  option produces more useable results.

Cancel any active work planes.

Use **SOLID MODEL EXTRACT > Projected Face Outlines to Work Plane**  select the top shallow curved face, and **<RClick>** to finish.

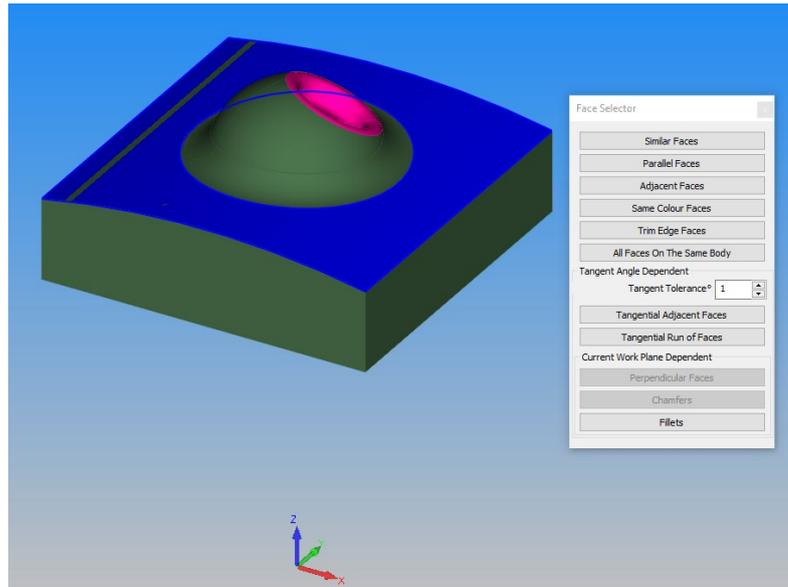


Figure 225 - Face selection for Projected Face Outlines to Work Plane

<RClick> to finish and create the geometries, when geometries are used as boundaries they can be above, below or even in the part.

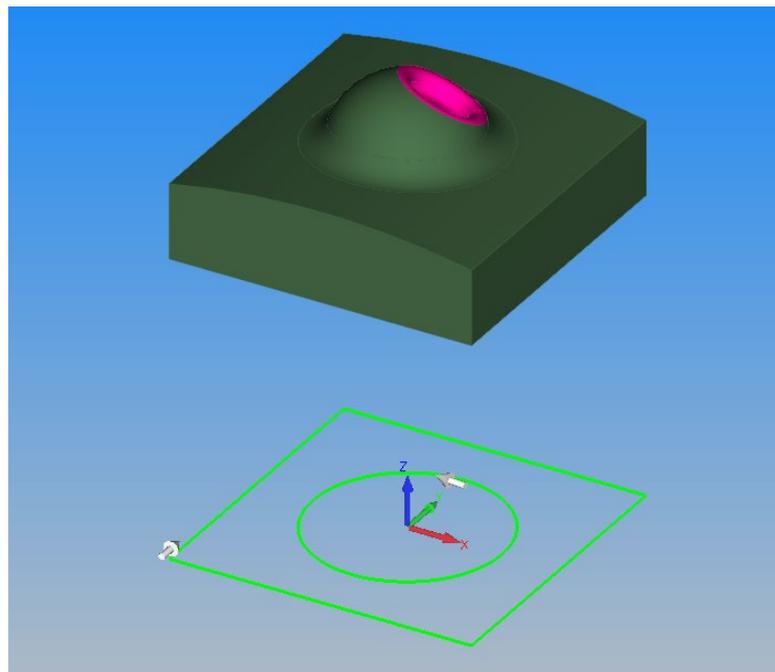


Figure 226 - Outlines projected to Flat-Land below the part

#10 Saw Cuts

Select **HOME > Input CAD**  and the following dialogue is displayed.

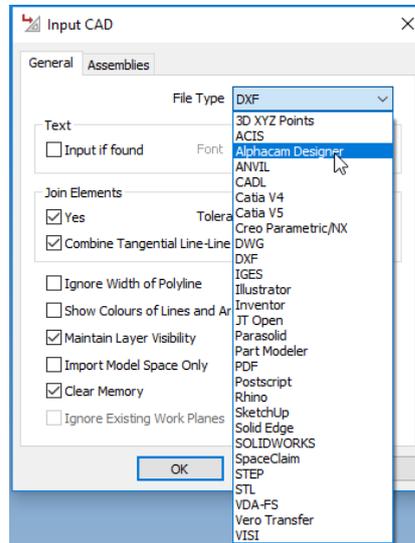


Figure 227 - Input CAD set to ALPHACAM Designer

Set the file type to **Alphacam Designer**, then select **[OK]**.
The file selection dialogue is displayed.

Navigate to the “...\\ALP TRG 114 Feature Extraction 2020\\Examples\\Alphacam Designer Examples\\” folder and select the “#10 Saw Cuts.vdf” then select **[Open]**.
The model is imported and displayed on screen.

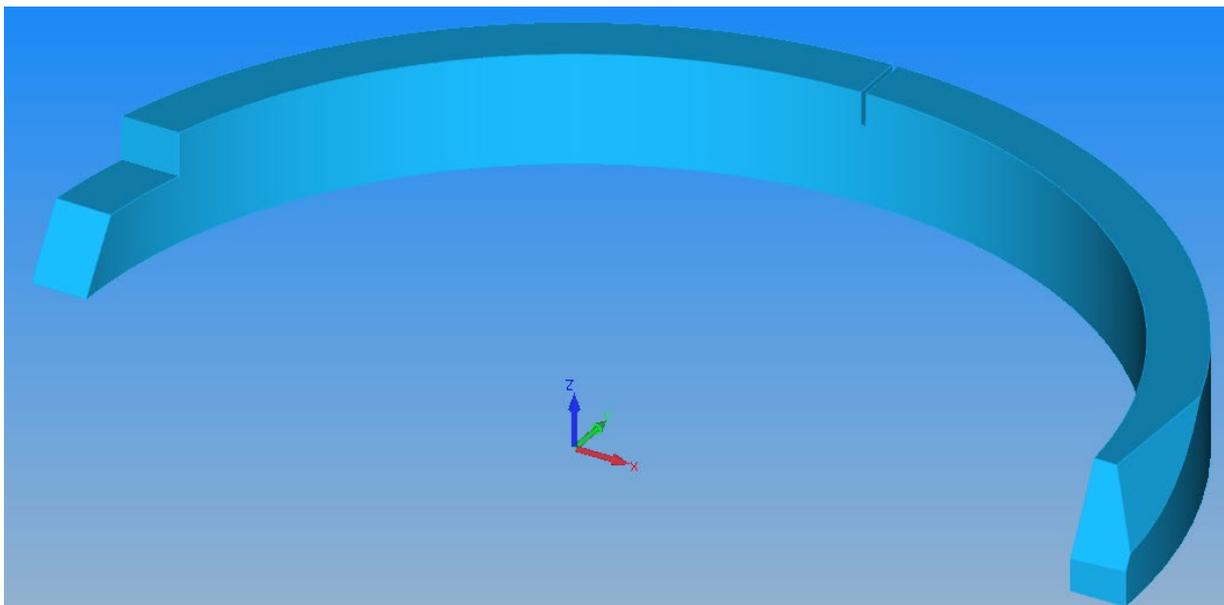


Figure 228 - Saw Cuts sample file



Select **SOLID MODEL EXTRACT > Edge for Sawing**

Pick one of the dark blue face as shown below, when the faces selector is displayed select **[Parallel Faces]**, the other faces parallel to the original picked item will be selected then **<RClick>** to finish.

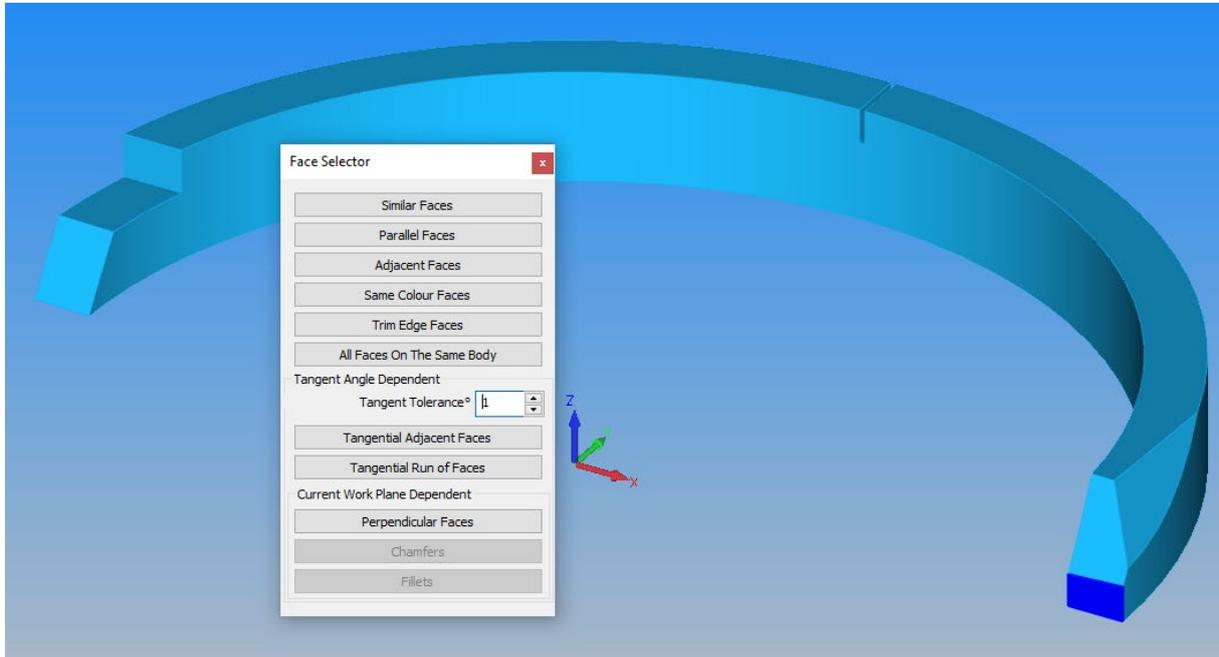


Figure 229 - Using the Face Selector for Parallel Vertical faces



Select **SOLID MODEL EXTRACT > Edge for Sawing**

Pick one of the dark blue faces as shown below, when the faces selector is displayed select **[Parallel Faces]**, the other faces parallel to the original picked item will be selected then **<RClick>** to finish.

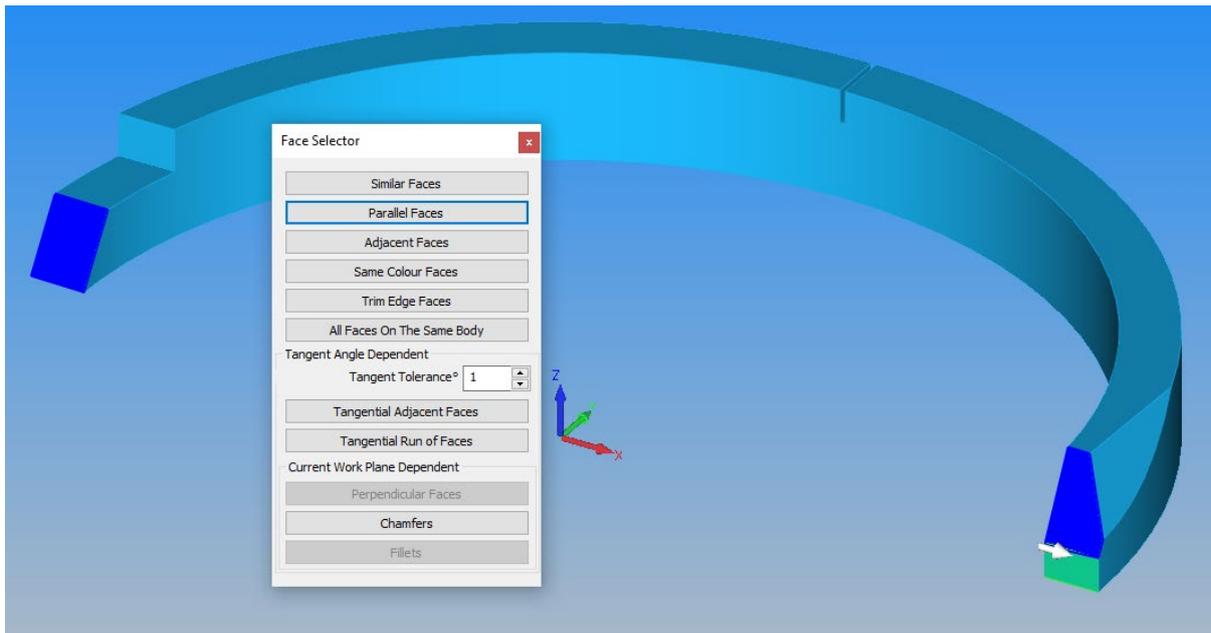


Figure 230 - Using the Face Selector for Parallel Angled faces

Select **SOLID MODEL EXTRACT > Edge for Sawing** 

Pick the dark blue faces as shown below, when the faces selector is displayed **<RClick>** to finish. As there are no other faces similar.

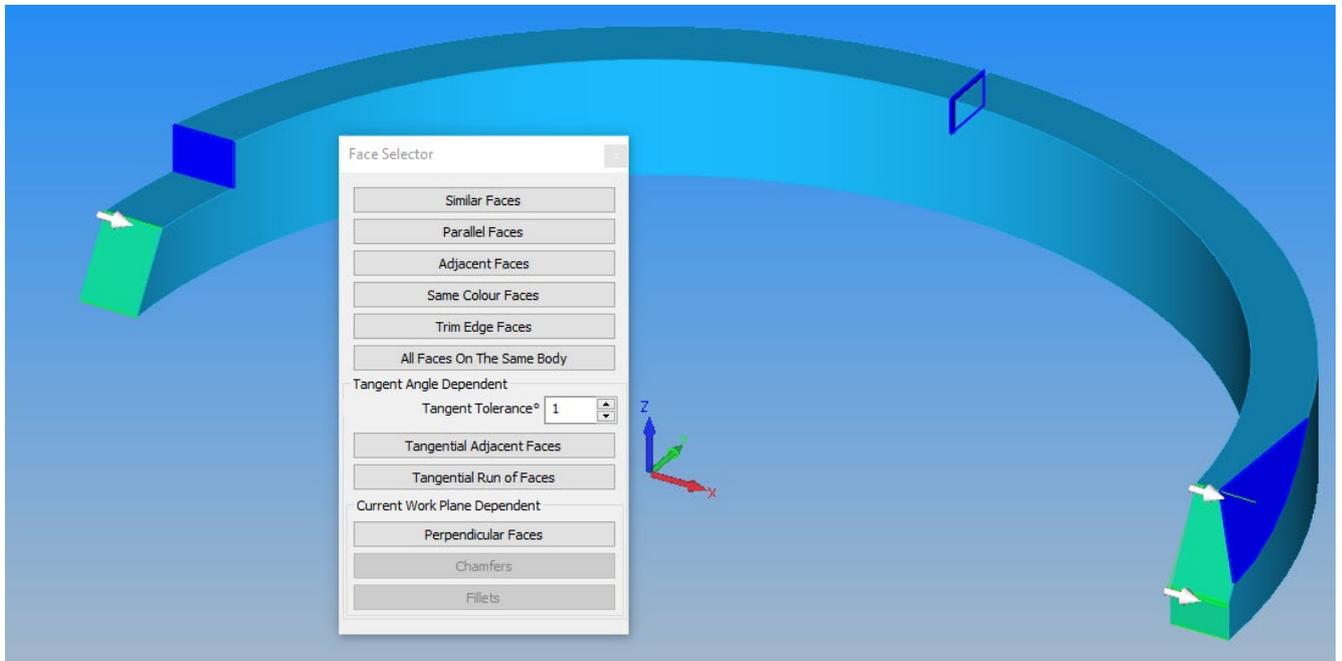


Figure 231 - Single face selection

Using these geometries, the saw cuts can be applied using the **MACHINE > Sawing**  machining option.

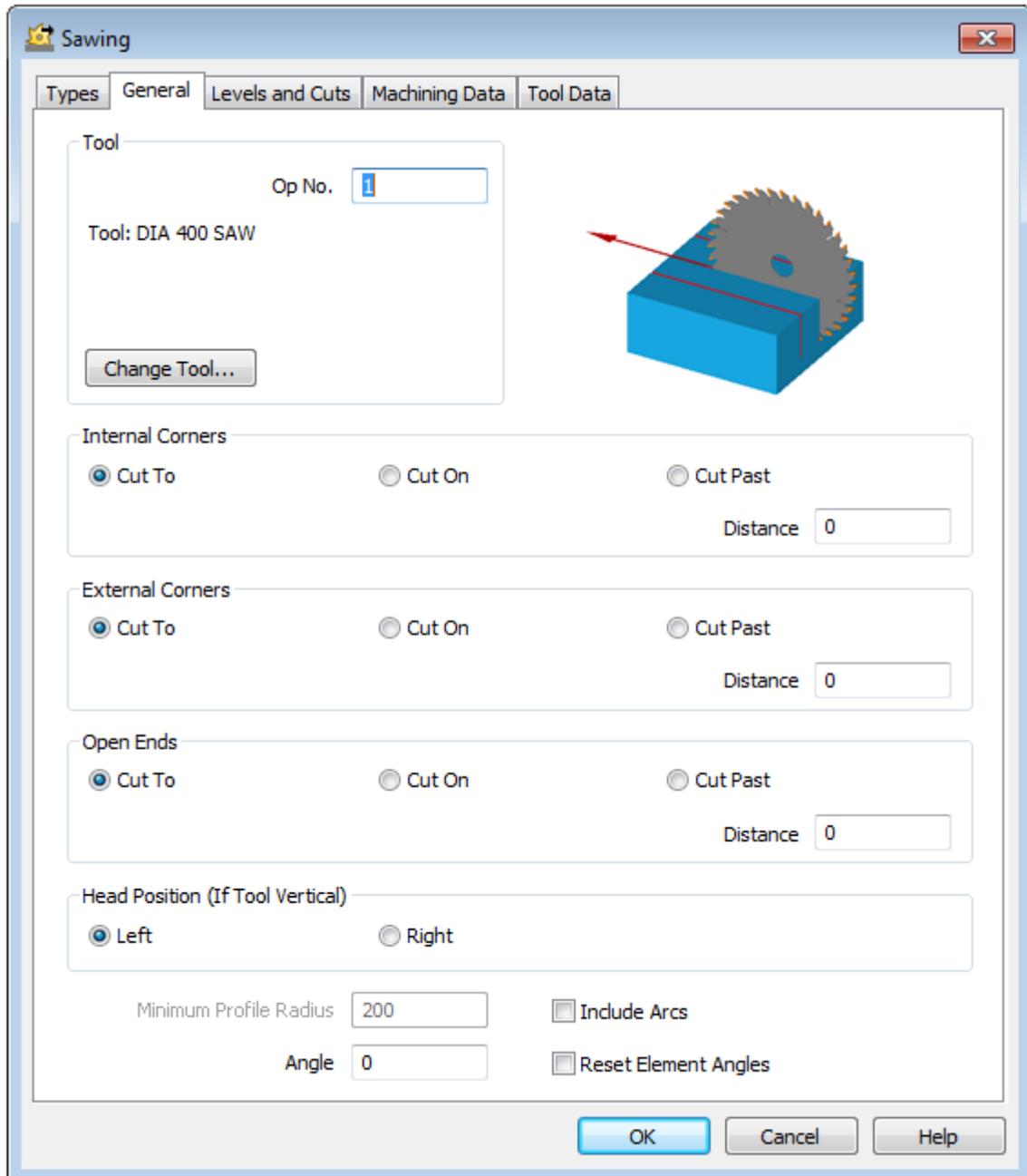


Figure 232 - General tab on Sawing

This dialogue allows you to set how the tool path is created with respect to **Internal**, **External** corners and at **Open** ended geometry/elements. You can also select the head side of the saw with respect to the geometry and any tilt angle.

Cut To

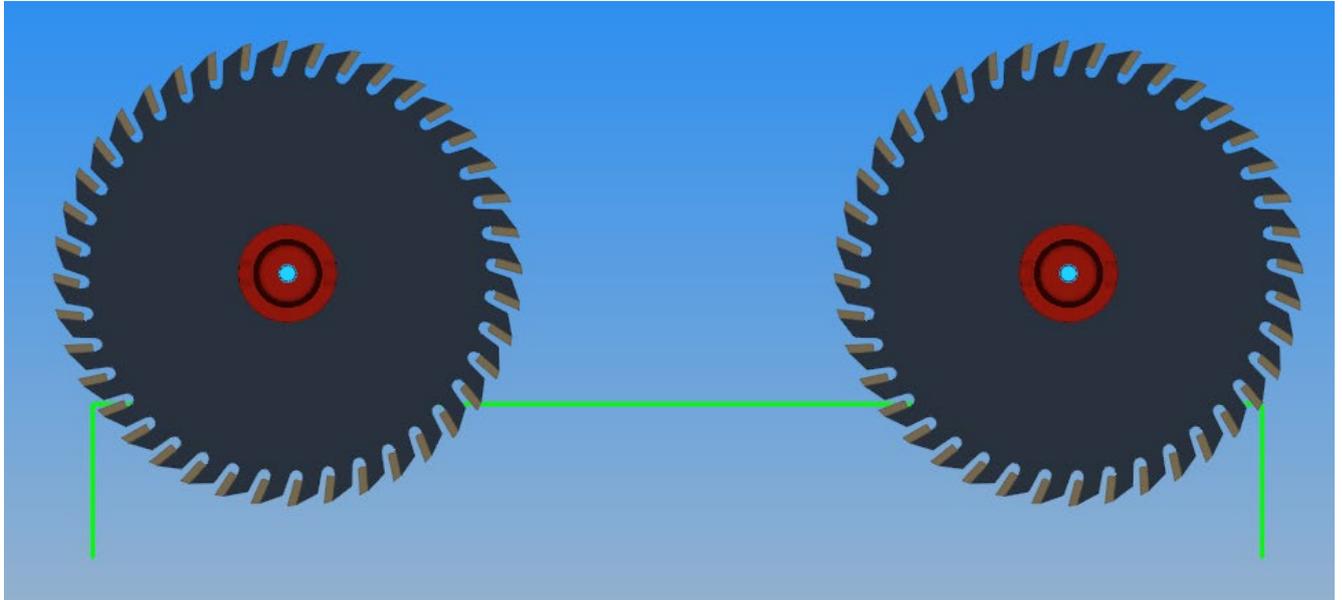


Figure 233 - Explanation of Cut To

Using the  **Cut To** option, the Saw will cut the required geometry or Solid Face from a point where the peripheral edge of the saw is tangent to the chosen item, to the same tangent point at the opposite end of the cut.

Cut On

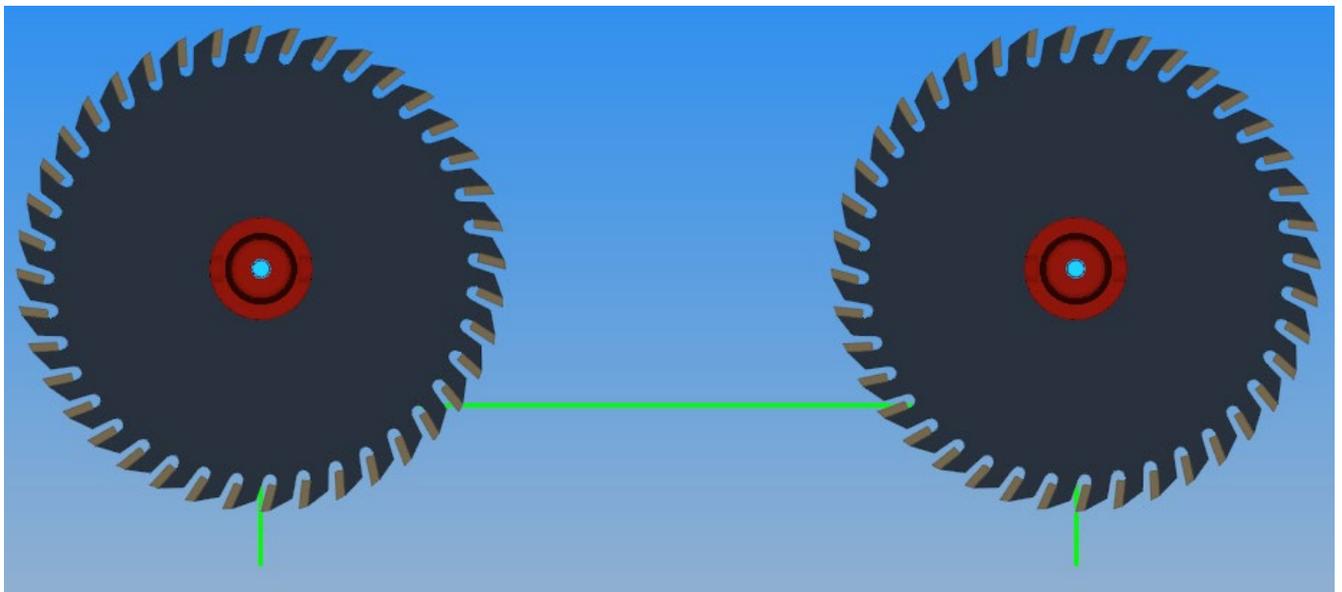


Figure 234 - Explanation of Cut On

Using the  **Cut On** option, the Saw will cut the required geometry or Solid Face from a point where the centre line of the saw is in line with the chosen item, to the same point at the opposite end of the cut.

Cut Past

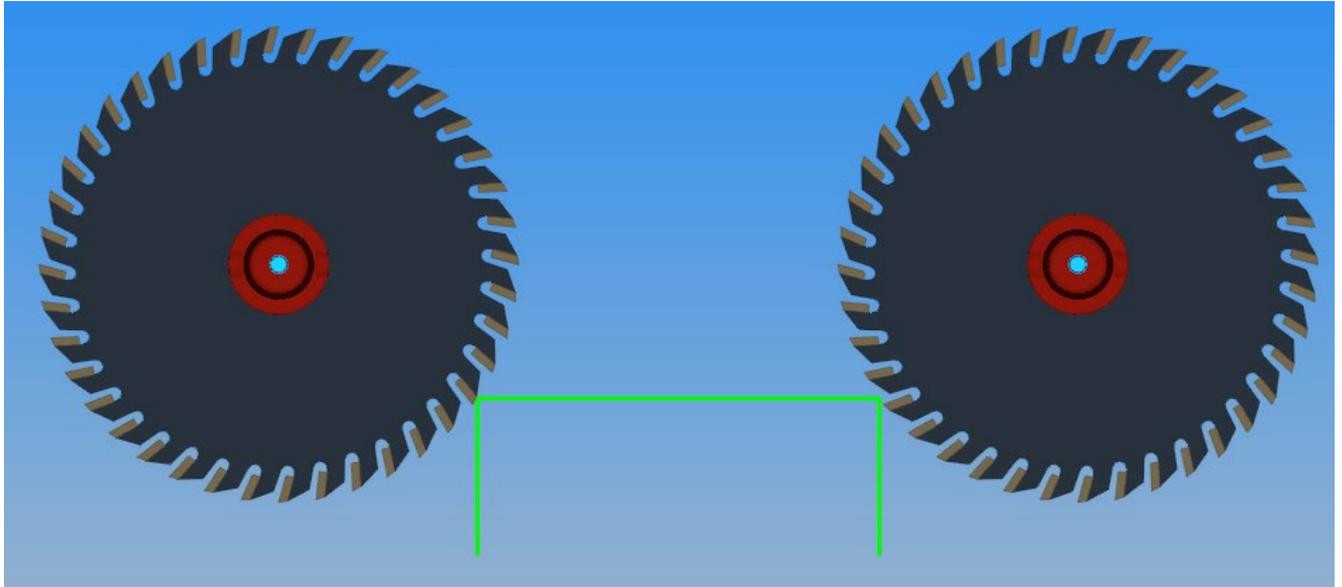


Figure 235 - Explanation of Cut Past

Using the  **Cut Past** option, the Saw will cut the required geometry or Solid Face from a point where the peripheral edge of the saw is tangent to the chosen item on the outer edge of the disc, to the same tangent point at the opposite end of the cut.

When using the Cut Past option, you also have the option to enter a value for how far past the chosen option the saw blade will commence and complete the cut.

Tricks and Tips

Introduction

Whilst ideally, you would wish Feature Extraction to extract all the features required for machining, in the correct orientation with the vectors/depths going the correct way.

In reality this is not always the case.

The way features are extracted is influenced by the methods involved in the models' construction and data structure.

The way features are required is influenced by the method of manufacture: machine type, work holding, stock material. This section is intended to provide insight into different methods of extracting that is required from those models that do not provide the ideal situation. Through feedback and customer examples ALPHACAM will be developing the Feature Extraction options so that most model features are extracted correctly automatically.

Drillable Hole Extraction

This command allows individually selected geometries to be extracted from hole features. The command line prompts you to select a circular edge or inside of a cylinder this will create a circular geometry in the Drillable Holes Layer with the Z values set. When individual drillable holes are extracted they are created in the orientation of the holes' creation. This may result in the hole being the wrong orientation for drilling.

To reverse the hole direction;

1. Activate the hole work plane.
2. Reverse the work plane.
3. Swap the top and bottom Z values of the geometry.

Measurements from the model

When there is not a work plane active, measurements can be taken from the model using the

SOLID MODEL UTILITIES > Face/Edge/Point Details  or the **CAD > Distance/Angle**  commands. Remember the values displayed can be copied and pasted into another command.

Draw Geometries on Work Plane

Sometimes it can be awkward to extract the geometry required using the feature extraction commands.

Geometries can be drawn on a work plane or flat land, whilst snapping to points on the model. Then the Z values set after taking measurements from the model.

Auto Align

If you frequently manually move models for suit positioning on the machine tool you may want to consider removing the Auto Align command from the Ribbon to avoid selection by error.

Features do not extract as required.

When extracting edges from a model face occasionally the feature geometry is not correct.

When the extracted geometry is not exactly as required, remember these are normal geometries and can be edited: broken, trimmed, and joined as normal.

The option **SOLID MODEL EXTRACT >_Projected Face Outline to Work Plane**  uses a different algorithm and may produce better results.

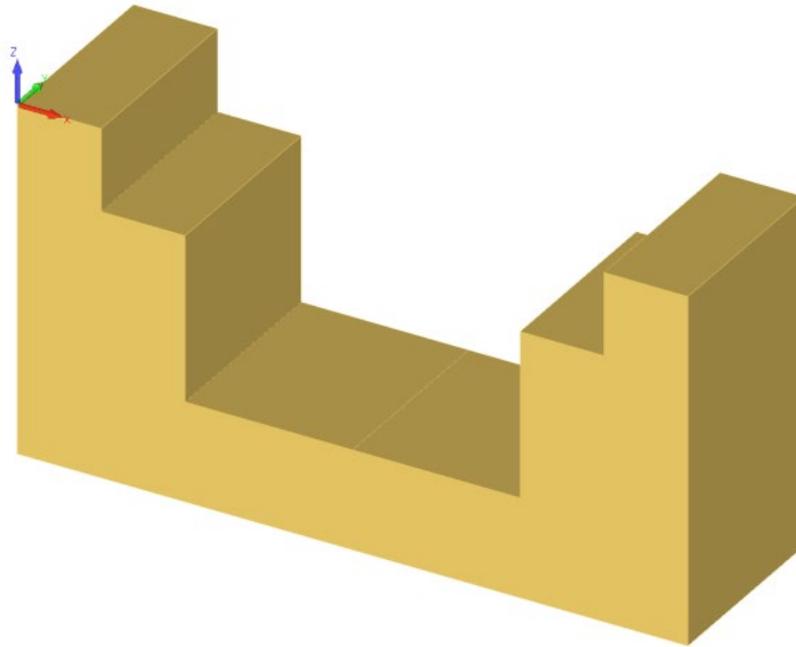
The option **SOLID MODEL EXTRACT >_Contour through the Current Plane**  can also be used to produce the required geometry.

The command **SOLID MODEL EXTRACT >_Contour around_Picked Faces**  put the resultant geometry on a work plane that is parallel to flat land.

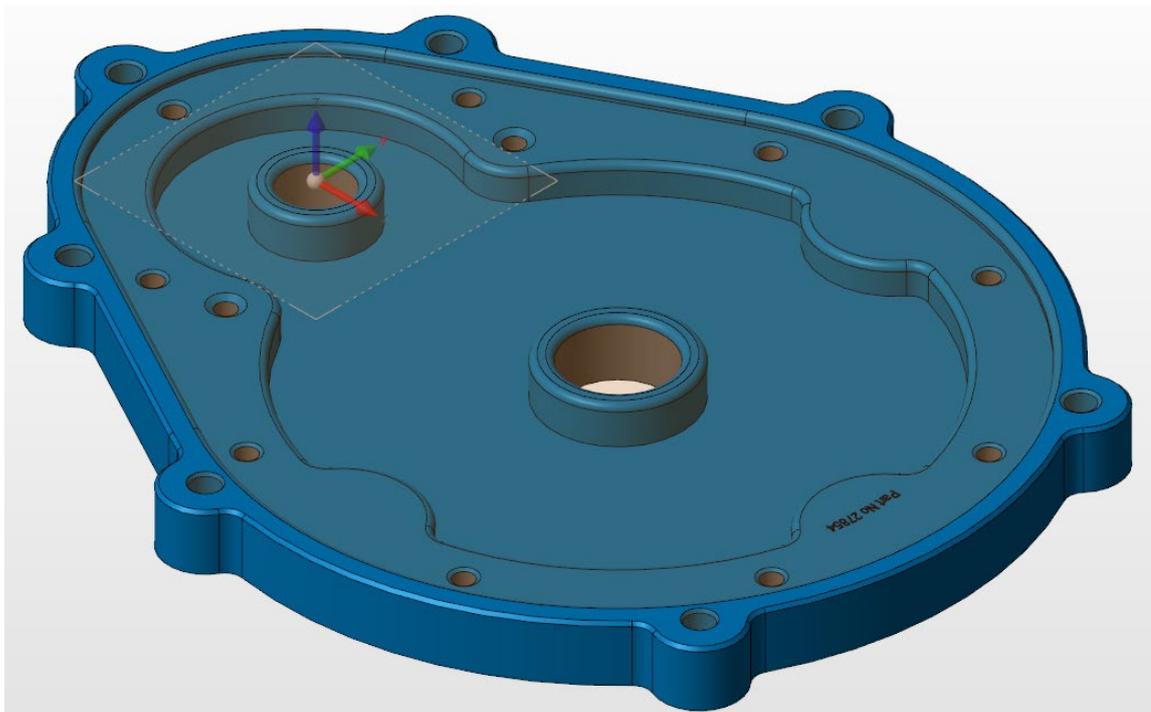
Training Course Example Models

“...\\ALP TRG 114 Feature Extraction 2020\\Examples\\Alphacam Designer Examples\\”

#11 Fork Plate



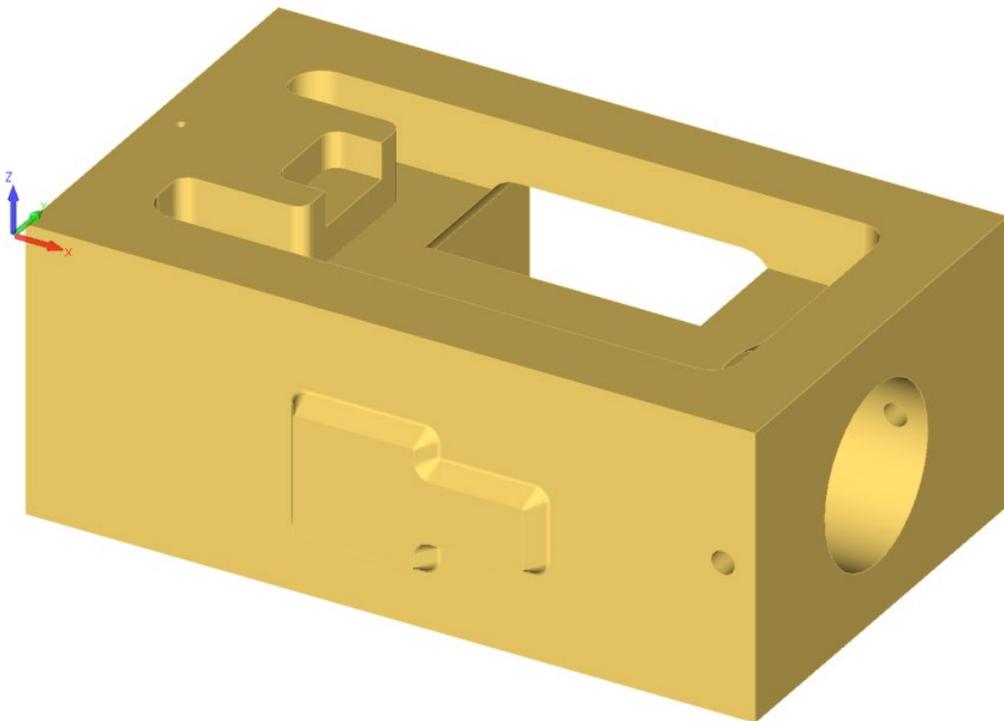
#12 Housing plate



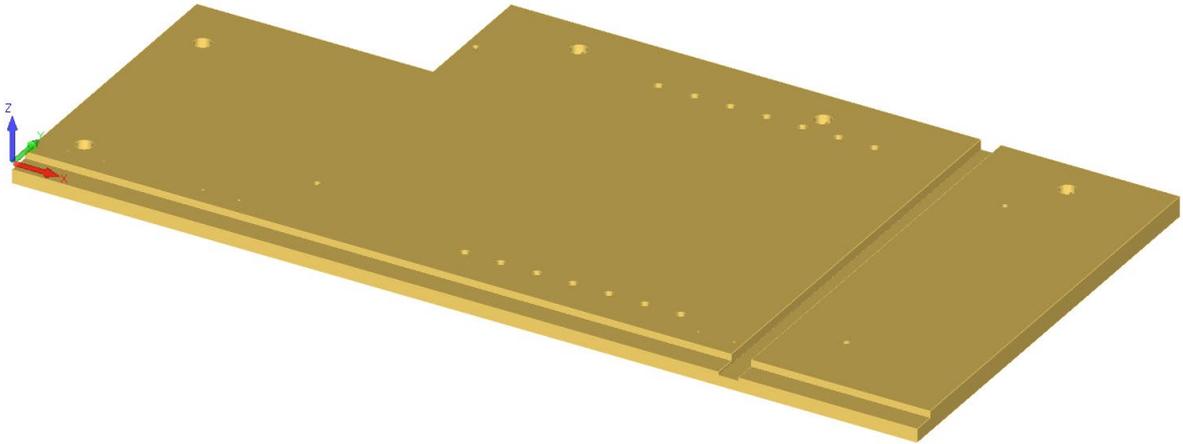
#13 Router Part



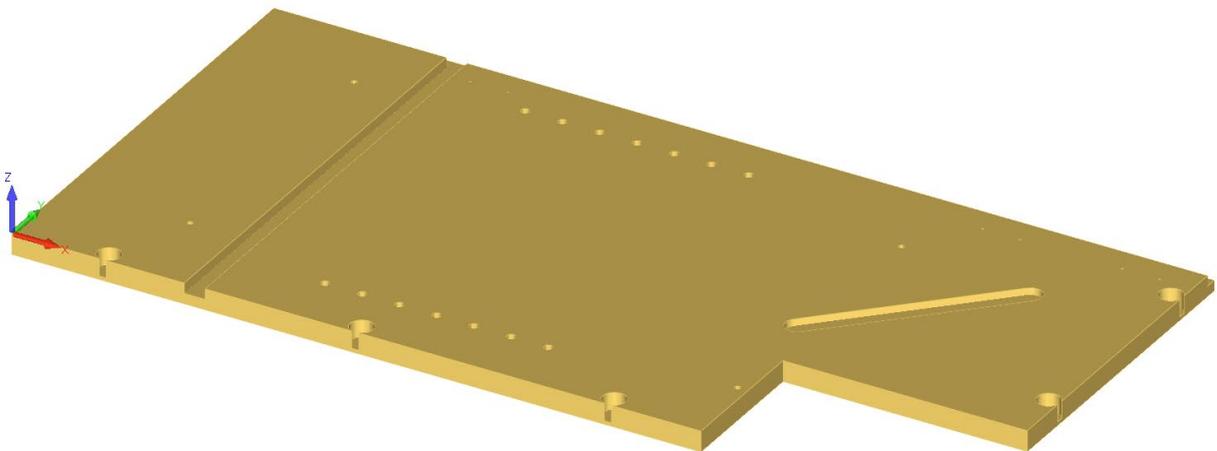
#14 Orthogonal Planes



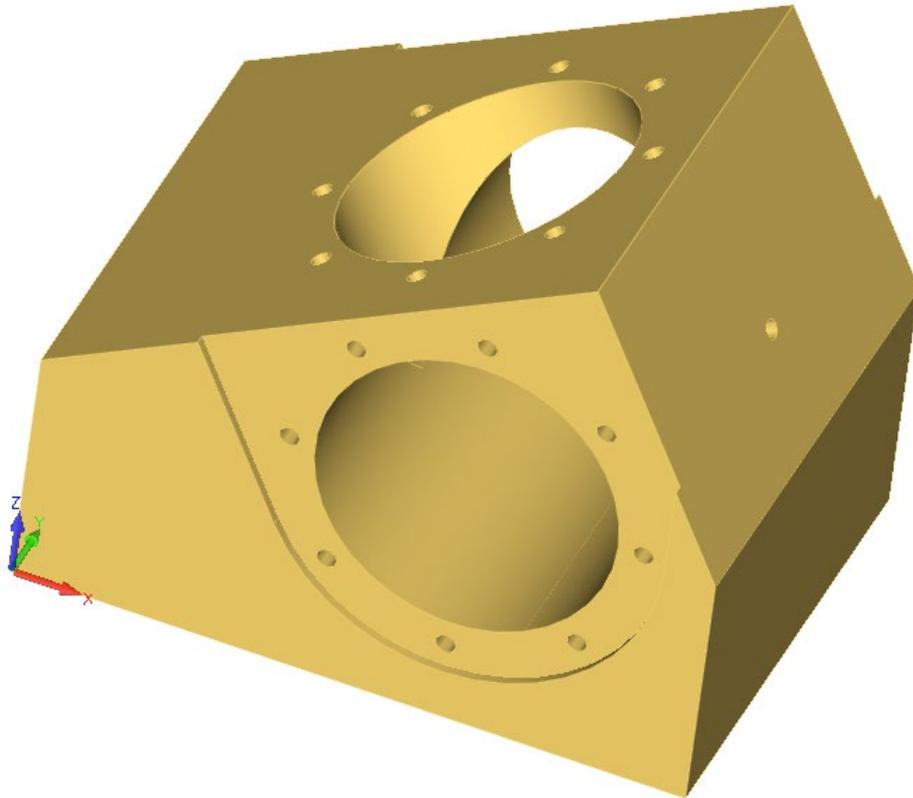
#15 Cam and Dowel fixing Panel



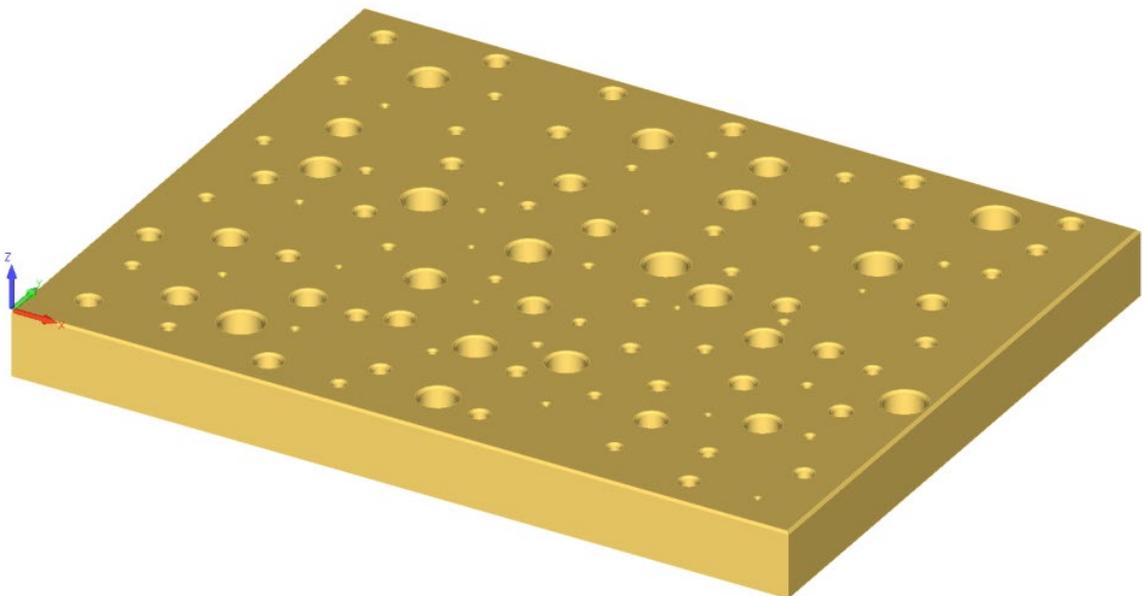
#16 Rafx Panel



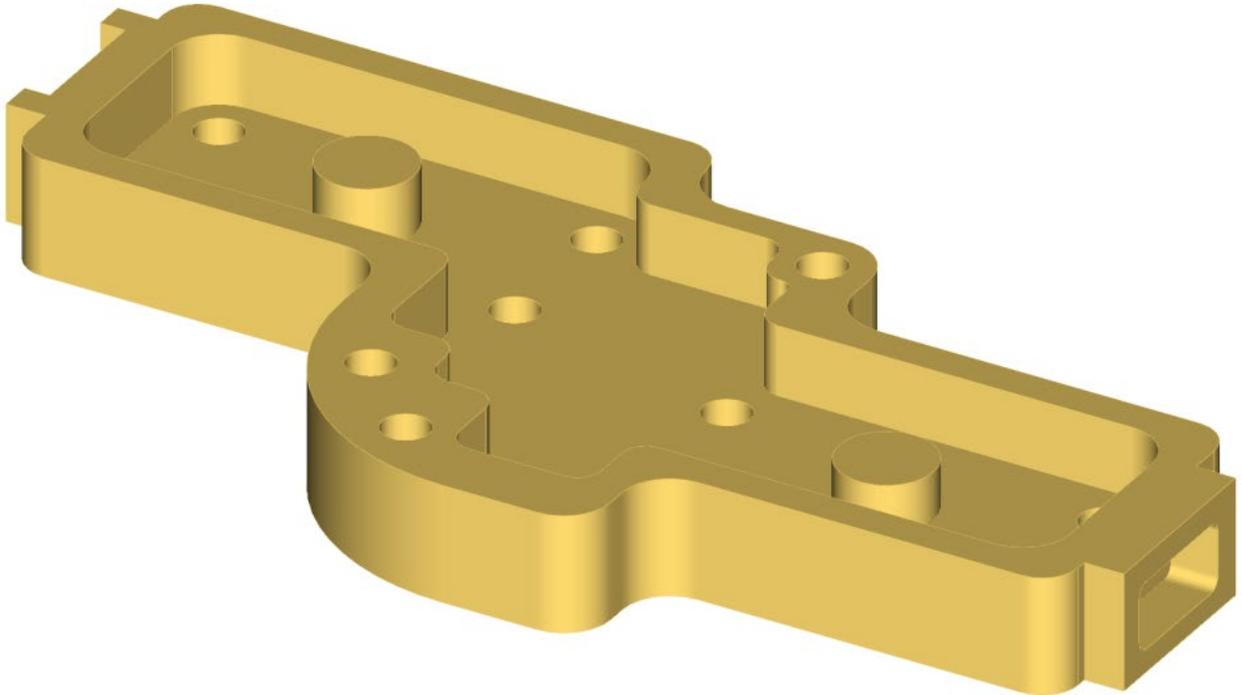
#17 Planes Part



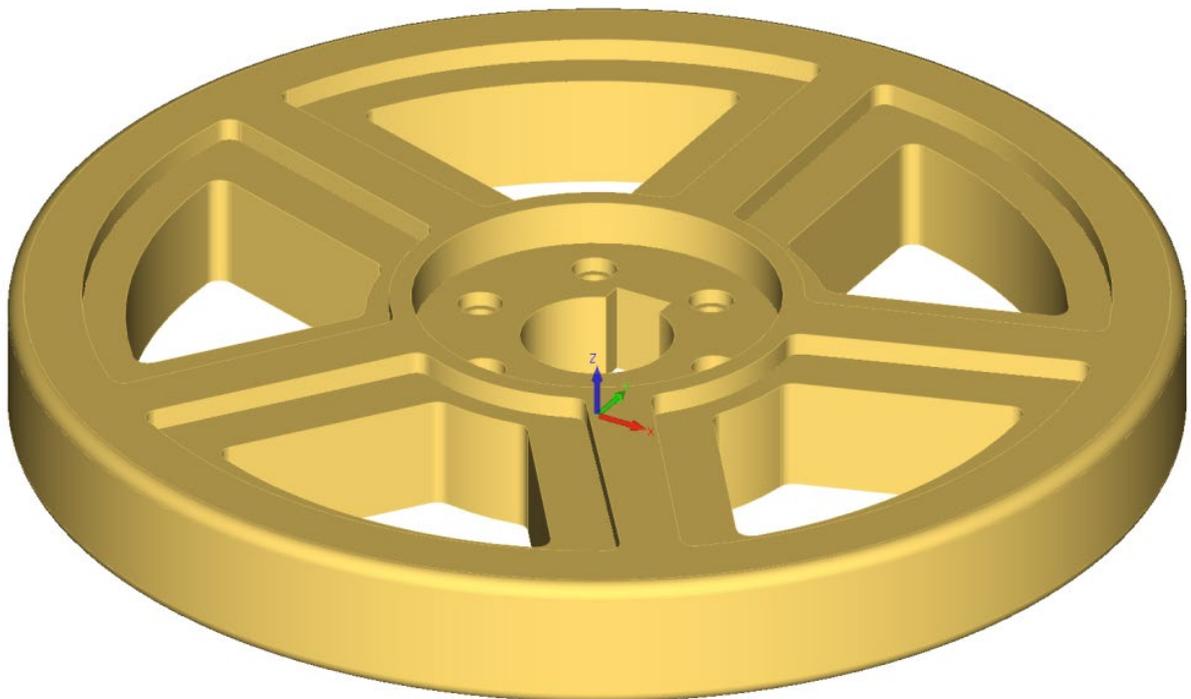
#18 Drilled Block



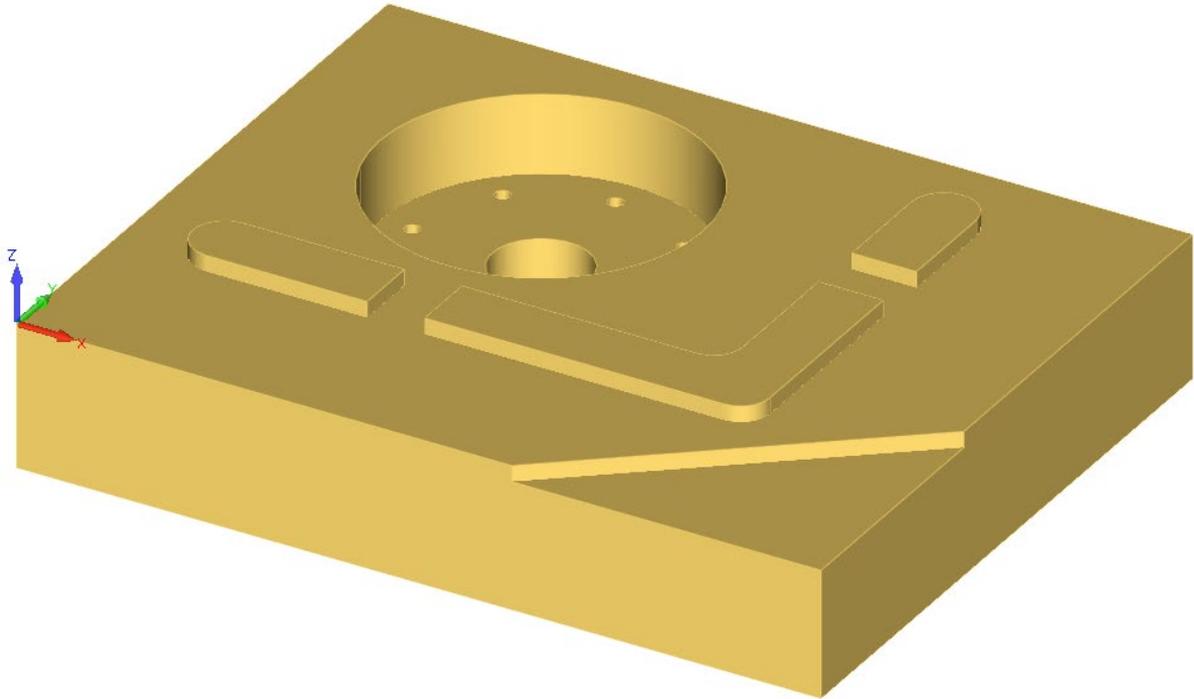
#19 Housing Mount



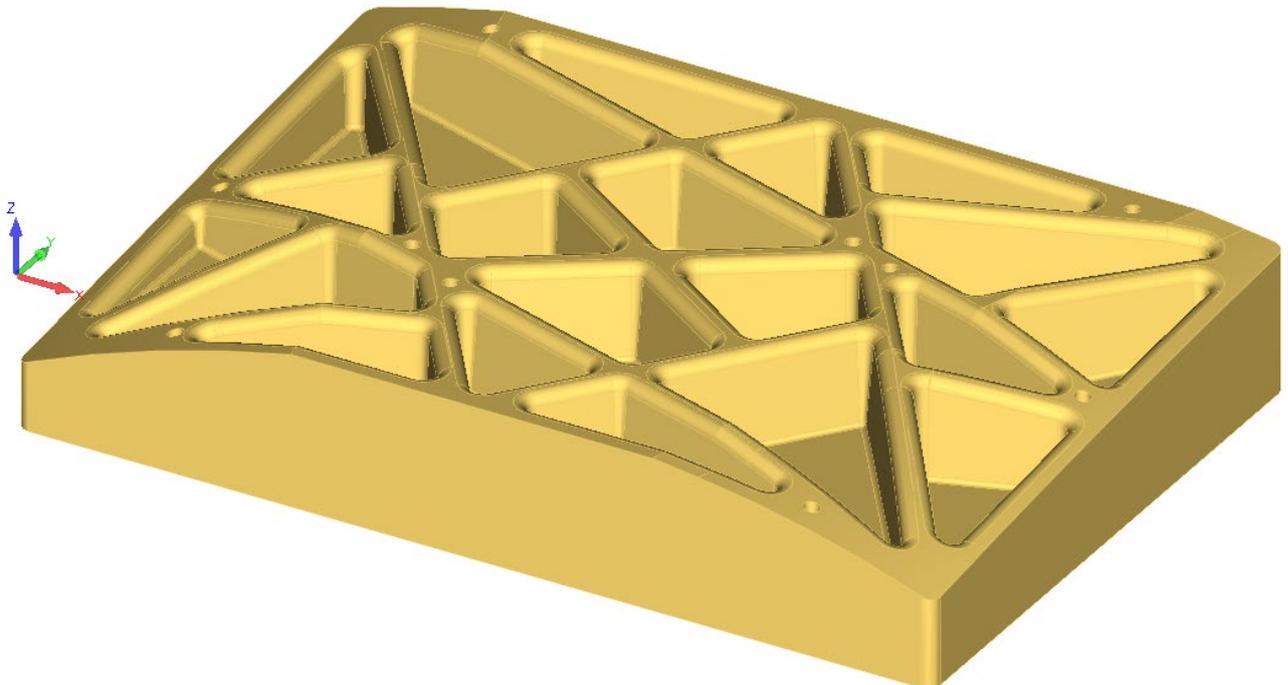
#20 Wheel



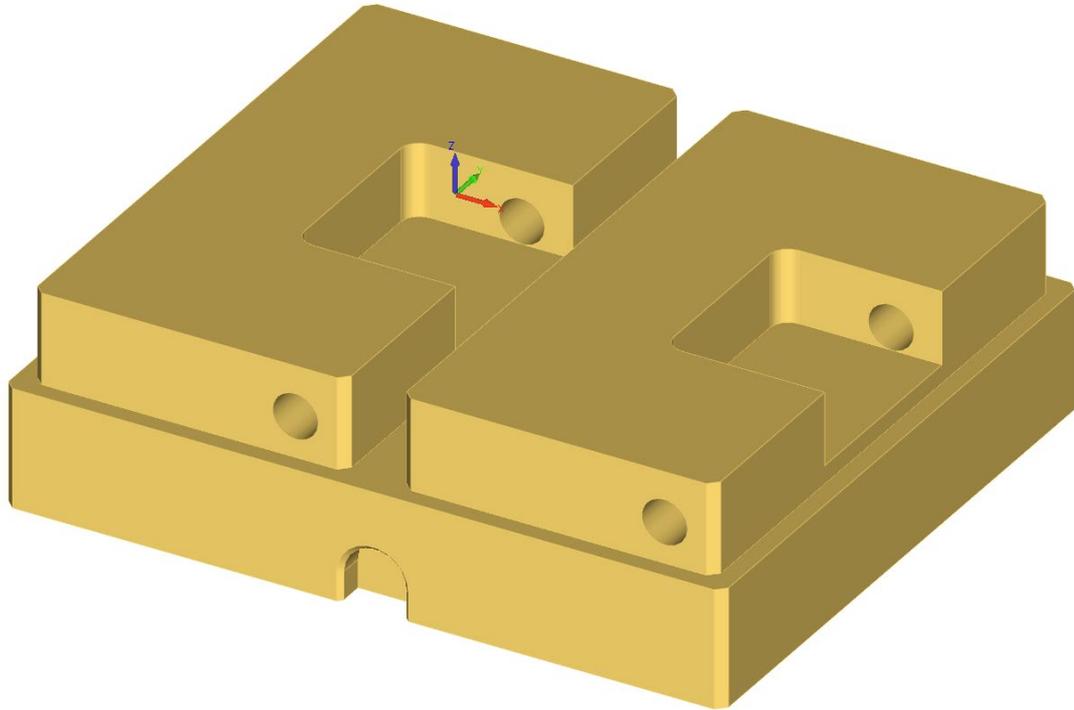
#21 Alignment Block



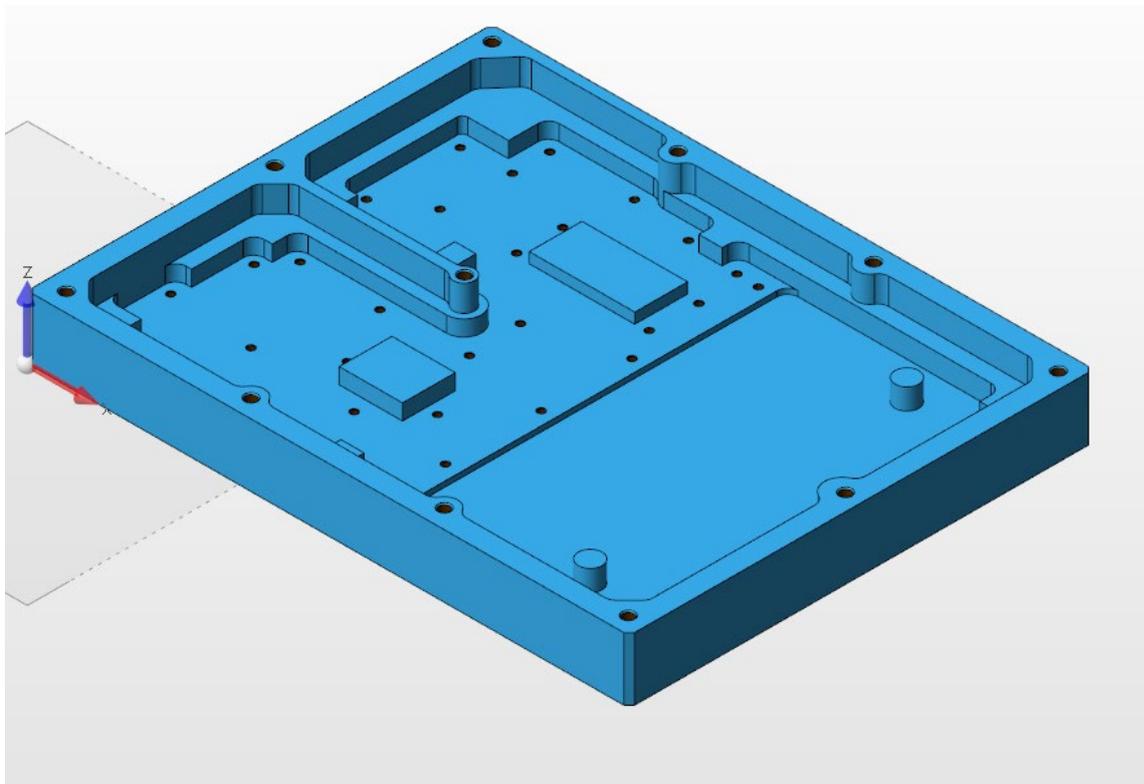
#22 Pocketed Plate



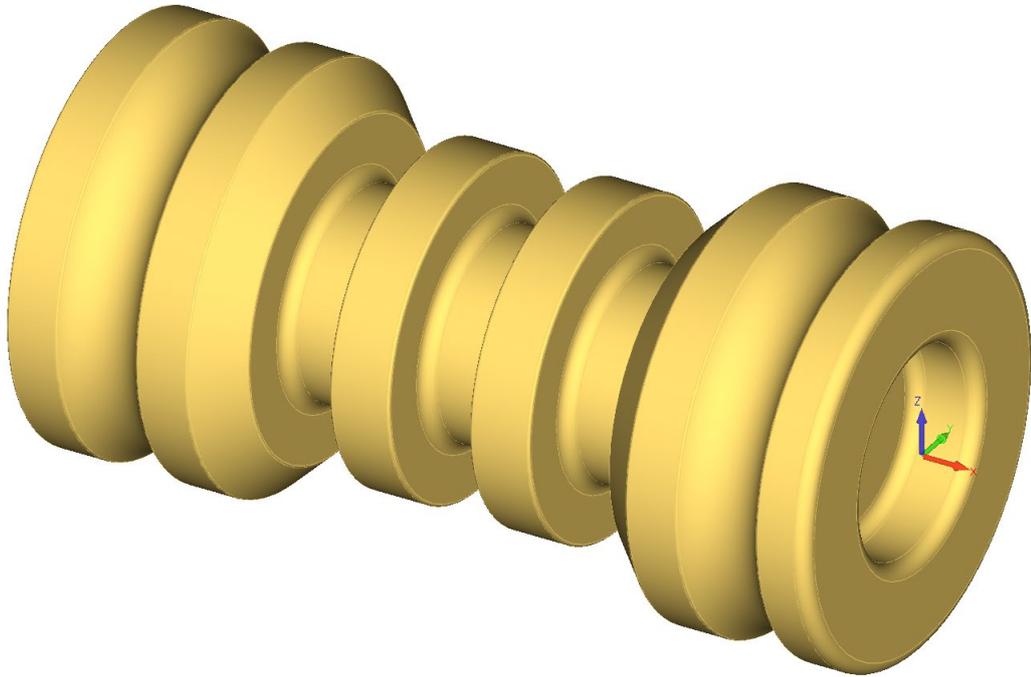
#23 4th Axis Block



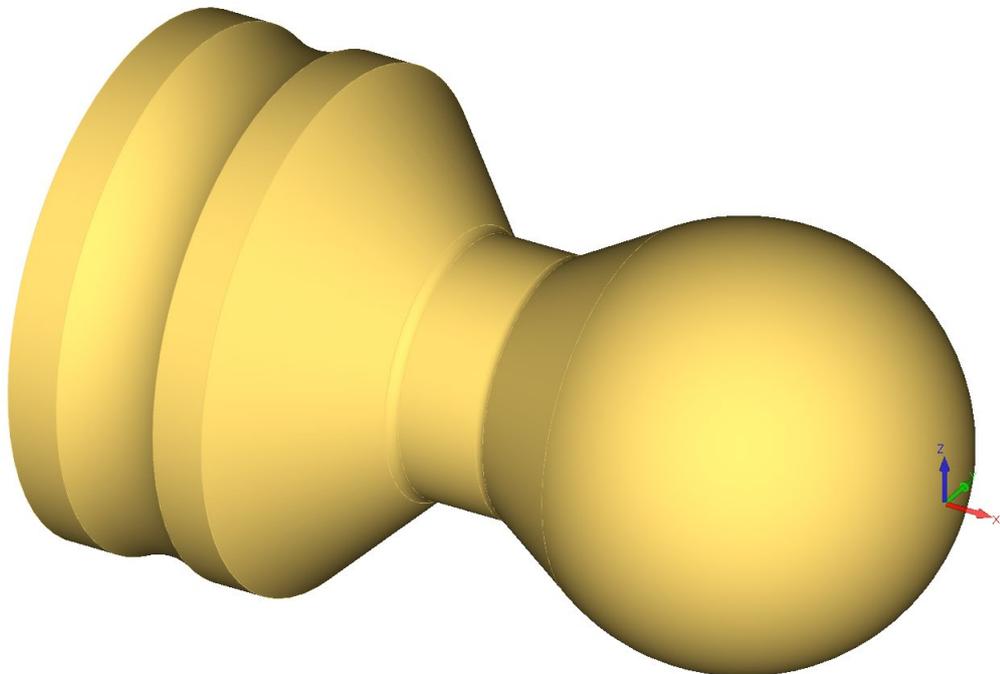
#24 Circuit Board Housing



#25 Castle Chess Piece



#26 Pawn Chess Piece



“...ALP TRG 114 Feature Extraction 2020\Examples\Inventor Cabinet Assembly”

#27 Mobile Ped Drawer.iam

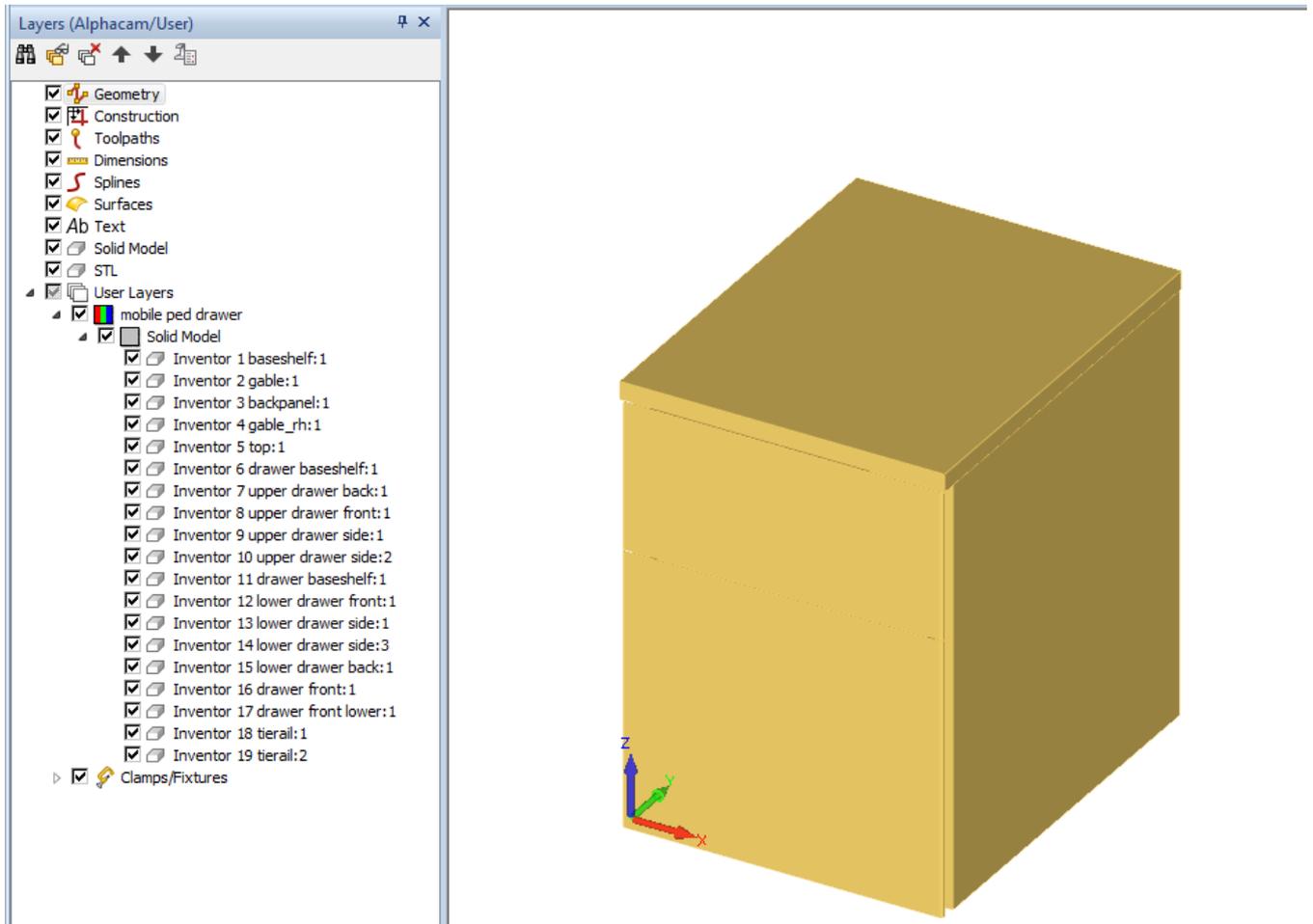


Figure 236 - Mobile PED solid model

You will find the Assemblies tab on the Input CAD command very useful with this particular model.

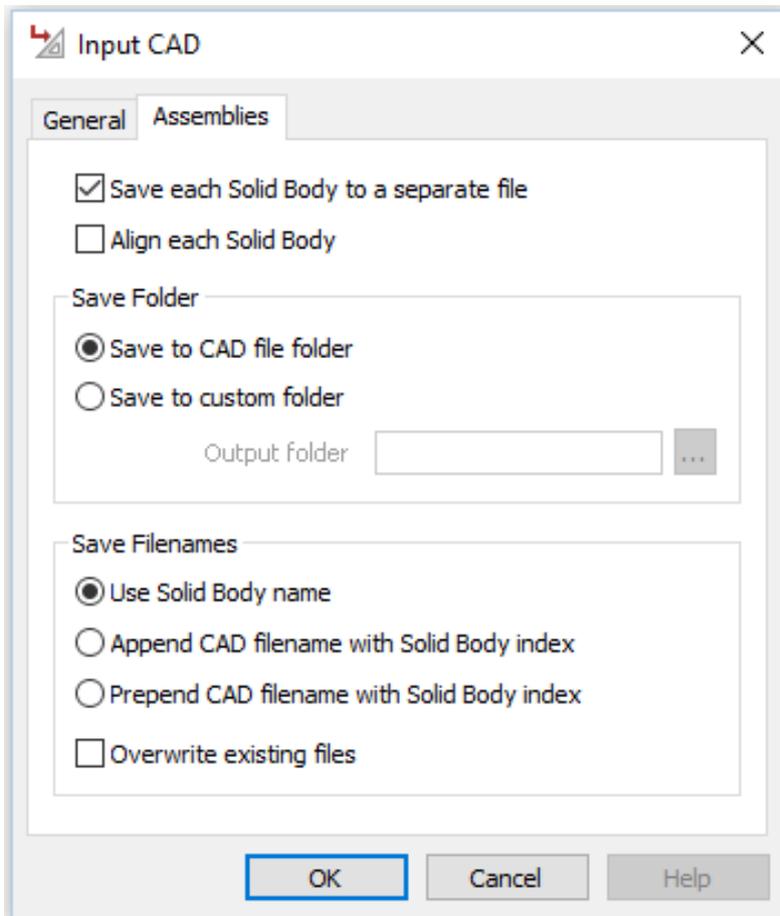
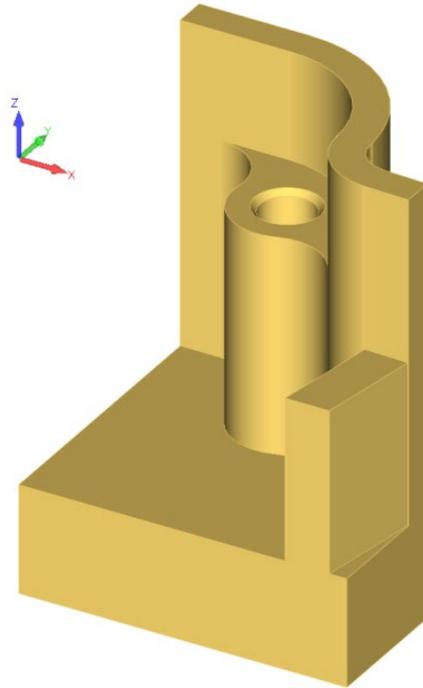


Figure 237 - Assemblies options on Input CAD

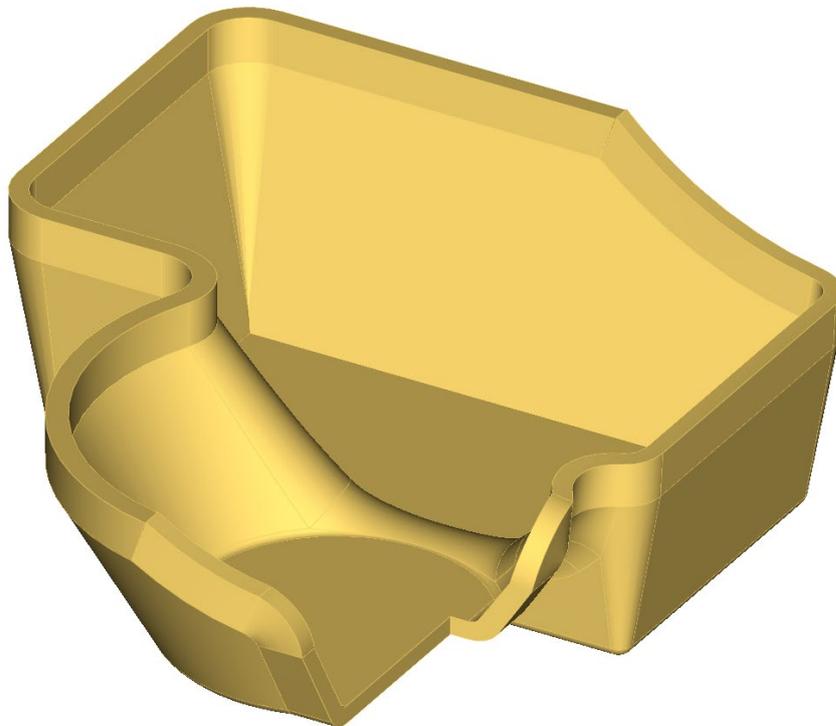
This will save each solid part in the assembly to a separate **Alphacam drawing** and auto align it in the folder you specify depending on your option choices.

 For this part there is also an ALPHACAM Designer version.

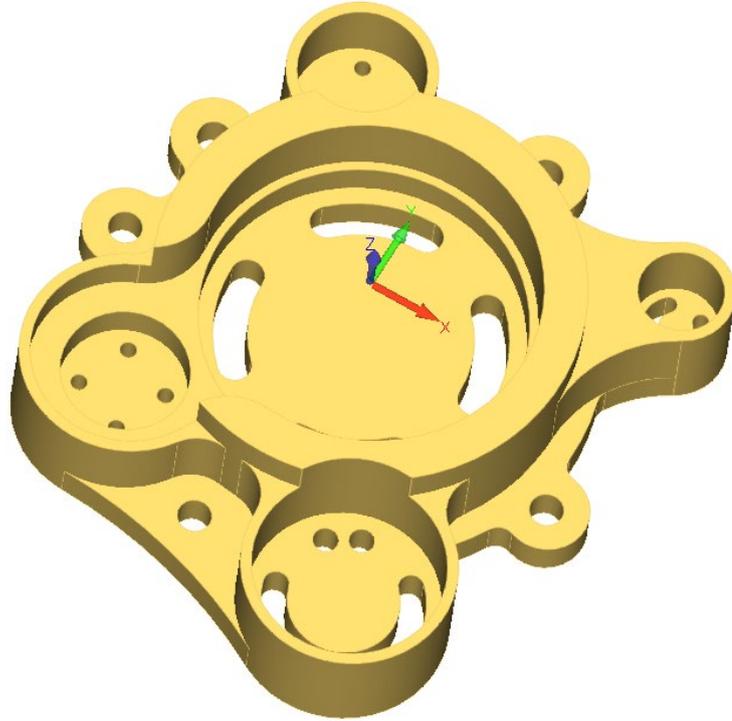
#28 Small Part Requires Work



#29 Poor Trim Part



#30 Pump Top





Version amendments

V	Amendment Description	A	Software Version	Amended Date
15	Minor text formatting alterations.	1	2020.1	04/10/2019
15	Template altered to Hexagon branding	0	2020.0	20/02/2019



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