Delcam Advanced Manufacturing Solutions

FeatureCAM 2012

Training Course

FeatureCAM 2012

Training Course

FeatureMILL 3D 5axis Positioning

FeatureCAM

Copyright © 1995-2011 Delcam plc. All rights reserved.

Delcam plc has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of software.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence.

Delcam plc grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

FeatureCAM Version: 2012 Date: 03 August 2011 11:10

Contents

FeatureCAM overview 7

Finishing Basics 9

Leave Allowance 9
Tolerance.... 10
Stepover 11

Parallel Finish Machining 12

Overview.... 12

3D Spiral & Z Level Finishing 18

Introduction....18
Z Level machining 19
3D Spiral machining 25
Interleaved Z Level machining....30
Horizontal and Vertical machining 31

Corner & Pencil Finishing 36

Introduction.... 36
Pencil Finishing 37
Corner Along Finishing.... 42
Corner Across Finishing.... 48
Combo Along and Across Corner Finishing 49
Corner Multi Pencil Finishing 51

Isoline Finish Machining 52

Introduction.... 52

Radial and 2D Spiral Finishing 65

Introduction.... 65
Radial Finish Machining 65
2D Spiral Finishing 72

Flowline Finish Machining 77

Introduction....77
Flowline Example 77
Undercut Flowline Machining 83

Between 2 Curves Finishing 90

Introduction 90
Between 2 Curves Finish Machining 90

3-Axis Swarf Finishing 94

Introduction 94
3-Axis Swarf Finishing 94
3-Axis Swarf – Multiple passes 103

4-Axis Rotary Finish Machining 107

Overview 107
4-Axis Finishing 107

Wall tolerance for Z-Level 116

Overview 116

FeatureCAM overview

FeatureCAM is a CAD/CAM software suite that automates machining and minimizes programming times for parts on mills, lathes, and wire EDM. Unlike operations-based CAM systems, FeatureCAM generates toolpaths based on the features of the part, and automatically selects appropriate tools, determines roughing and finishing passes, and calculates feeds and speeds. The selections made can be based on the built-in machining knowledge that Delcam supplies 'out-of-the-box' with FeatureCAM, or from experience captured from your company, project or individual users' preferences.

FeatureCAM includes five stand-alone modules:

- **2.5D Milling** - 2.5D design and toolpath generation for 2- and 3-axis mills.
- **3D Milling** - 3D surface modeling and 3-axis toolpath generation.
- **3D Lite** - a limited version of **3D milling**.

  ***3D Lite** lets you mill only one surface per feature, but you can create multiple features. The strategies available in **3D Lite** are Z-level rough, Parallel rough, Parallel finish, Isoline, and 2D spiral.*
- **Turning** - 2-axis design and toolpath generation for 2-axis lathes.
- **Turn/Mill** - Supports lathes with C and Y-axis milling capabilities.
- **Wire EDM** - 2- and 4-axis wire EDM toolpath creation.

The following add-on modules are also available:

- **RECOGNITION** - 3D surface and solid import and the recognition of 2.5D features from solid models. Accelerates making 2.5D and turned parts from solid models.
- **Tombstone** - Multiple part manufacturing for horizontal or vertical milling machines with indexers.

- **Solid Modeling** - Solid modeling and tools for creating molds from solid models.
- **5-Axis Positioning** - Manufacture 2.5D features from 5-axis orientations.
- **Native Import Modules** - Native data can be read directly from SolidWorks, SolidWorks Assemblies, Autodesk Inventor, SolidEdge, Catia, NX, Pro-Engineer, and Step files.
- **Machine Simulation** - Modeling and simulation of a CNC machine.
- **Advanced Turn/Mill (MTT)** - Includes support for Turn/Mill in addition to support for B-axis (5-axis positioning) and multiple turret synchronization.
- **Network Database and Licensing** - Flexible product licensing allows sharing FeatureCAM licenses across a network.
- **5-Axis Simultaneous** - Manufacture 3D features while changing the tool axis.

Why creating toolpaths is so fast

FeatureCAM has the unique ability to generate toolpaths and create NC code to run the machines with a minimum amount of user input.

Traditional CAM systems are *operations-based* and require you to program every operation, one at a time, to create your part. FeatureCAM is *feature-based*; this means the part is created using features that describe that part, from simple holes, to complex pockets, to turned grooves. Machinable features contain information and rules describing how and where material removal should occur, cutting depths, whether to use climb cutting, whether to spot drill or center drill, and preferred machining strategies for roughing and finishing. This means that after you import or draw the part and identify its features, FeatureCAM automatically:

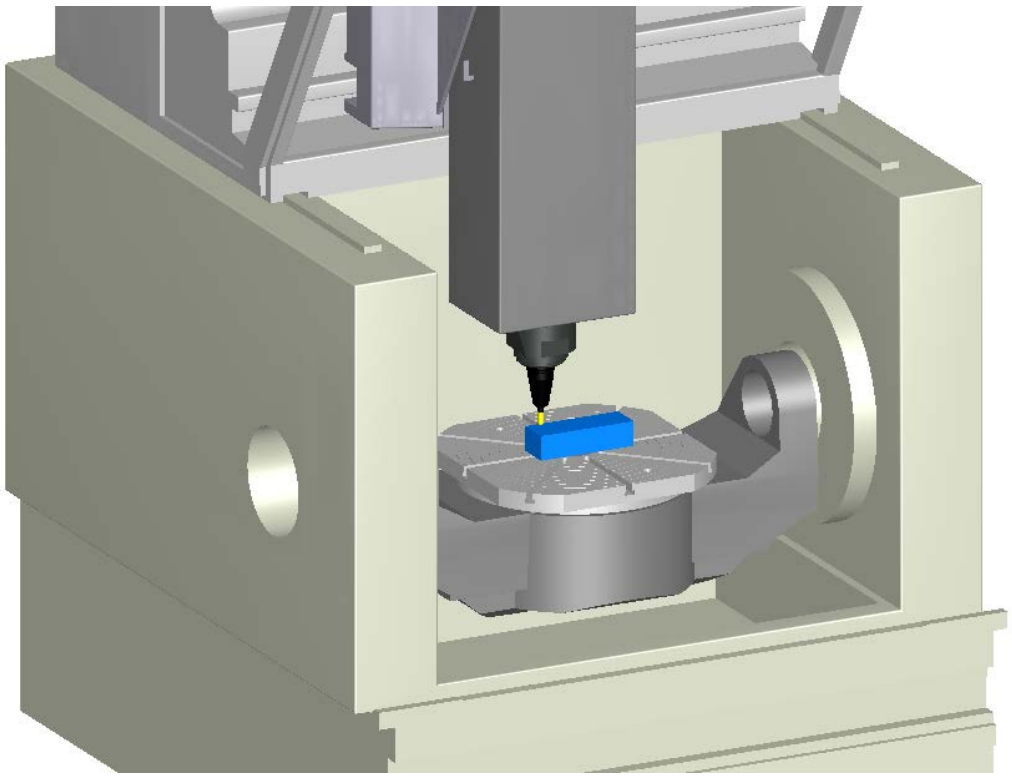
- Selects the most appropriate tools and operations;
- Recommends machining strategies;
- Calculates speeds and feeds;
- Generates toolpaths and creates the NC code.

You can customize this built-in 'intelligence' to your own style of cutting.

5-Axis Positioning

Introduction.

This module will provide the user with an introduction to 5-axis positional machining in FeatureCAM. In normal, 3-Axis machining, the machine tool has three linear axes. In 5-axis positional machining, two additional rotational axes are added. This is often referred to as 3+2 machining. The 3+2 refers to the three linear axes which are controlled simultaneously, and the two rotary axes which can move the part to a new position and then stop. Once the part is in the new position, 3-Axis simultaneous machining is carried out. This type of machining can only move the linear axes or the rotary axes - not both at the same time, i.e. positioning is discrete from machining.



The image above shows a typical 5-Axis machine. In this case, the 3 linear axes (X, Y & Z) are above the machine, and the two rotary axes are on the trunnion and rotary table.

The use of 5-Axis Positioning gives a number of benefits:

A part can be cut in a single setup including undercuts which would normally require the part to be dismounted from the machine and set up in a different orientation. This reduces the setup time and errors that can be introduced by having to do multiple setups.

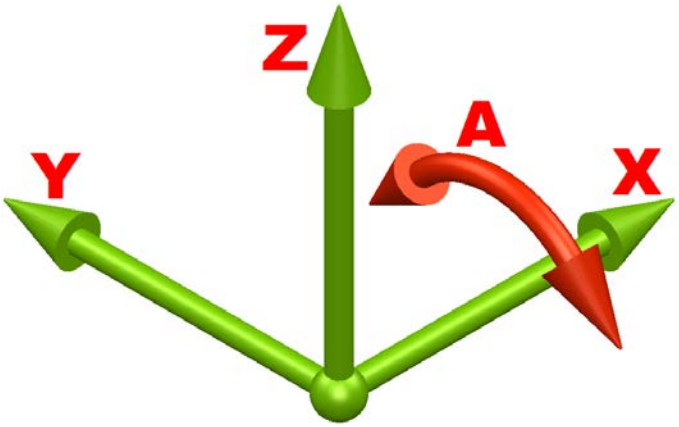
By tilting the tool relative to the surfaces being machined, the contact point of the tool on the surface can be changed giving better cutting conditions.

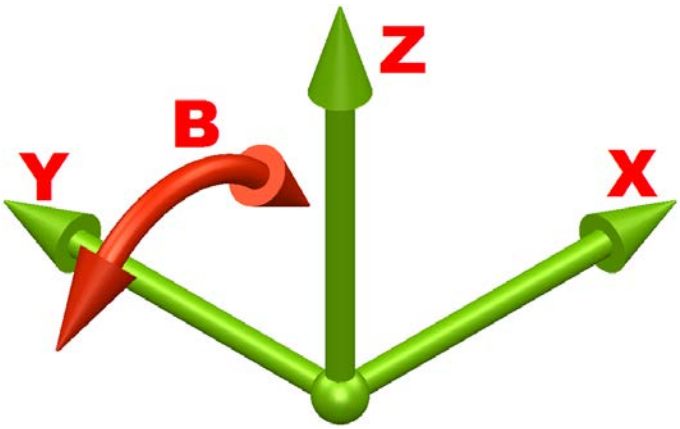
Using shorter tools increases the rigidity of the tool reducing deflection and chatter giving improved surface finish and accuracy.

Axis Conventions

A 3-Axis machine has just the three principal linear axes, X, Y and Z. A 5-Axis machine will have two more rotary axes; in most cases these axes will rotate around one of the principal axes. The naming of these axes differs from manufacturer to manufacturer, but in most cases the convention is:

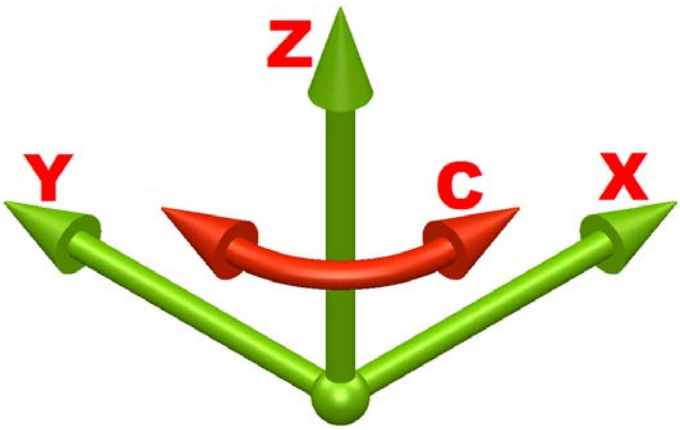
The A axis rotates around the X.





The B axis rotates around the Y.

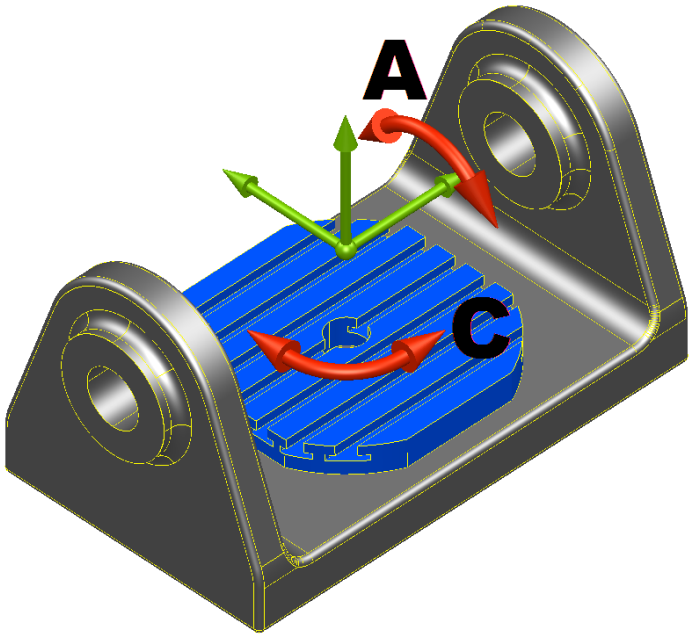
The C axis rotates around the Z.

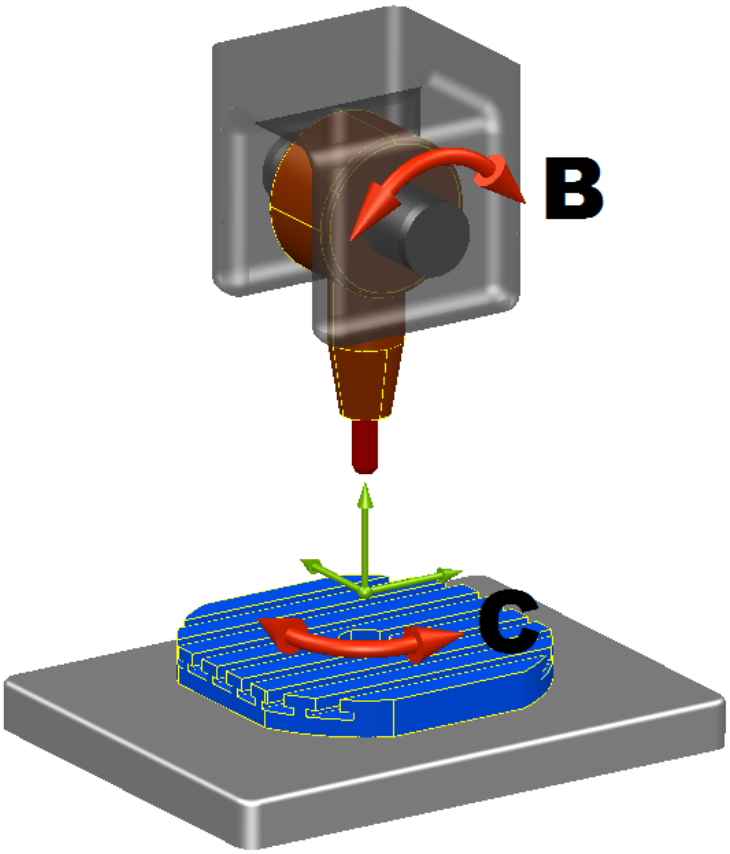


Machine Configurations

A 5-Axis machine will have the X, Y and Z axes plus two of the above rotary axes. The machine shown below has an A axis motion where the trunnion table rotates about X axis. It then has a C axis rotary table mounted on the trunnion which rotates around Z. Please note that the Z axis here refers to the axis of rotation of the C rotary table when the trunnion is in its home position.

This is one of the most common machine configurations. It is called a Table/Table configuration as the two rotary motions are both positioning the workpiece rather than moving the tool. The C axis is on top of the A so this would be called a Table/Table machine with C stacked on A.

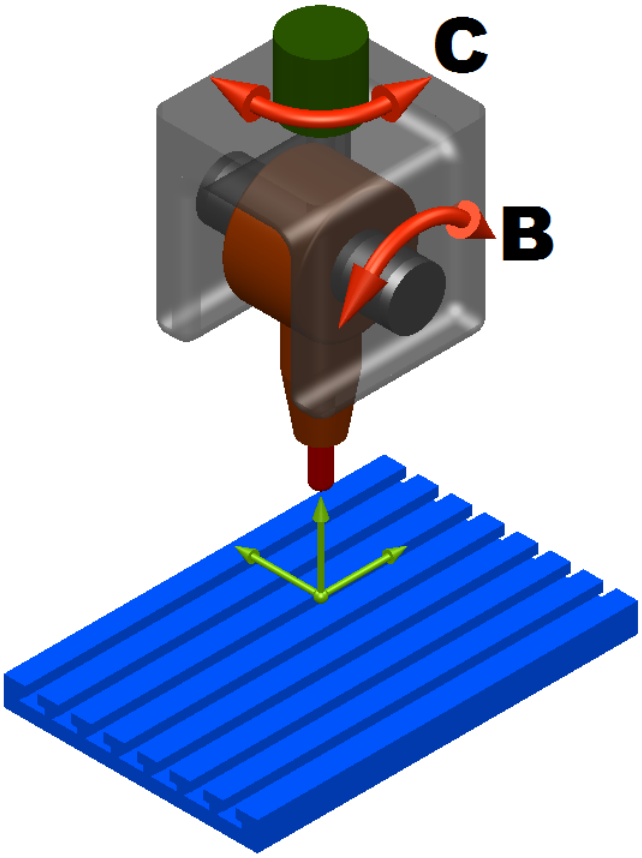




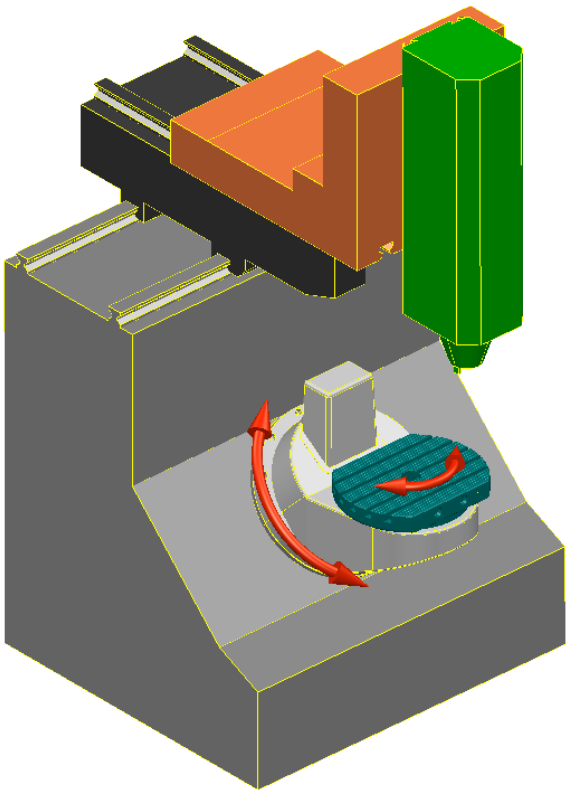
The machine shown to the left moves the tool to a different rotational position by rotating the head of the machine about the Y axis this is a B rotation. The part and table are rotated around the Z axis, i.e. a C rotation.

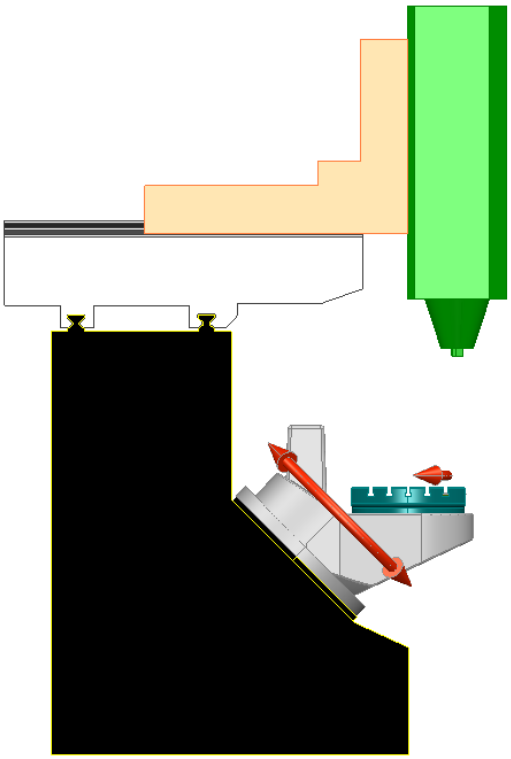
This is another common configuration and is known as a B on C Head/Table machine.

In this machine both of the rotary motions are on the head. The upper part of the head can rotate around Z and the lower part around the Y. This is a B & C Head/Head machine configuration.



The final machine configuration below is an Inclined Table/Table machine which has a C axis rotary table mounted on another table which is inclined at 45 degrees. In this case the lower rotary table is designated as the B axis even though it is not rotating around the Y axis. This machine configuration is becoming much more popular in recent years as it offers a large working envelope with a small footprint.





The output of the post processor which is sent to the machine can take a number of different forms; the most common will output the rotation position explicitly as an A and C angle pair (or A&B or B&C depending on the machine configuration). For example:

N55 G1 X-0.3398 Y1.3561 Z3.0245 A-29.206 B-116.565 S7500 M03

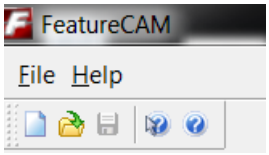
Other machine controllers require the rotation position as a direction vector of the form I, J & K where the three letters give the components of the vector in the X, Y & Z directions respectively. The necessary calculations are all carried out by the post processor. Once the post has been configured for a particular machine the customer will just program FeatureCAM in the normal way.

Turning on 5-Axis

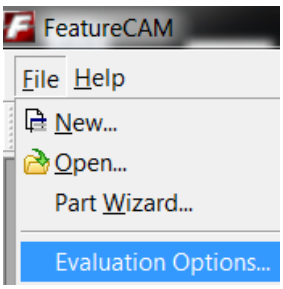
By default in FeatureCAM the tool is oriented along the setup Z axis. In order to use 5-Axis positioning, we first need to enable 5-Axis simultaneous in the Evaluation options. Once this is enabled, we also need to change the stock properties to allow 5-axis indexing.

- Open FeatureCAM but **DO NOT open an FM document**

On the main toolbar you should only have two options, **File** and **Help**.



- Click on **File** and then select **Evaluation Options**



- Check **5 Axis Positioning**

FeatureCAM will give a warning that the product component has not been licensed. This just means that it has not yet checked the license file and dongle to see if you have the rights to use 5 Axis Simultaneous positioning.

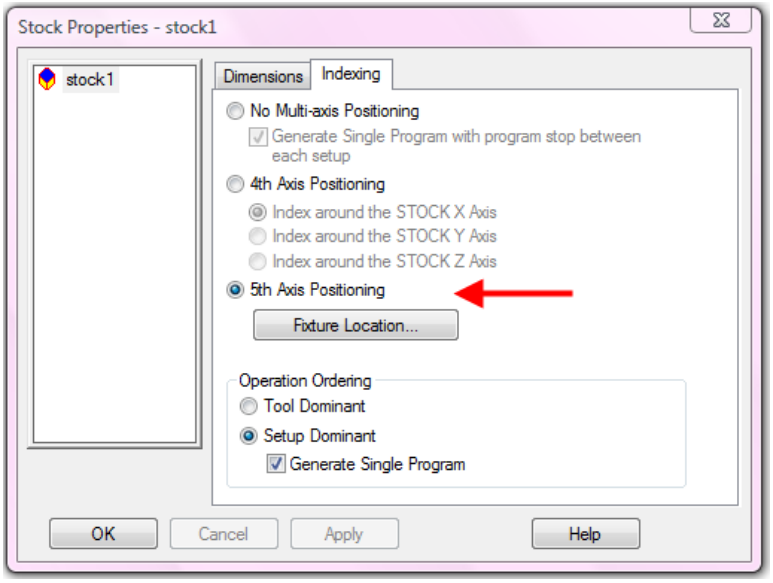
- Click **Apply**

If you have the correct options in your license file, you will then see the word **Licensed** next to the **5 Axis Positioning option**. If you do not see this, then your license file is not set up to license this option on the dongle you are using. In this case you can either continue to use FeatureCAM in Evaluation mode (this does not allow you to save), or you can contact your dealer to arrange a license.

- Click **OK** to close the Evaluation Options form

FeatureCAM now has the 5-Axis Simultaneous option enabled. In order to create a 5-Axis program, we also need to set up 5-Axis indexing in the stock properties.

- Create a new **Inch Milling document**
- On the **Dimensions** form click **Finish**
- Select the **Indexing tab**
- Check **5th Axis Positioning**



- Click **Apply** and then **OK**

The new document is now ready for programming of a 5-Axis part. There are a number of other things that need to be set up in order to position the part relative to the axes of rotation of the machine so that the program will produce the part correctly. This is rather more involved that the setup for a 3-Axis part and differs depending upon the machine configuration and includes some variables which are machine specific. We will look into the machine specific setup variables next.

- Leave the document **open** for the next exercise

Setting Up for 5-Axis Machining

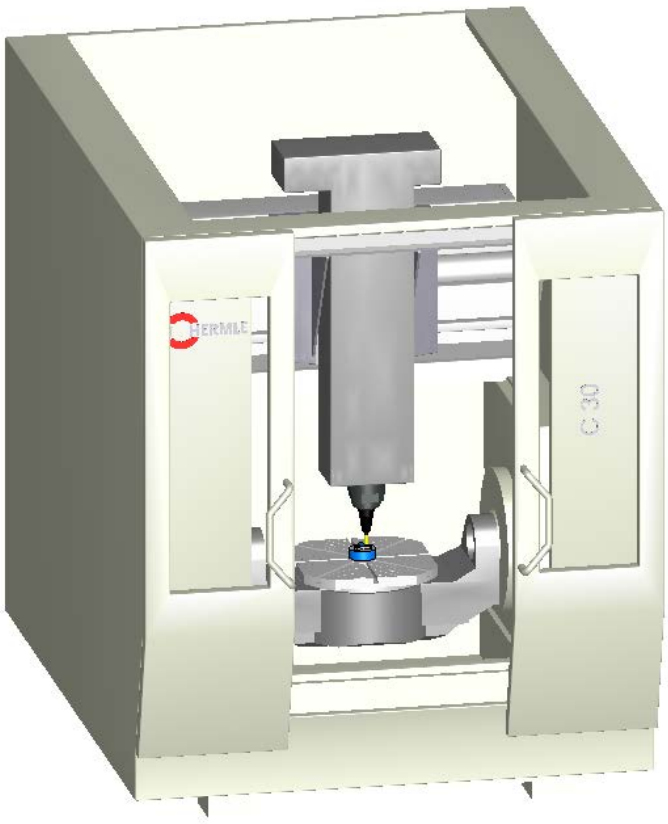
Setting up for 5-Axis machining is more complicated than for 3-Axis machining, and is a two step process.

First, we have to configure the post processor so that it matches the actual machine being used. This is a “one off” process and involves inputting a number of offset values into the post to allow it to carry out the necessary coordinate transforms to accurately position the machine. This involves relating the **pivot point** of the machine to a known **reference point** on the machine. For example, on a table/table machine this would normally be the center of the topmost table. The process will involve a certain amount of testing to fine tune the calibration of the post. Once this has been set up, it should not be touched again unless the machine is physically changed, e.g. after a rebuild of a table component.

It should be noted that the post configuration is machine specific, that is even if a shop has two machines of identical make and model there will be small differences between them and each will require its own calibrated post processor.

The second stage of setting up applies to the particular part being machined and involves relating the relative distances in X, Y and Z **From** the **part setup** origin **To** the **reference point** of the machine. This allows FeatureCAM to work out how the setup and the part will move for any given combination of rotations of the machine axes.

Once these two stages have been completed FeatureCAM will be able to accurately output 5-Axis NC code for the machine. The final piece of the puzzle is to set up the simulation parameters in the part setup. This will have no effect on the code but is important if you want to carry out machine simulations for gouge and collision checking purposes.



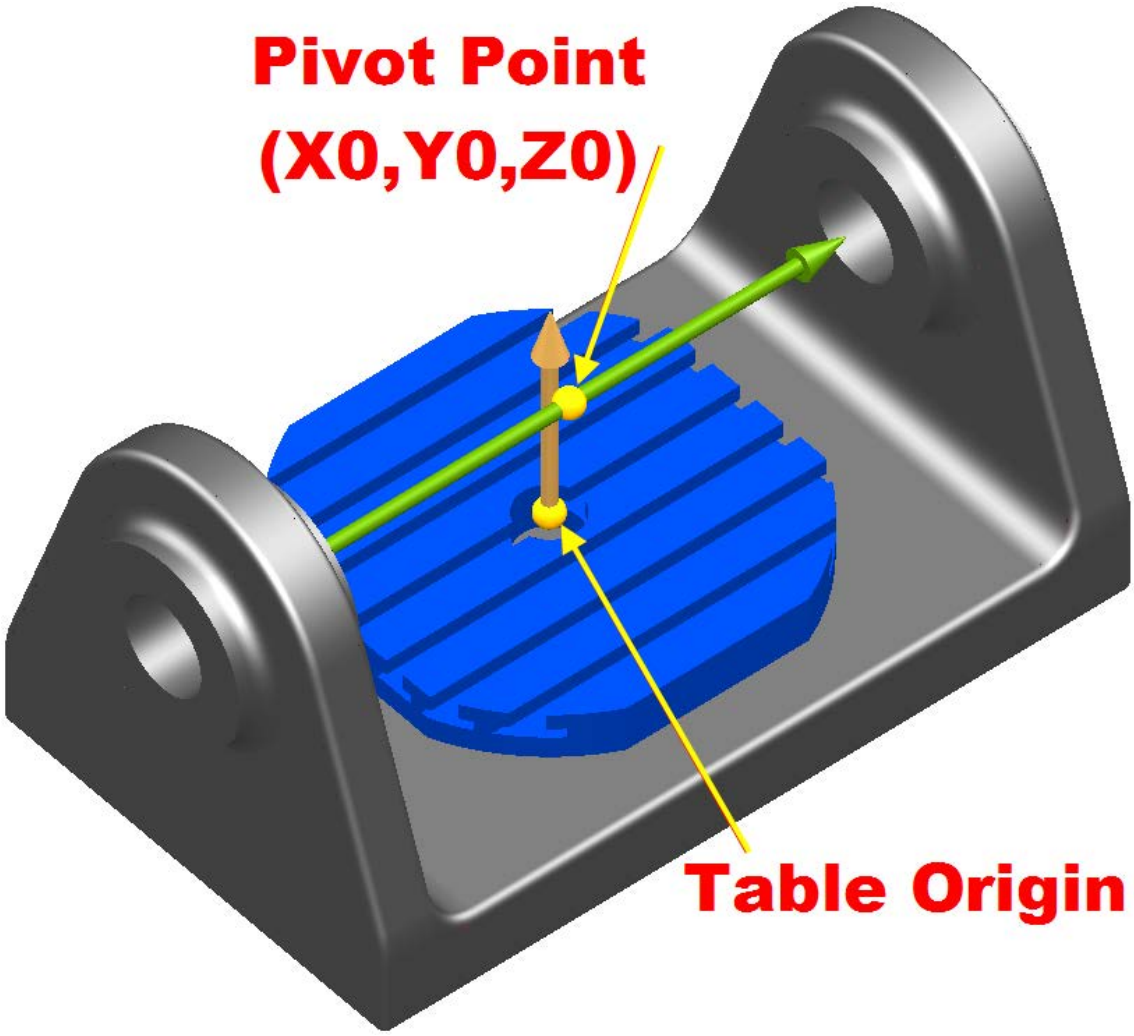
Post Configuration

Before any 5-Axis machining can be carried out, the post processor must be first configured. The configuration process is only done once as it is related to the physical properties of the machine. As long as the rotary axes are not removed from the machine the configuration should remain valid.

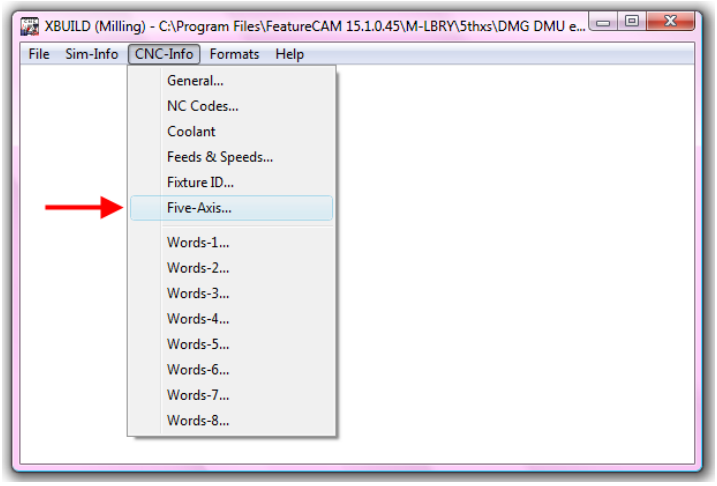
The post processor contains a set of coordinates that relate the Pivot Point of the machine to the Table Origin.

The **Pivot Point** of the machine is the point in space that remains still when either or both of the rotary axes index. The pivot point is the **Program Origin i.e.** G54X0Y0Z0. For a vertical spindle - table/table configuration, X0 is typically at the B/C-axis centerline. Y0 and Z0 are always on the A-axis centerline.

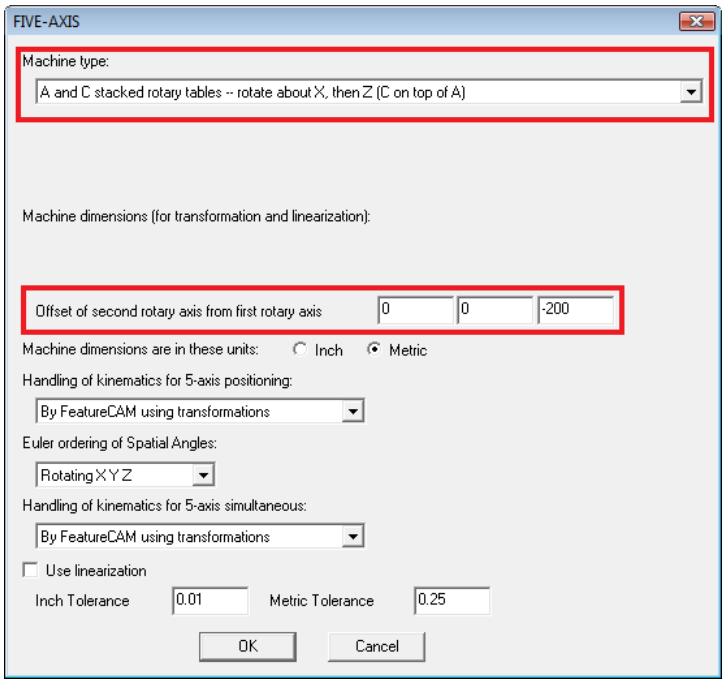
The **Table Origin** is a known location on the machine table. For table/table machines, this is the B/C-axis centerline and typically the face of the B/C table. This location is used by the post processor to calculate transformations whenever there are rotations of the A and/or B/C axis.



- From the **Manufacturing** menu select **Post process**
- Click **Browse** and go to the **Data** folder
- Select the **Mazak Variaxis.cnc** post processor
- Click **Edit**
- Click on **CNC Info** and select **5-Axis**

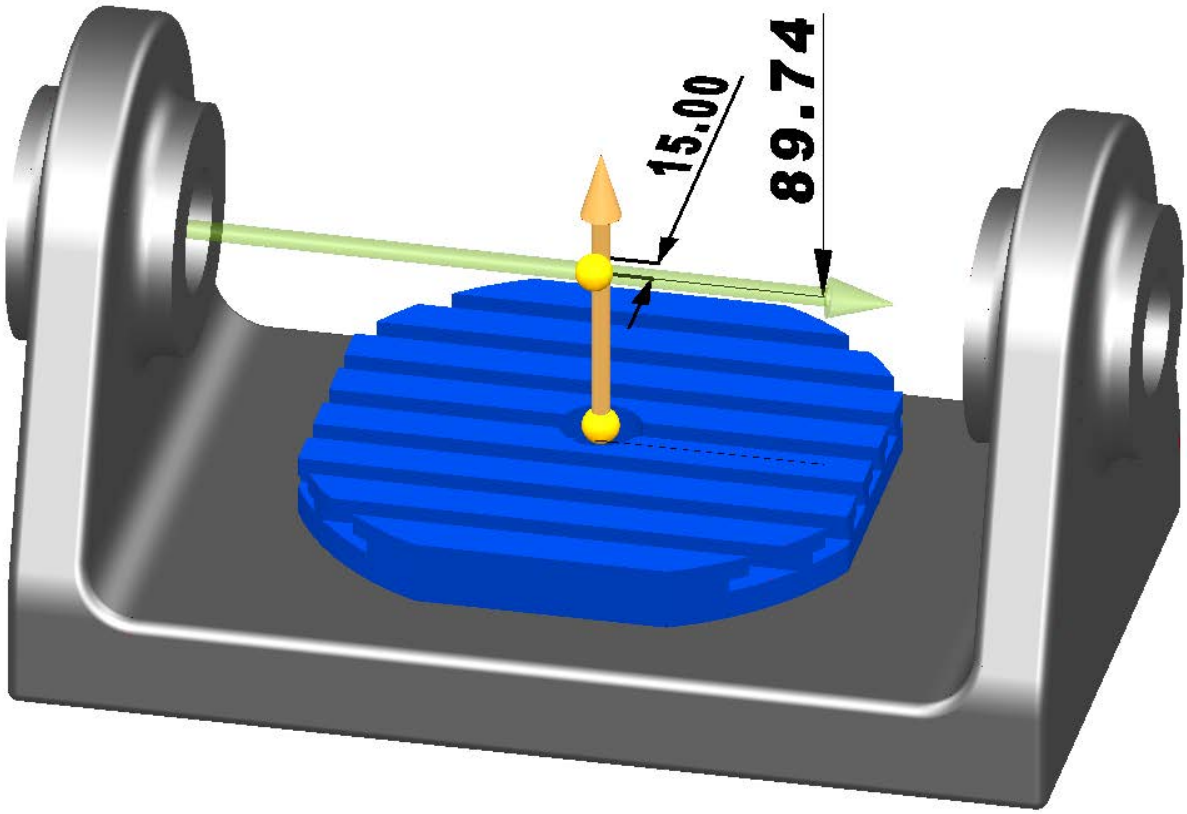


This opens the form where you can configure the post. At the top of the form you will notice that this is a C on A Table/Table machine.



In the center of the form you will see the X, Y and Z offsets. These are the distances **FROM** the **Pivot Point TO** the **Table Origin** when the machine is in its home position. In this case you see a X and Y offset of Zero and a Z offset of -200. This means that when the tables are at **A0** and **C0** the pivot point is vertically above the table origin by a distance of 200mm.

The image below shows the offsets for another Table/Table machine. In this case the table origin is behind and below the pivot point and has the same X location. The offsets that you would enter in this case would be **X 0.00**, **Y -15.00** and **Z -89.74**.



It should be stressed again – Once the postprocessor is configured, do not change these figures or the machine will not cut parts correctly. The process of getting the figures to enter into this form usually involves entering the figures given by the machine tool supplier and then running a series of test cuts to further refine the calibration figures.

- Click **Cancel to close the form without saving any changes**
- **Close XBUILD without saving changes**

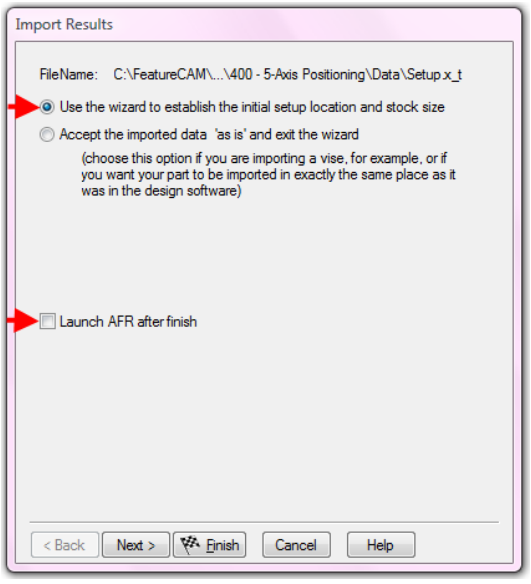
Loading a part into FeatureCAM

Once we have configured the post processor, FeatureCAM knows where the table origin will be relative to the pivot point for any combination of rotary moves. Once this is established, we can proceed to produce part programs.

The first thing we need to consider is the orientation of the part relative to the **Stock Axis**. When we are setting up a 3-Axis part the stock axis is irrelevant, as there are no indexing moves. With a 5-axis part it is vitally important to set the orientation of the part correctly **before** you start programming. The stock axis fixes the angular orientation of the part relative to the axes of the machine in the home position i.e. which direction X and Z are pointing relative to the part when the machine is at A0, C0. It also sets the point around which indexing moves will rotate; ideally it should be somewhere in the center of the part.

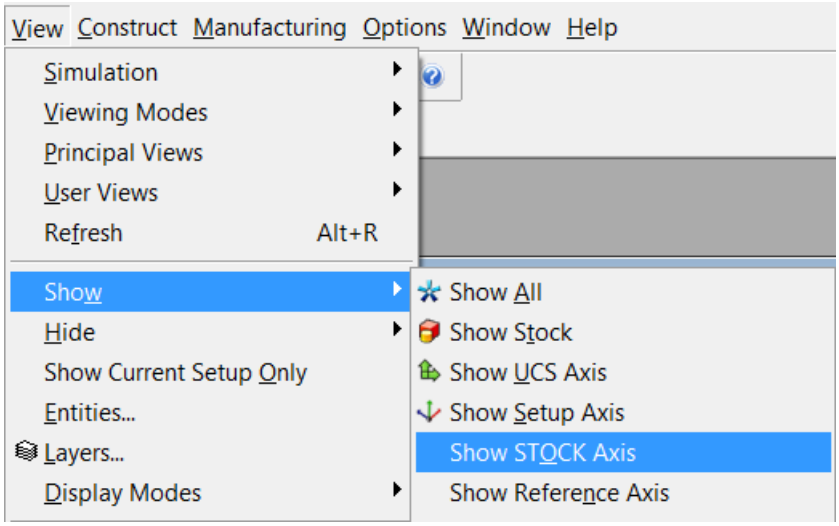
If you use the stock wizard during import, the stock axis should be set up correctly; if you do not use the wizard, you will need to check manually. We will now import a part and set it up.

- Create a new **inch milling document**
- Click Cancel on the stock dimensions form
- Import the file **Setup.x_t**
- Check **Use the wizard**….
- Uncheck **Launch AFR after finish**

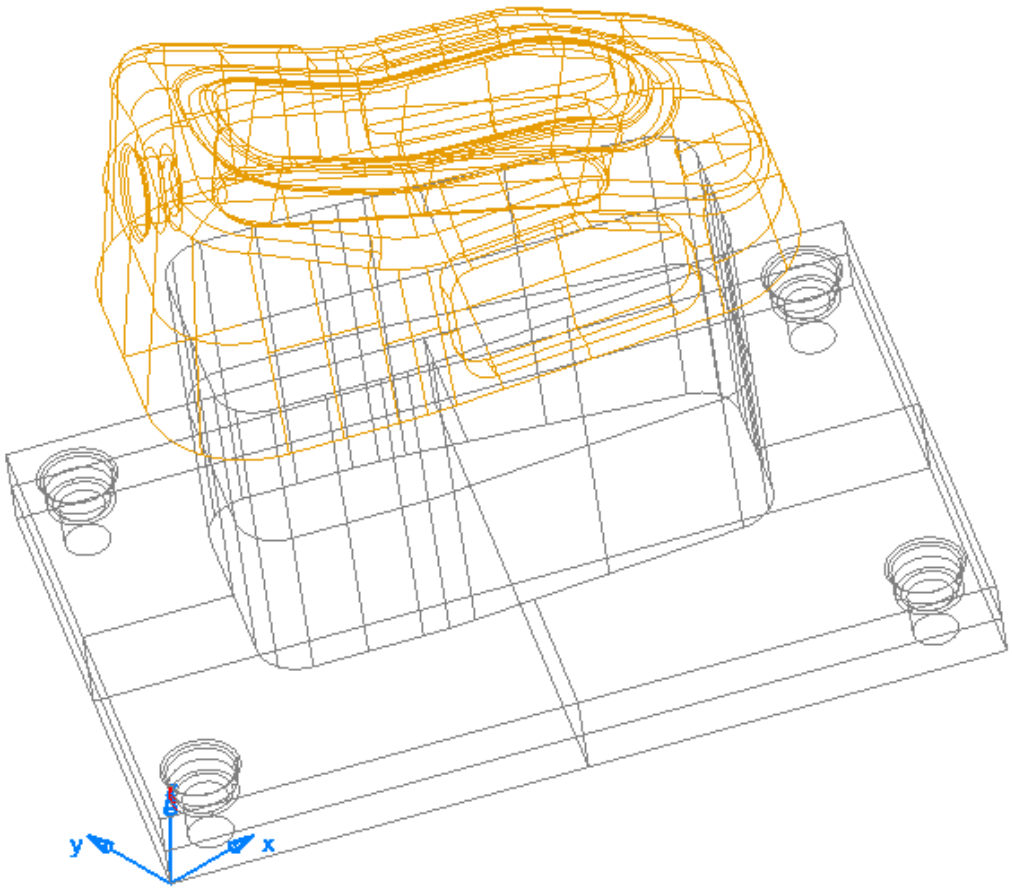


- Select an **Isometric view**

- From the **View menu** select **Show > Stock Axis**

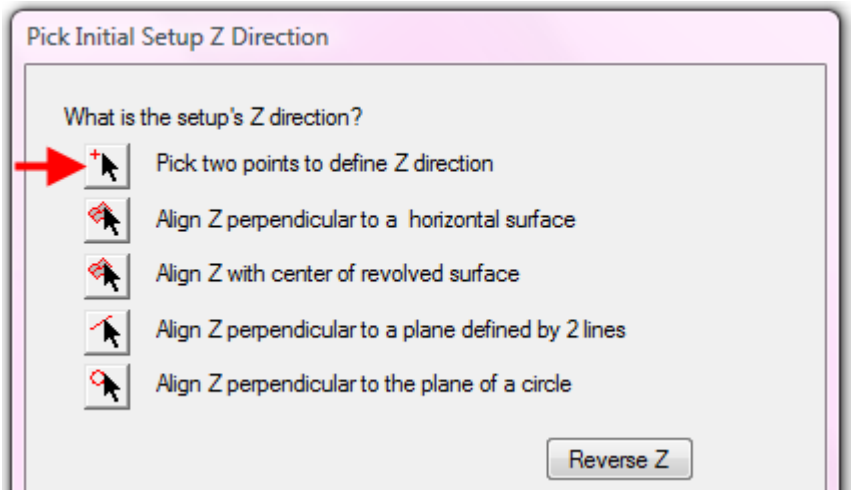


The imported file contains a part on a jig. The stock axis is appearing at the lower left corner of the jig; it looks like a setup but is colored blue. As you can see, the part is not aligned correctly with the stock axis. Even if we were to place a setup aligned with the jig, FeatureCAM would still regard the Stock Axis as being the A0, C0 position on the machine. When we ran the program the first move would be to realign the part so that the setup was no longer aligned with the machine axes.

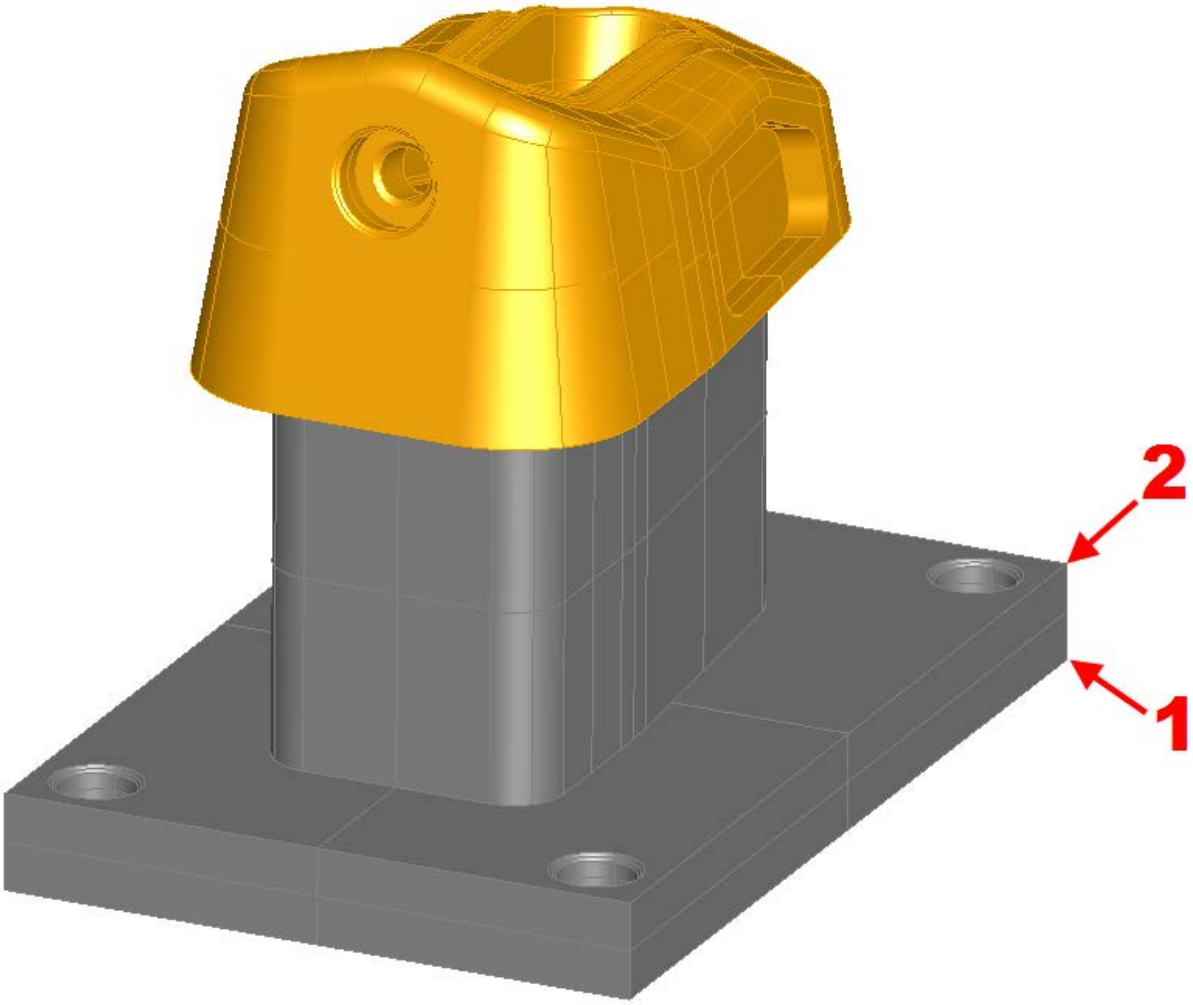


- Click **Next**

- Click **Pick 2 points to define Z direction**

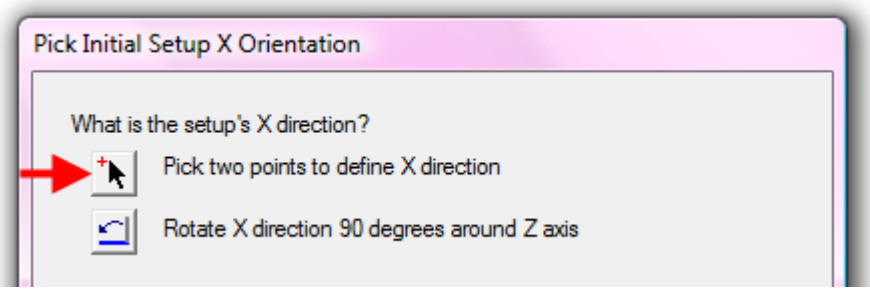


- Select two points up the corner of the jig

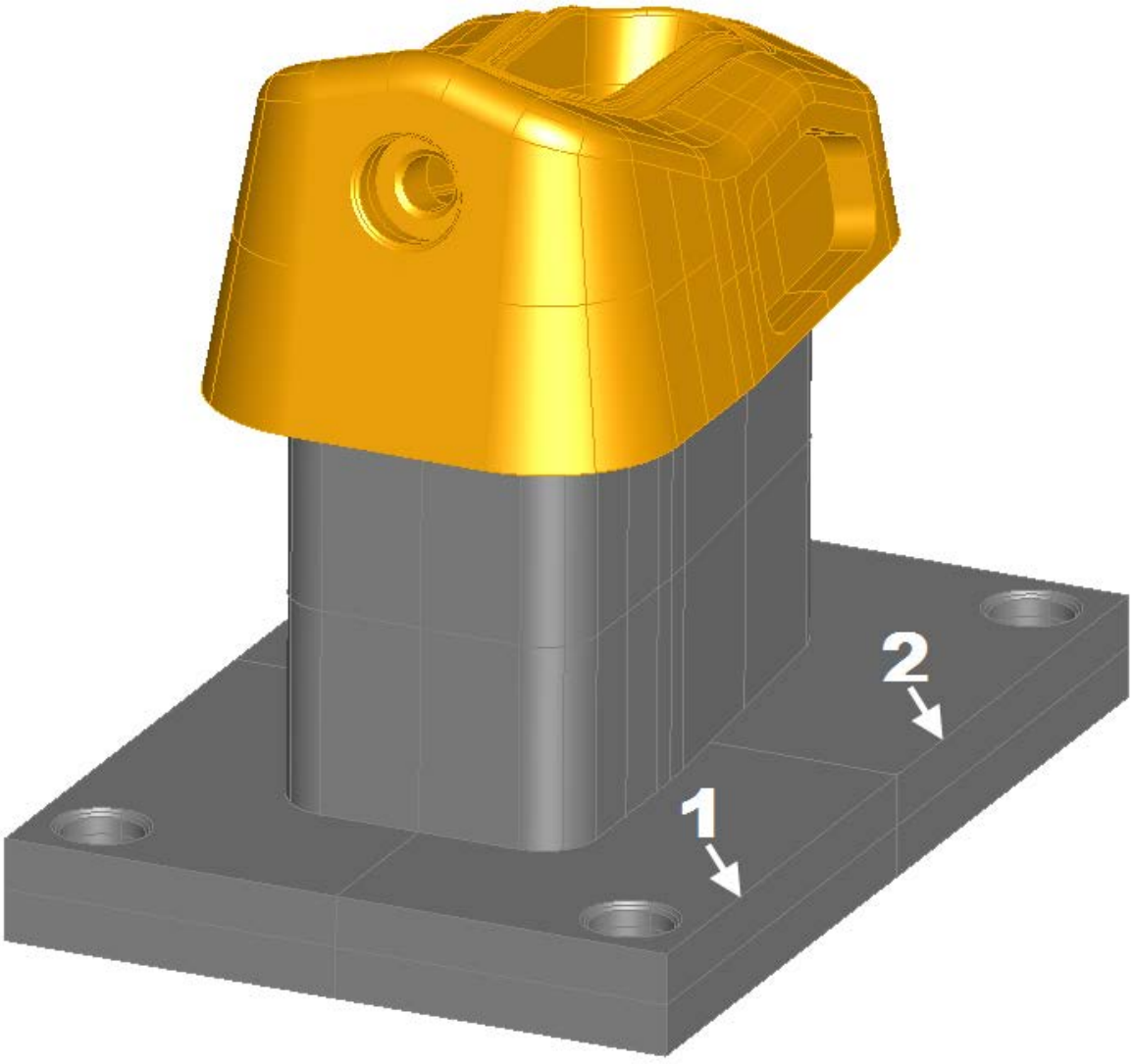


- Click **Next**

- Click **Pick 2 points to define X direction**

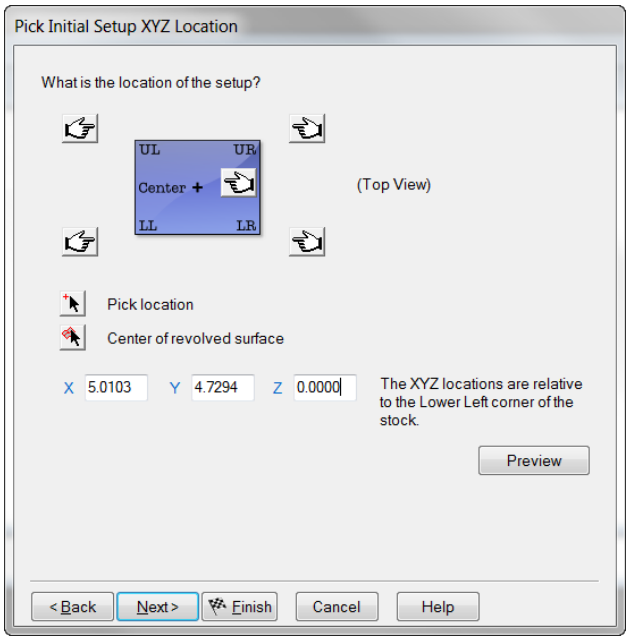


- Select two points on the long edge of the jig

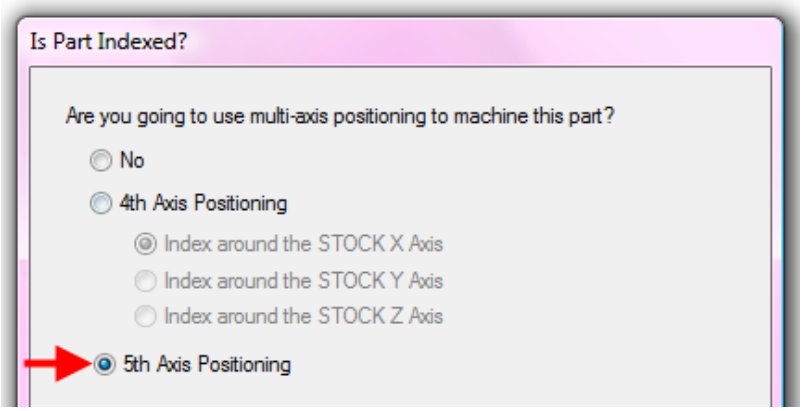


- Click **Next**
- Select a **Block stock** and click **Next**
- Check **Compute stock size from the part**
- Click **Next**

- Place the setup at the center top of the block



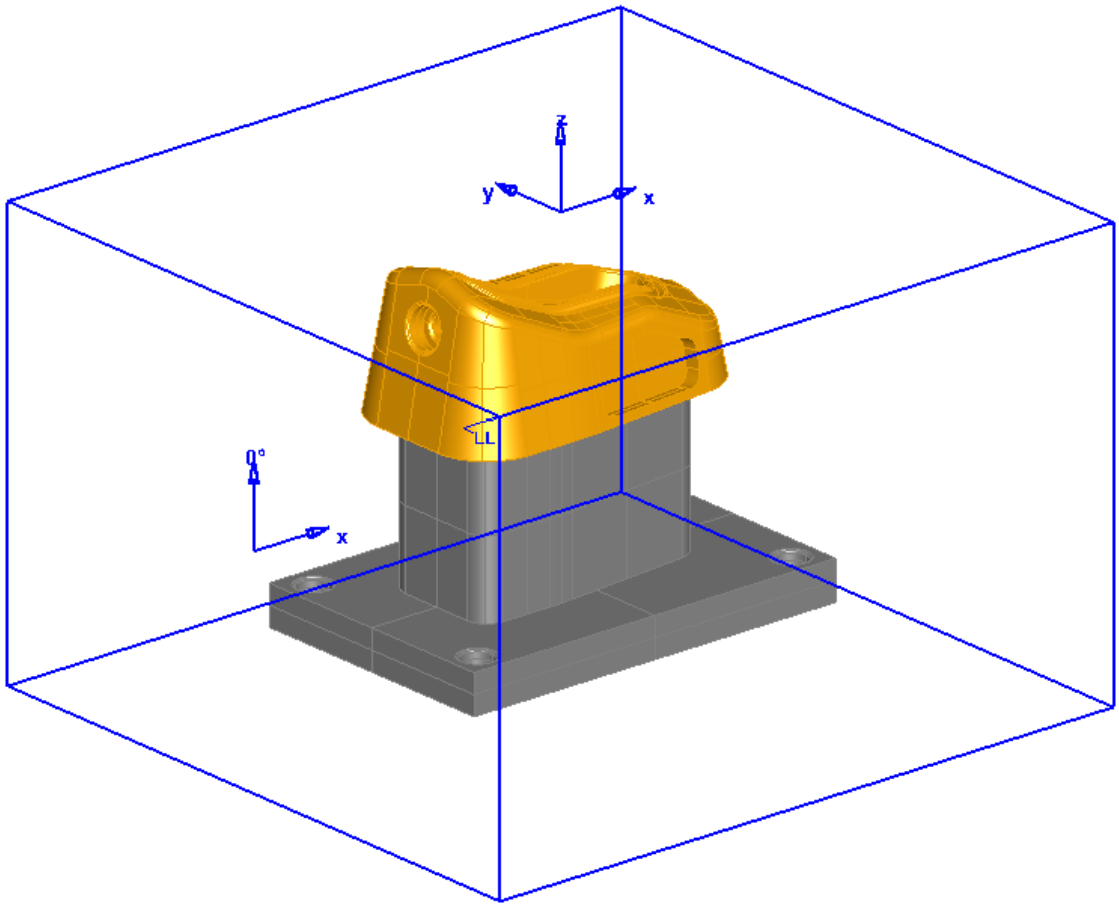
- Click **Next**
- Select **5th Axis positioning**



- Click **Next**

FeatureCAM now shows the Setup position (in the center of the top of the block) and the repositioned stock axis (in the center of the left face of the block). Note that the first indexing axis in this case is the X axis. The 0^{o} indicates that the part will be “this way up” when the A axis is at Zero degrees.

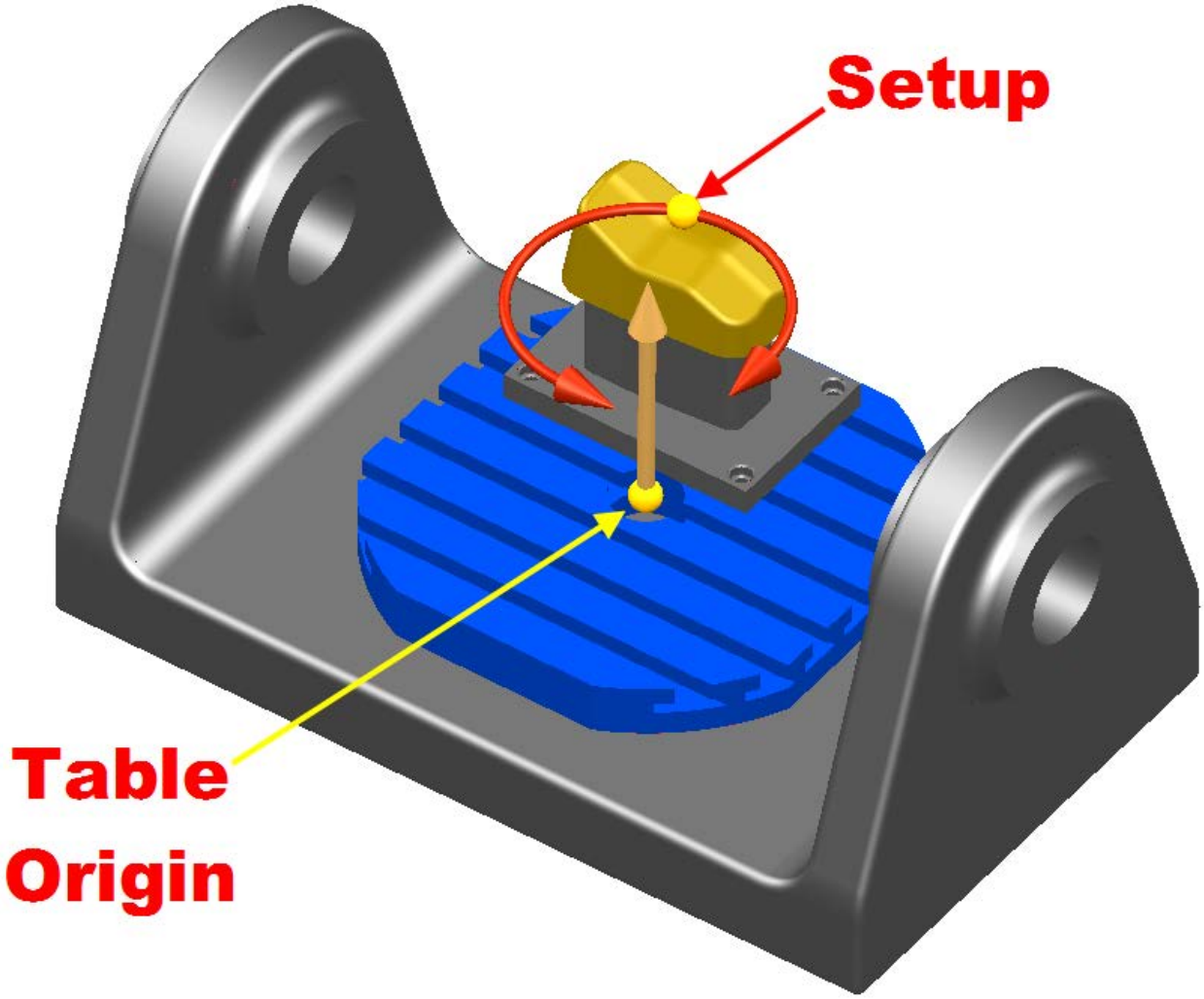
- Click **Finish**



Now that we have the setup axis aligned correctly, we will set up the stock etc.

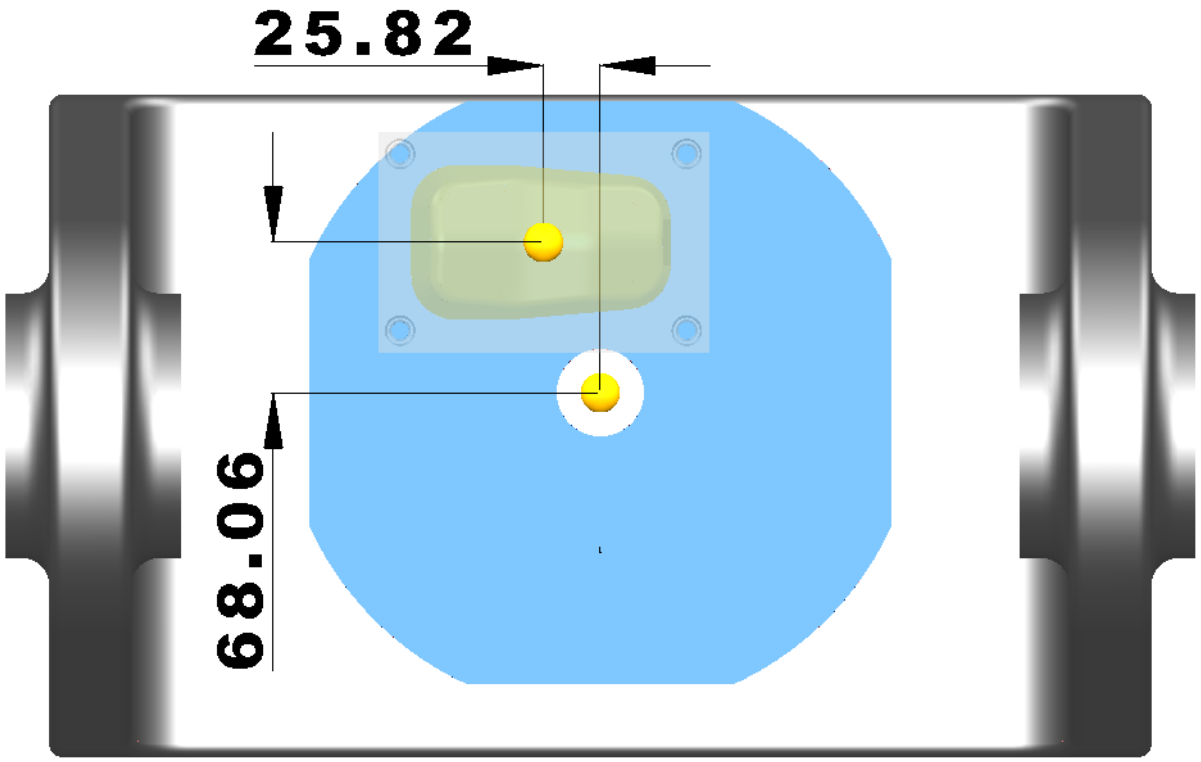
Positioning the part on the machine.

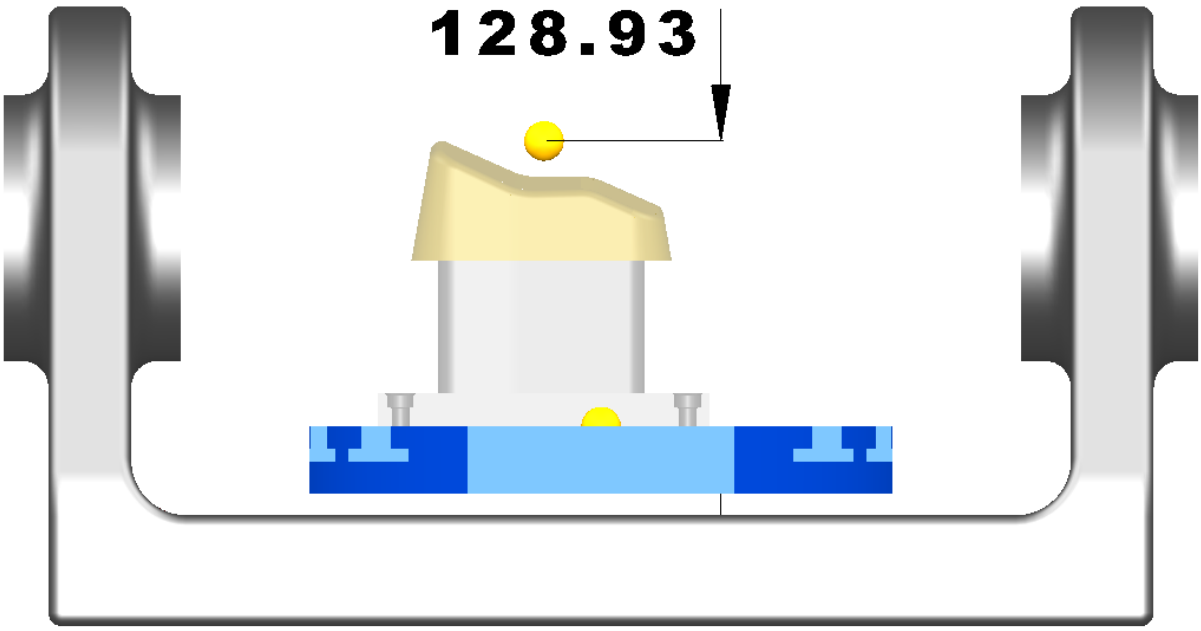
Once we have the part oriented and positioned correctly relative to the stock axis the next step is to position it relative to the Table origin. FeatureCAM needs to know where the part is situated on the machine tables so that it will know how its position will change when the tables rotate. In the illustration below, the part is placed off center on the C table, when the table rotates the part will move eccentrically in a circle around the Table origin as shown.



In order for FeatureCAM to be able to output the correct XYZ/AC coordinates, it needs to know what the X, Y and Z offsets are **FROM** the **Part Setup** (also known as the Part Reference point - this will be the first setup in the part) **TO** the **Table Origin**.

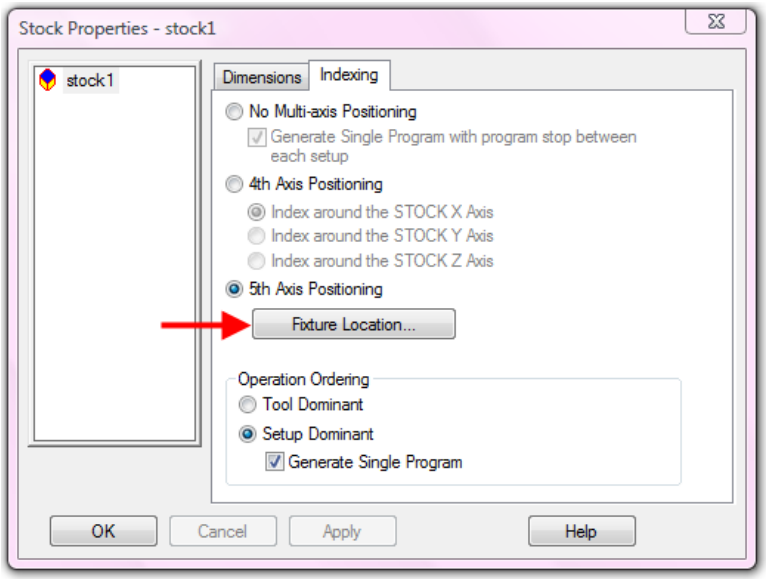
In the illustrations below the Setup is Behind (-Y), to the Left (-X) and Above (+X) of the Table origin.



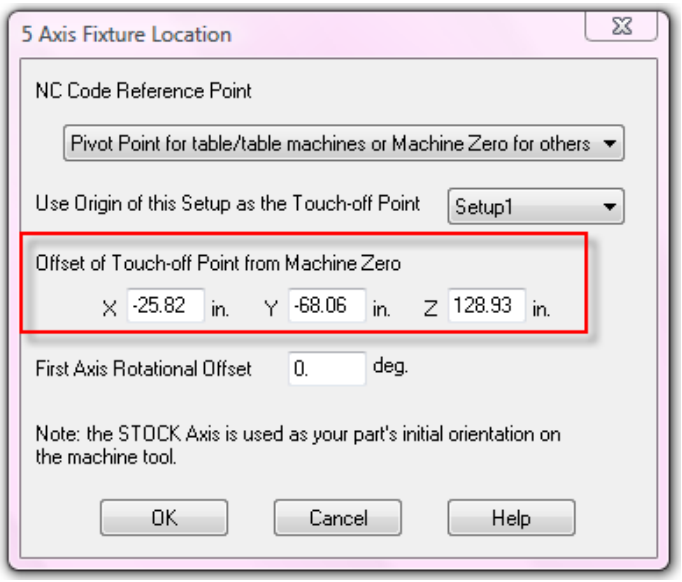


Its offset coordinates would be **X-25.82**, **Y-68.06** and **Z+128.93**. These coordinates are entered into the stock properties of the part.

- Double click on the **stock** to open its properties



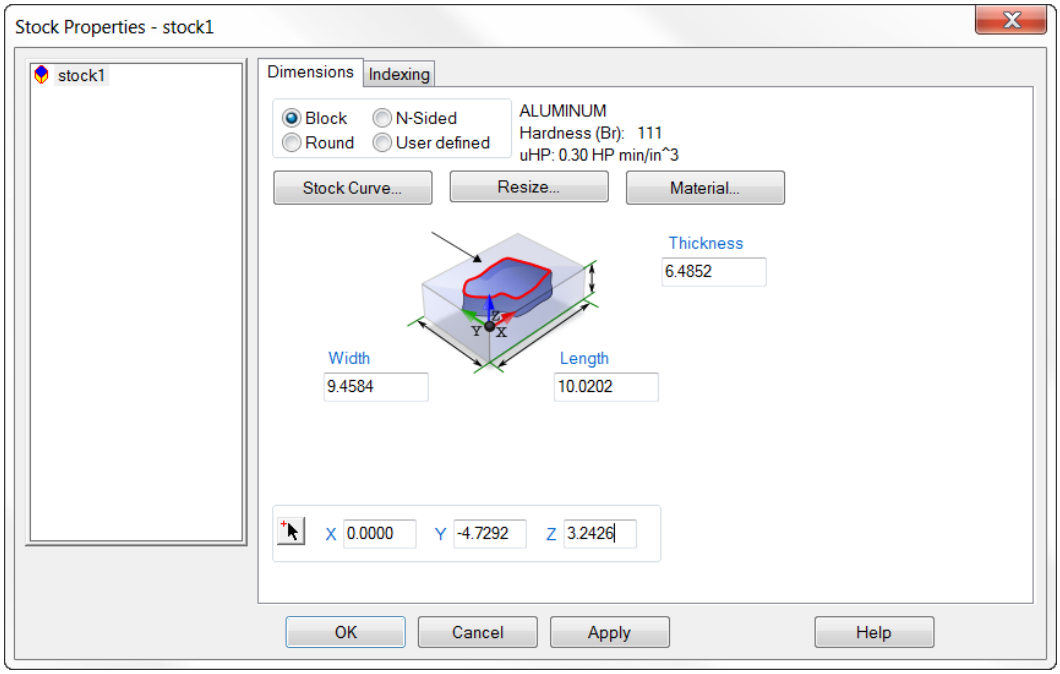
- Select the **Indexing tab** and click on **Fixture Location**



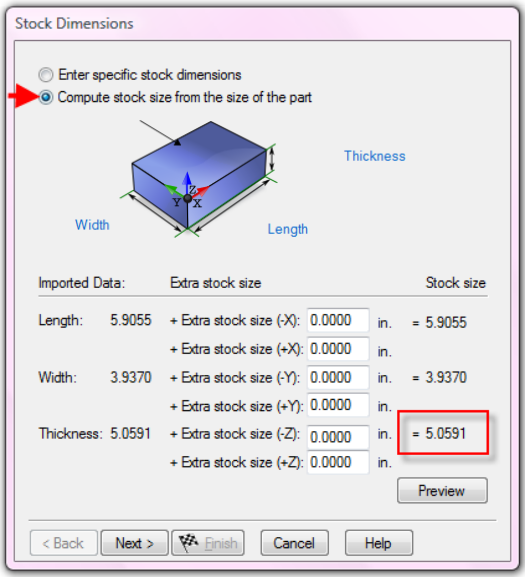
This is where the offsets from the setup to the table origin are entered. The figures shown above relate to the previous illustrations. In our case we want to place the part setup in the center of the table, so we will have Zero X and Y offsets and a positive Z offset. Before we enter the Z offset we need to find out the overall height of the part.

- Click **Cancel**

- On the **Dimensions tab** click **Resize**



- Click **Compute stock size from the part**

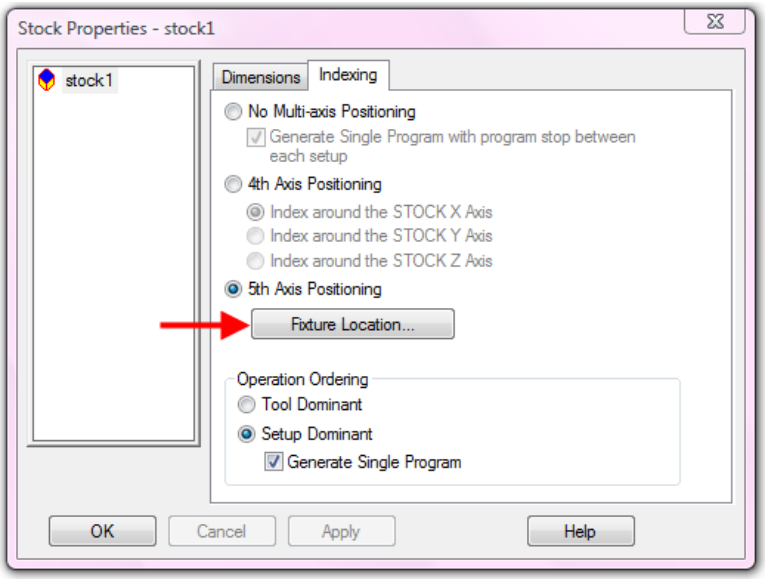


The overall height of the part is 5.0591” we need to now position the setup in the center of the part at the top so we can then enter the necessary offsets.

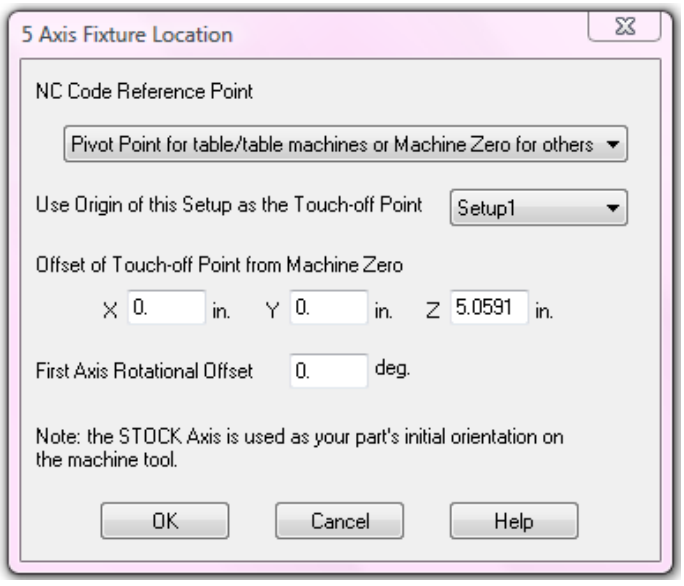
- Click **Next** and then **Finish**
- Double click the **setup** to open its properties
- Click **Edit**
- Click **Next** and check **Align to stock face**

- Click **Next** and select **Top** and **Center**
- Click **Finish**

- Double click on the **stock** to open its properties



- Select the **Indexing tab** and click on **Fixture Location**
- Change the **Z offset** value to be **5.0591”**



- Click **OK** then **Apply** followed by **OK**

The part is now correctly aligned with its setup 5.0591” directly above the center of the machine table. If we were now to generate a toolpath and post process it the NC code created would be correct for this location on the table.

If we were to decide to locate the part elsewhere on the table then we would need to go back into the stock properties and change the X and Y values in the fixture location and then re-post the program. Provided we do not change the height of the jig then the Z offset value does not need to be changed.

To Re-cap, the setup for 5-Axis machining consists of 2 parts:

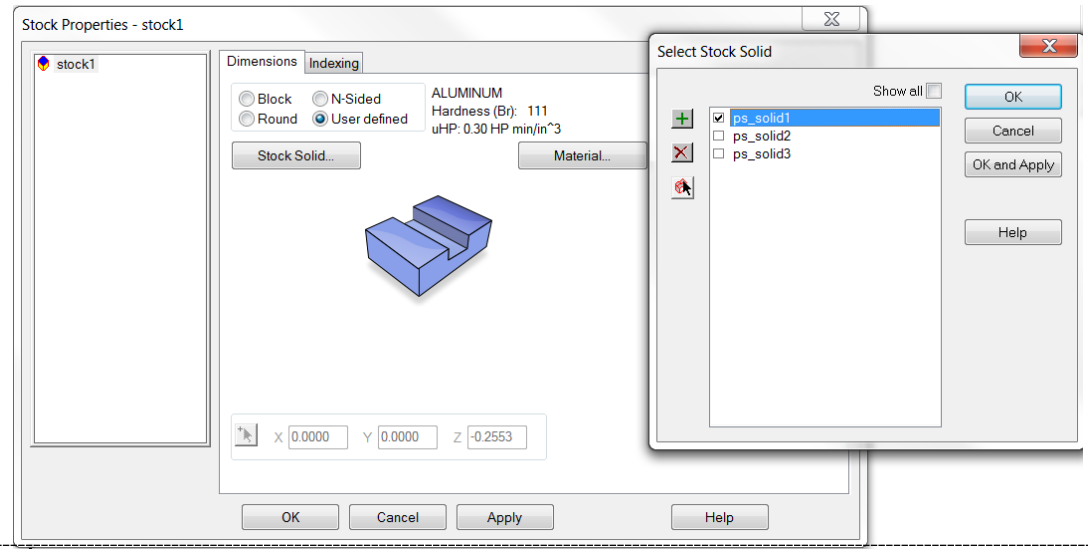
1) **Setting up the post processor to fix the offsets from the machine pivot point to the table origin:** This is machine specific and only needs to be done once unless the machine is dismantled for maintenance etc. The offsets are initially taken from values provided by the machine tool supplier and then refined by a series of cutting tests to zero in on the exact values required. This process is referred to as "Calibrating the post processor"

2) **Setting the position of the part setup relative to the table origin:** This varies from part to part and may be modified if the operator decides to reposition the part on the table. Any modifications will require the NC code to be regenerated.

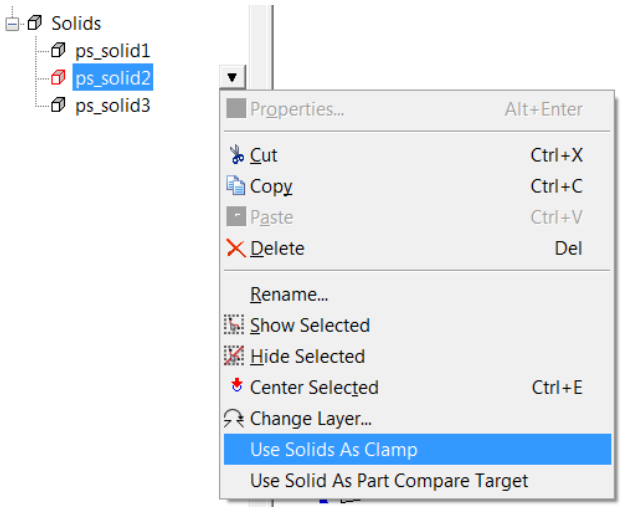
Simulation Setup.

Finally, we shall set up the stock type, include the jig solid as a clamp, and set up for machine simulation. These steps are primarily for visualization of the machine behavior during the cutting process and will not affect the actual NC code that is output. This said it is very useful to use a machine simulation to identify any possible collisions of the head with the table, over limit conditions and so on.

- Double click on the **stock** to open its properties
- On the **Dimensions tab** select **User defined**
- Select the solid **ps_solid1** as the stock
- Click **OK & Apply** and then **OK**



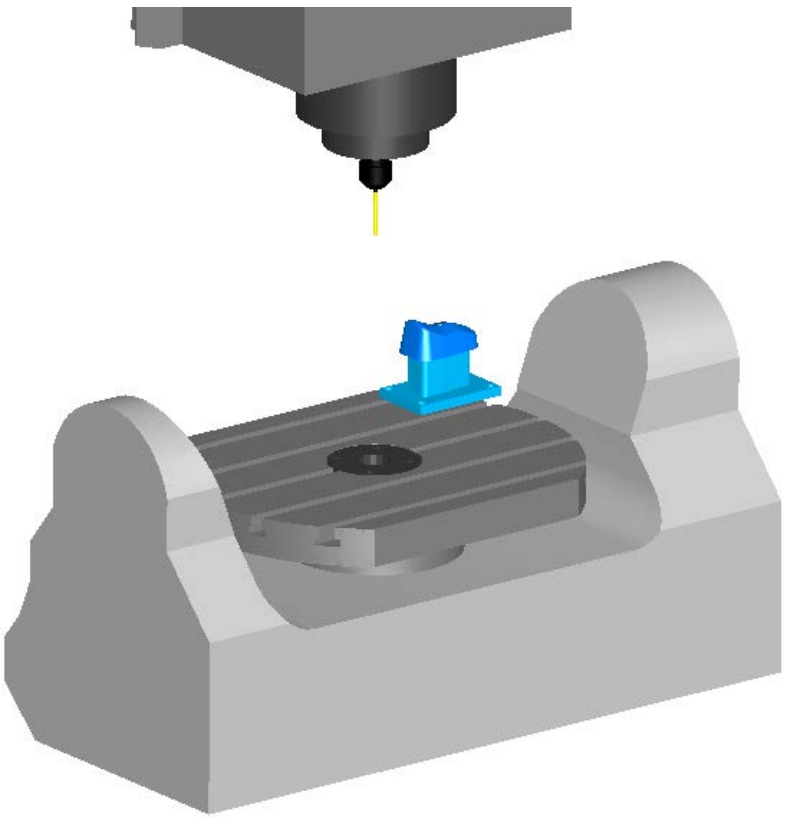
- Right click on **ps_solid2** and click **Use solid as clamp**



Finally, we need to position the part in the machine simulation. This is a purely visual location and does not affect the output NC code. In order to set this up, we need at least one operation so that we can run a simulation.

- Create a **hole** with all of the default **values**
- Select a **Machine simulation** and click **Single step**
- Click on the **"Select" arrow** on the main menu
- Click on any parts of the machine that are "in the way" so that you can see the position of the part on the table clearly

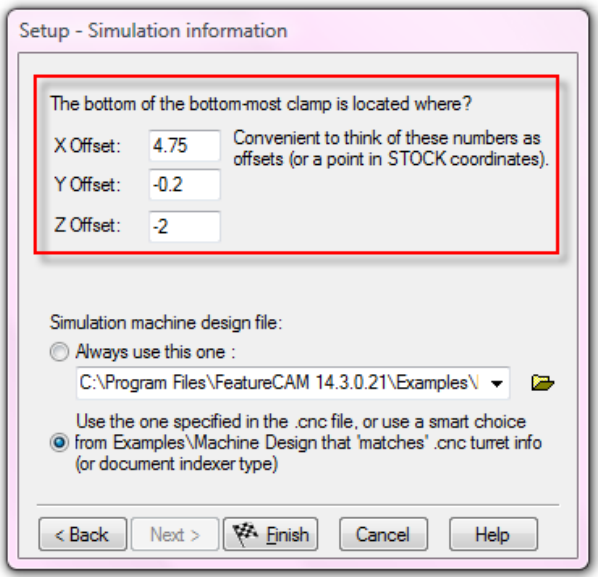
The part is off center and floating in space above the table. We now need to modify the location of the part relative to the table in the setup properties.



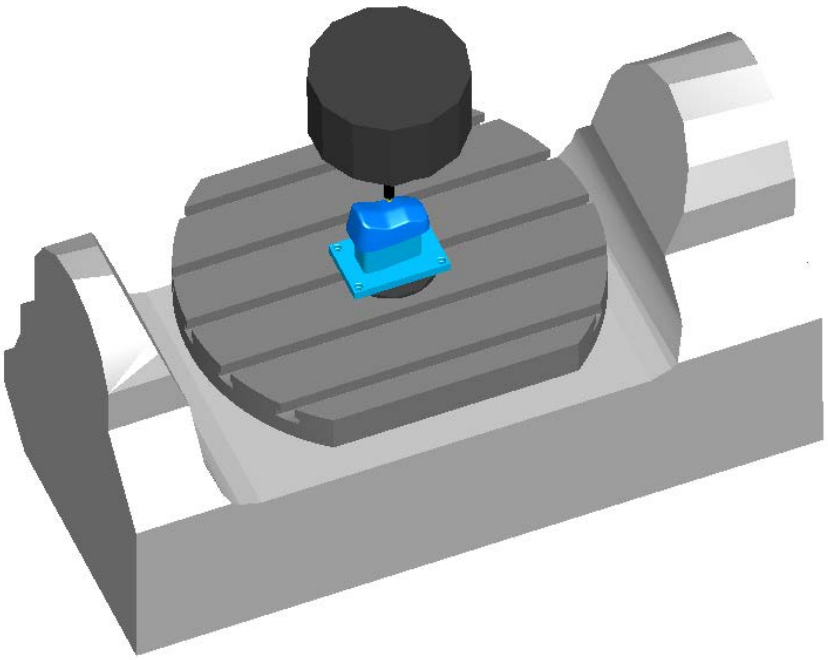
- Double click on **setup1** to open its properties
- Click **Edit** and then **Next twice**

The X,Y & Z offsets control the position of the part relative to the table.

- Enter the values as shown below



- Select a **Machine simulation** and click **Single step**
- Click on the **"Select" arrow** on the main menu
- Click on any parts of the machine that are "in the way" so that you can see the position of the part on the table clearly

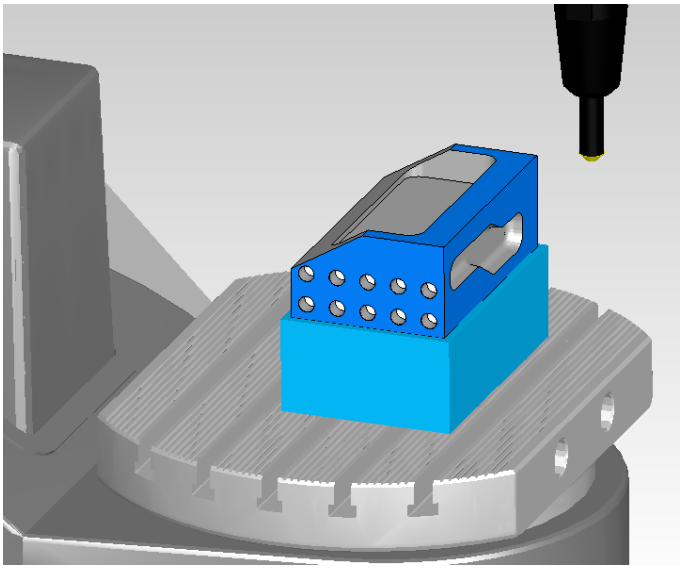


The part is now centered on the table. We are now ready to prepare and simulate 5-Axis toolpaths and output to the machine.

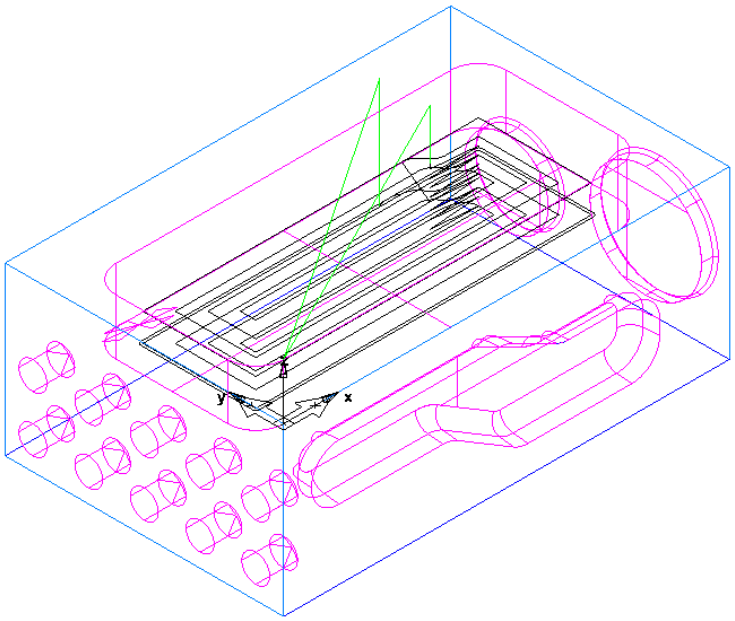
3+2 Positioning for 2.5D Milling

Introduction.

5-Axis positioning can be used together with FeatureCAM 2.5D milling to machine a part from multiple directions. In the first example we will look at a part which has five setups. If a user does not have access to 5-Axis positioning the part would have to be set up on the machine in five different orientations one of which would require setting the part up on an angle. The setup process would be time consuming and open to operator error.

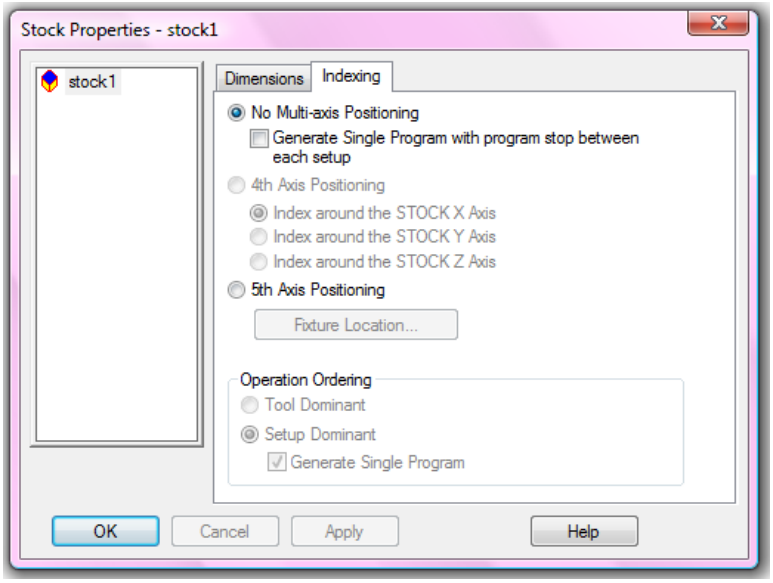


- Open the part **Simple_3+2.fm**
- Select **setup1** and run a **Centerline simulation**



As the part is set up for 3-Axis milling the NC code is produced for just the one setup.

- Edit the **Stock properties** and click on the **Indexing tab**

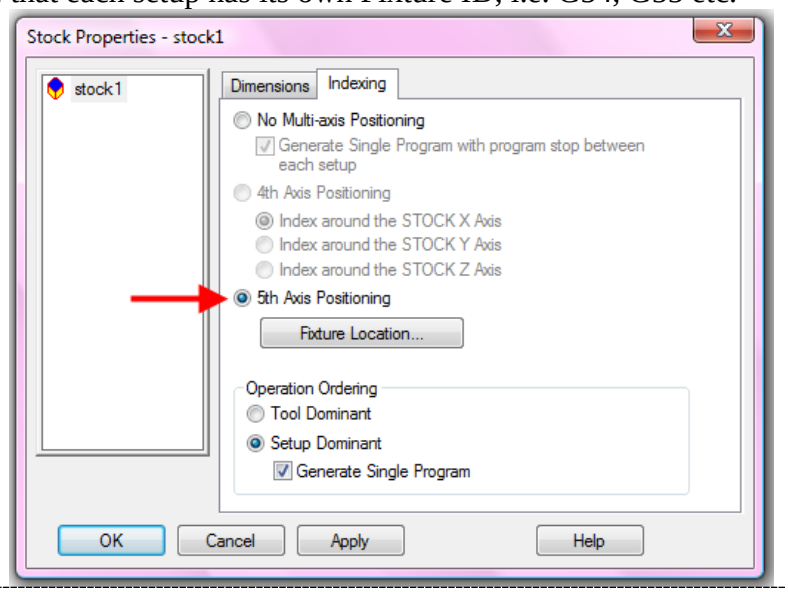


With **No Multi-axis Positioning** selected we have just two choices, we can post process each setup separately, or create a single program with a program stop between each setup. Either way, the operator will need to remove the part from the machine, replace it in the new position, clamp it, set it up accurately and then run the operations from the next setup. If it takes 5 minutes to set up in each position, this will add 20 minutes to the total time taken to produce the part.

- Check **Generate a single program**
- Click **Apply** and then **OK**
- From the **Manufacturing** menu select **Post process**
- Click **Browse** the **Data** folder
- Select the **Haas-5 axis.cnc** post processor
- Run a **Centerline simulation**

Check the NC code; you will see that each setup has its own Fixture ID, i.e. G54, G55 etc.

- Edit the **Stock properties** and click on the **Indexing tab**
- Check **5th Axis Positioning**
- Click **Apply** and then **OK**

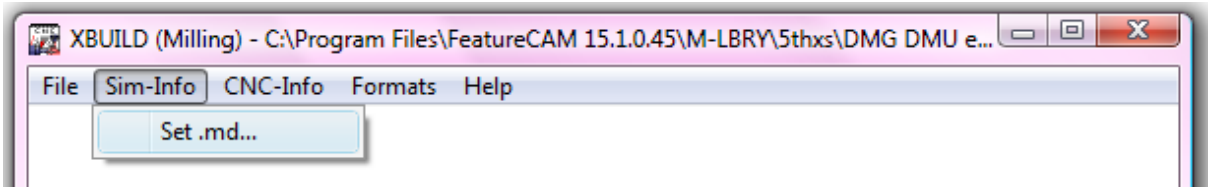


- Select **setup1** and run a **Centerline simulation**

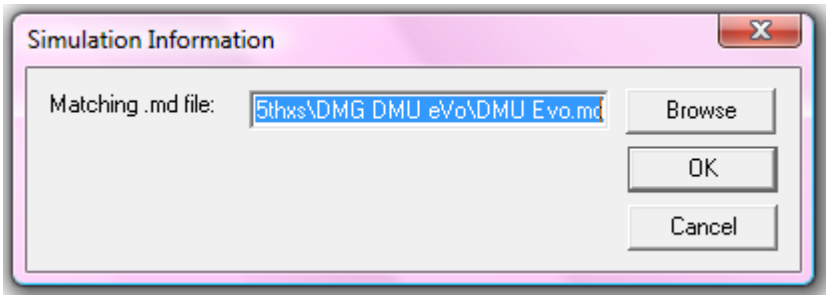
All of the setups are now compressed into a single program with no program stops. Each setup still has its own fixture ID but everything is now working off the initial setup. Once the operator has located the part accurately for the first setup the following setups are automatically in the correct position. The operator does not need to do any further setup. This saves a significant amount of time and eliminates the possibility of operator error in setting up the part in between operations.

In order to see how this will actually work on a machine we will now change to a different post processor and do a machine simulation.

- From the **Manufacturing** menu select **Post process**
- Click **Browse** and go to the **Data** folder
- Select the **DMG eVo 50 Heid iTnc 530.CNC** post processor
- Click on **Edit** and select **Sim info** then **Set .md…**

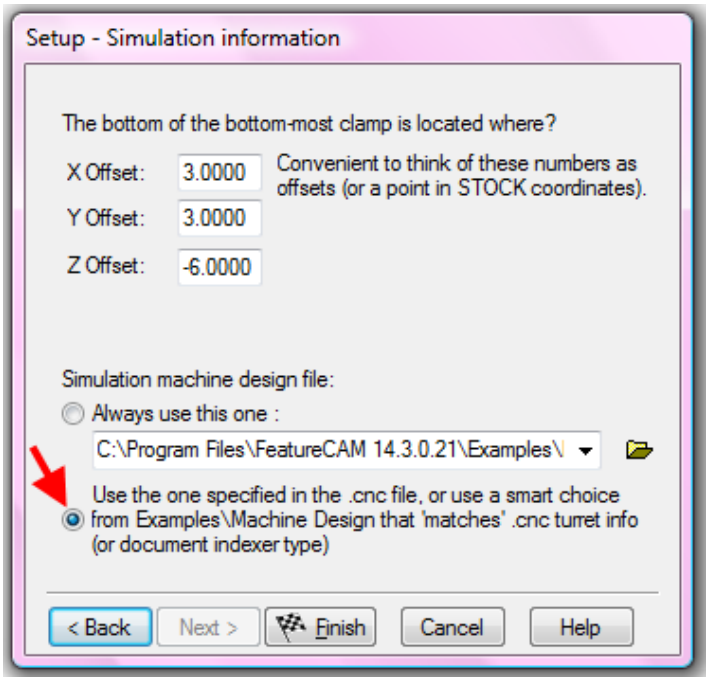


- Make sure that the **Machine Design file** matches the post

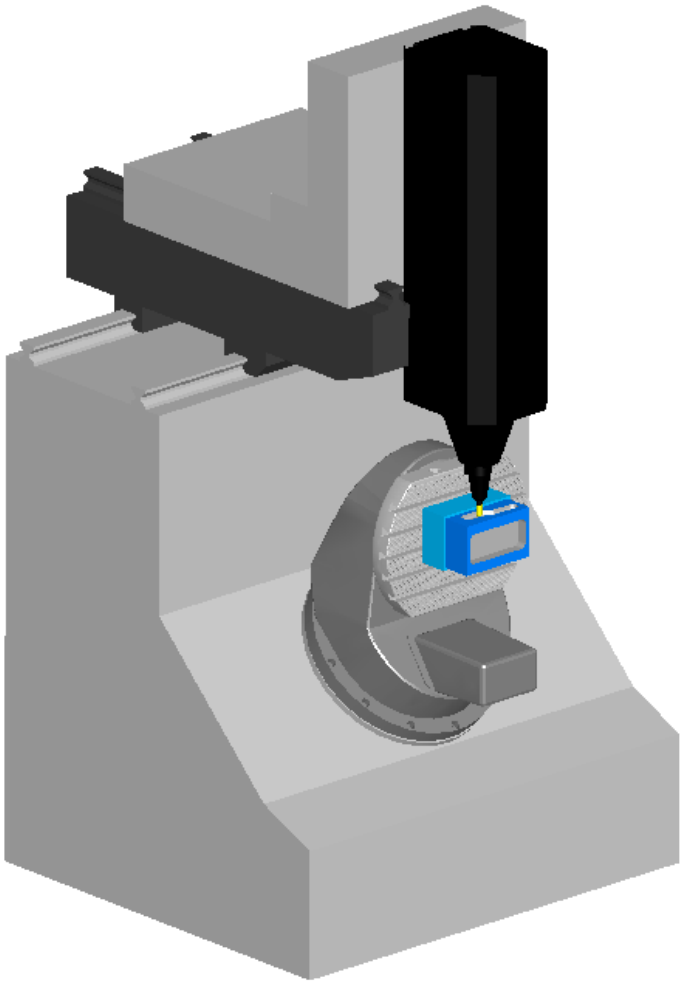


- Click **OK** and then close **XBUILD**
- Edit **setup1**
- Click **Next** until you reach the **Setup - Simulation information** page

- Make sure that FeatureCAM is set to use the Machine Design file that we just specified in the post processor



- Run a **Machine simulation**



All five setups are machined in a single operation eliminating the setup time and reducing the risk of operator errors. We have switched from the Haas post which was using a trunnion table on the A axis with a C table on top to the DMU Evo post which uses a C table mounted on a 45 degree inclined table (B axis). Take a look at the NC code below to compare that from the Haas with the DMG.

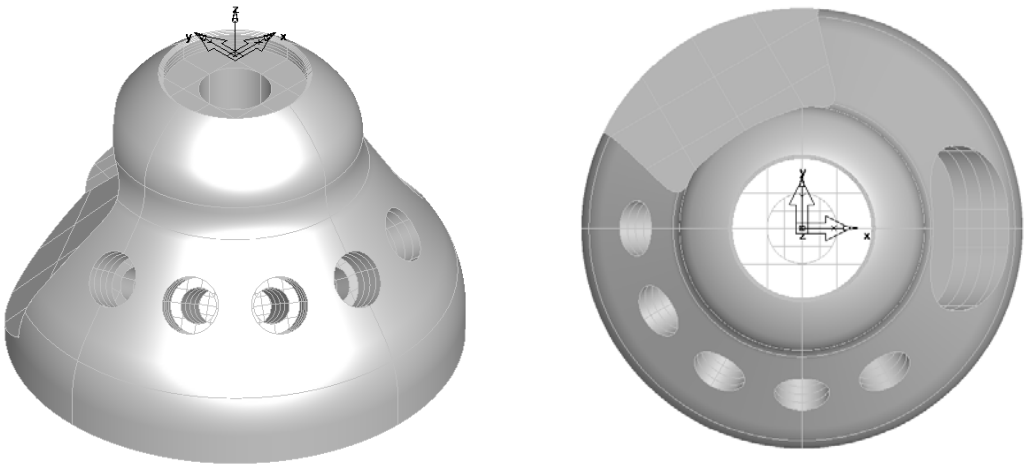
```
%
O1000
(SIMPLE_3+2)
( 02/18/2009 20:22:06 )
( FEATURECAM HAAS 5 AXIS MILLING )

N35 ( OPERATION= ROUGH1 RECTANGULAR POCKET
N40 ( TOOL: T1 = 0.875 dia. ENDMILL0875:REG )
N45 G00 G17 G40 G90 G94
N50 T1 M6
N55 G54 X5.2634 Y2.6384 A0. B0. S2837 M03
N60 G43 H1 Z1.0 M08
```

```
10 BEGIN PGM Simple_3+2 INCH
15 TOOL CALL 1 Z S2837 ; endmill0875:reg
20 L Z-0.1 R0 FMAX M91
25 L X-501.1 Y-421.1 R0 FMAX M91
30 CYCL DEF 7.0 DATUM SHIFT
35 CYCL DEF 7.1 X+0
40 CYCL DEF 7.2 Y+0
45 CYCL DEF 7.3 Z+0
50 CYCL DEF 7.4 B+0
55 CYCL DEF 7.5 C+0
```

3+2 with Feature Recognition.

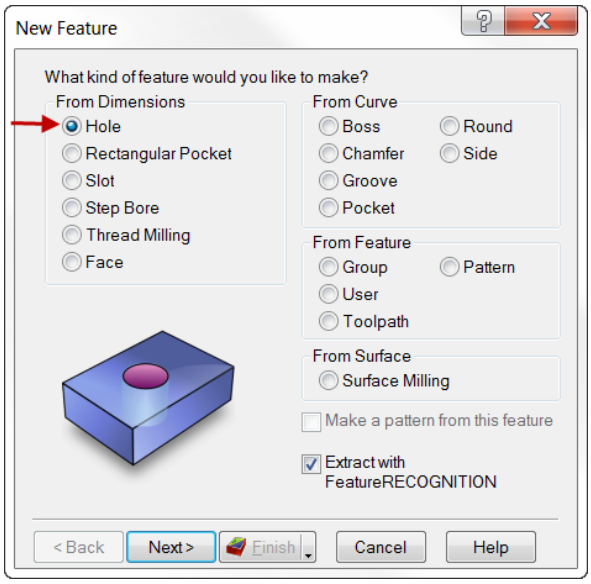
In this example we will be working off a solid model and using Feature Recognition to identify features off the part. In order to do this, we will need to create additional setups to set the Z axis for each feature to be recognized.

- Open the part **Feature_Recognition.fm** located in the **Data** folder
- Select an **Isometric view** then a **Top View**

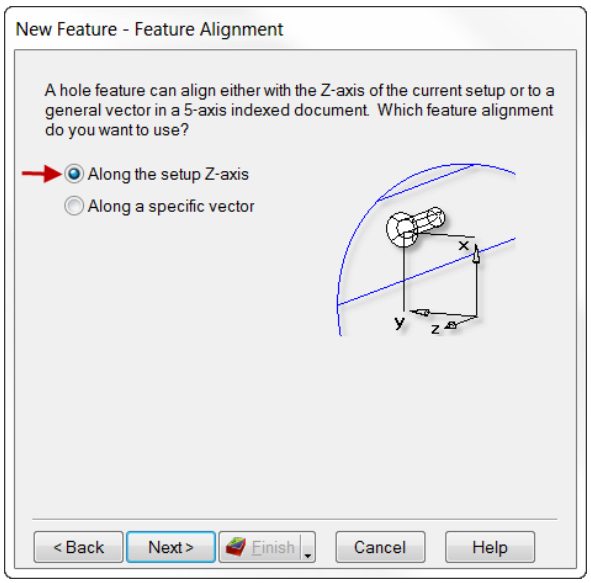


This part has a number of features including a Pocket, a Side and a number of holes which are oriented in different directions. First of all, we will prepare toolpaths for the holes.

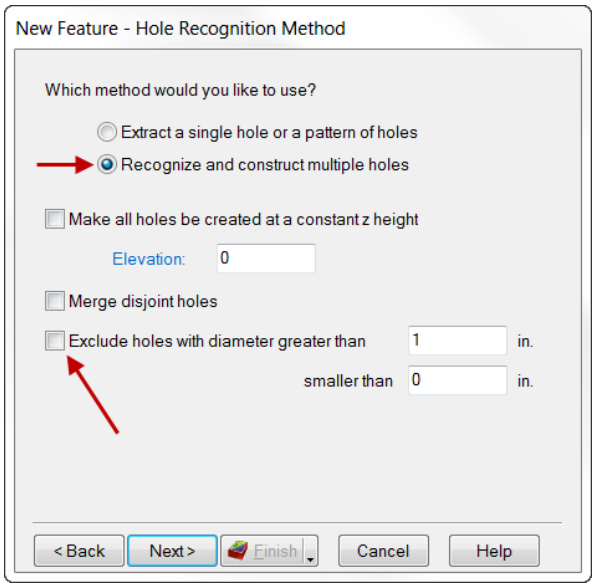
- Open the **New feature wizard**
- Check **Hole** and **Extract with FeatureRecognition**
- Click **Next**



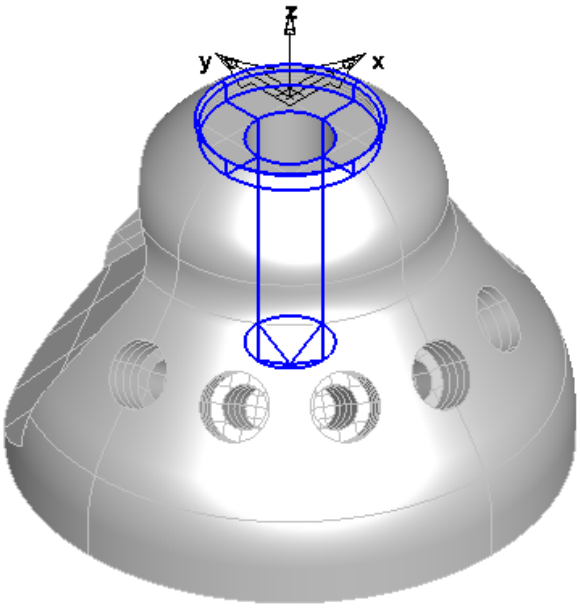
- Check **Along the setup Z-axis**
- Click **Next**

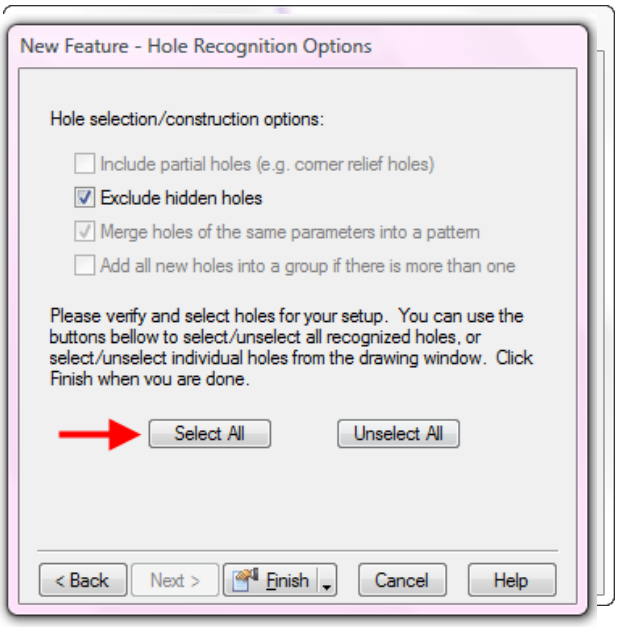


- Check **Recognize and construct multiple holes**
- Uncheck **Exclude holes…**
- Click **Next**



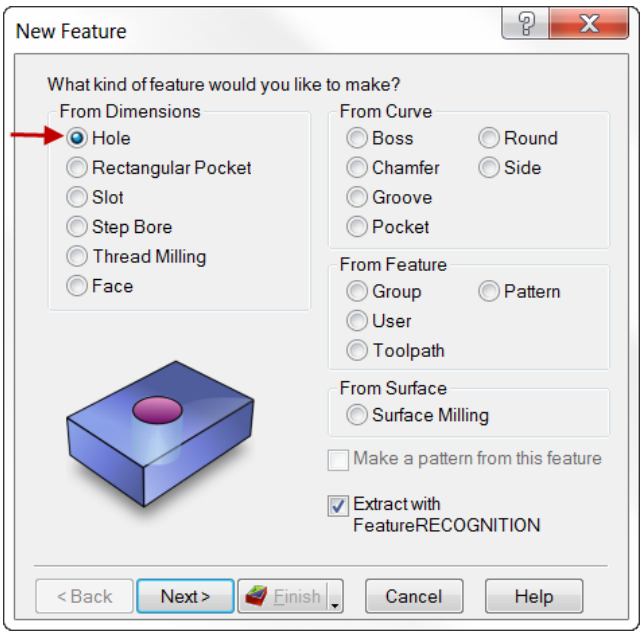
- Click **Select all** then **Finish** and then **OK**





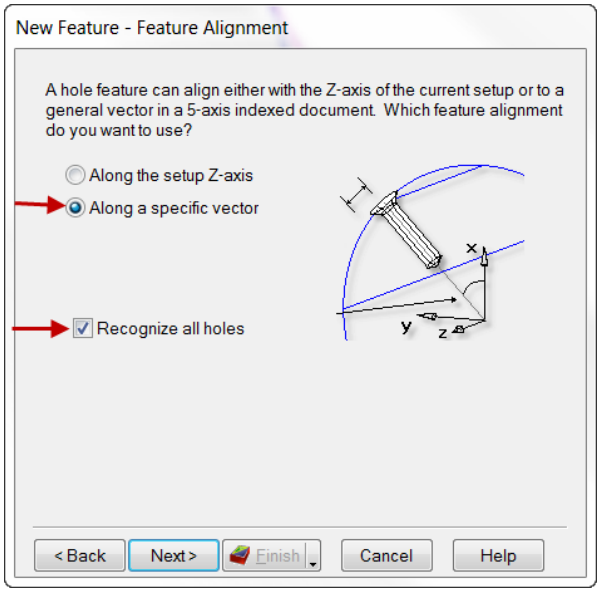
FeatureCAM has recognized the large hole that is aligned with the setup Z-Axis. We will now recognize the other, angled holes around the part.

- Open the **New feature wizard**
- Check **Hole** and **Extract with FeatureRecognition**

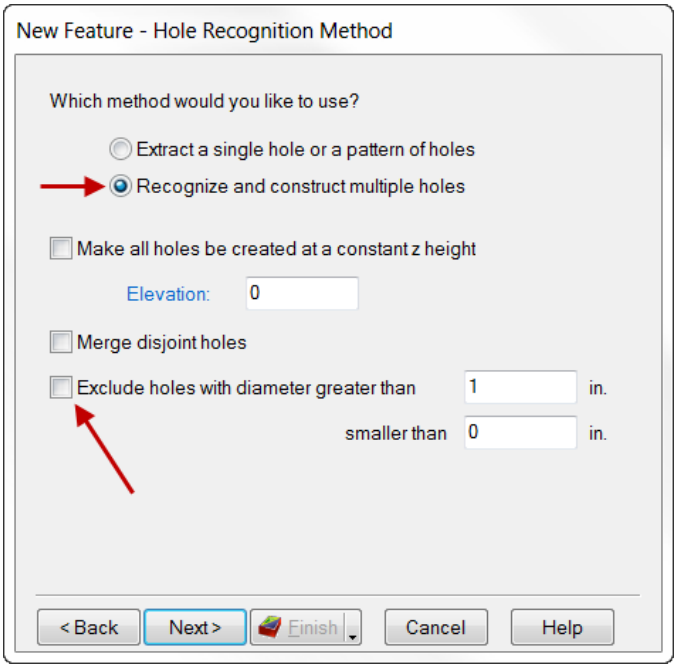


- Click **Next**

- Check **Along a specific vector**
- Check **Recognize all holes**



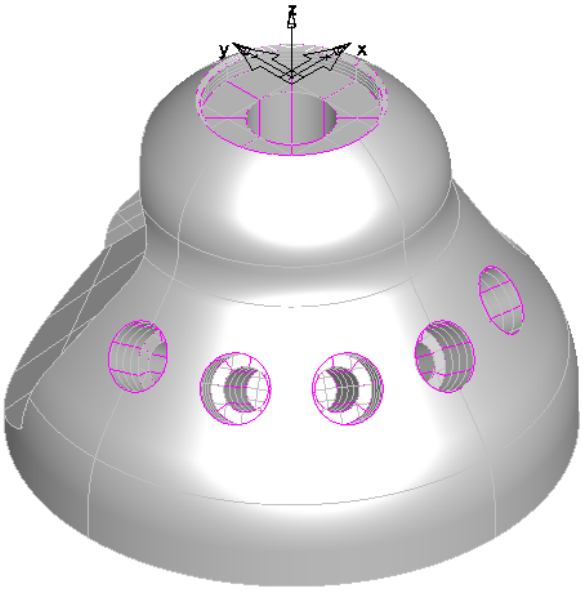
- Click **Next**
- Check **Recognize and construct multiple holes**
- Uncheck **Exclude holes…**



- Click **Next**

- Click **Select all** then **Finish** and then **OK**

FeatureCAM has now recognized all of the remaining holes in the part, regardless of their orientation to the setup Z-Axis. When we post process the program FeatureCAM will adjust the tool axis relative to the part so that it can reach each hole in turn. We will now post the program using the Haas 5-Axis post so that we can see the way the A and C axes change for each hole.

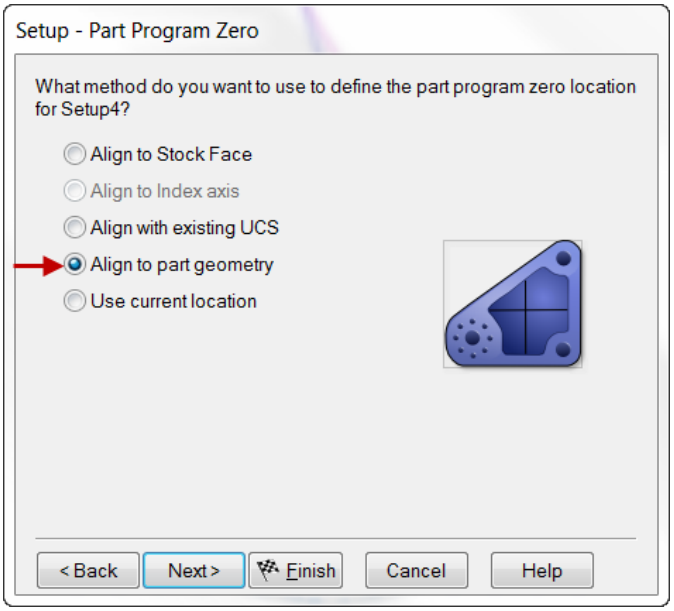


- From the **Manufacturing** menu select **Post process**
- Click **Browse** and go to the **Data** folder
- Select the **Haas-5 axis.cnc** post processor
- Run a **Centerline simulation**
- Check the **NC Code**

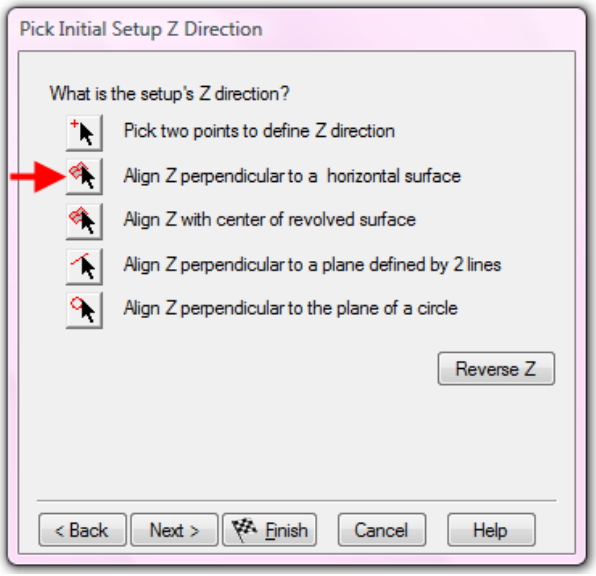
```
N770 ( COUNTERBORE POCKET HOLE3 )
N775 G00 G49 G53 Z0.
N780 G54 X0. Y-1.7371 A-55.0 B0.
N785 G43 H7 Z4.2023
N790 Z0.3414
N795 G01 Z-0.0342 F24.8
N800 X0.009 Y-1.7281 F49.6
N805 G03 X0. Y-1.7191 I-0.009 J0. F24.8
N810 X0. Y-1.7191 I0. J-0.0094
N815 X-0.0045 Y-1.7203 I0. J-0.0094
N820 X-0.008 Y-1.7325 I0.0044 J-0.0078
N825 G01 X0.0042 Y-1.736 F49.6
N830 G00 Z1.2415
N835 Z4.2023

N840 ( COUNTERBORE POCKET HOLE4 )
N845 G00 G49 G53 Z0.
N850 G54 X0. Y-1.7371 A-55.0 B-90.0
N855 G43 H7 Z4.2023
N860 Z0.3414
N865 G01 Z-0.0342 F24.8
N870 X0.009 Y-1.7281 F49.6
N875 G03 X0. Y-1.7191 I-0.009 J0. F24.8
N880 X0. Y-1.7191 I0. J-0.0094
```

We will set the part up so that we can create the pocket feature. We will do this by creating a setup aligned with its Z-Axis perpendicular to the bottom of the pocket; we will then use Feature Recognition to create the pocket feature.

- **Eject** the **simulation**
- Double click on **setup1** to open its properties
- Select **New**
- Click **Next**
- Check **Align to part geometry**

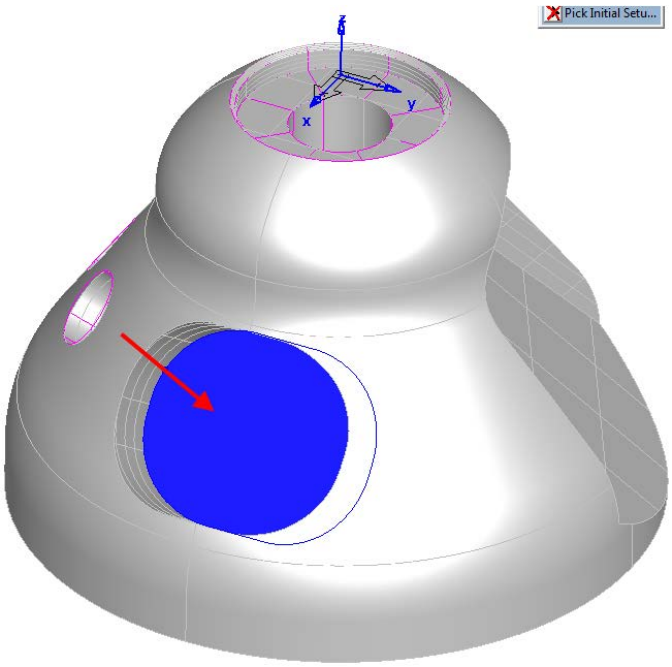


- Click **Next**
- Click **Align Z perpendicular to a horizontal surface**



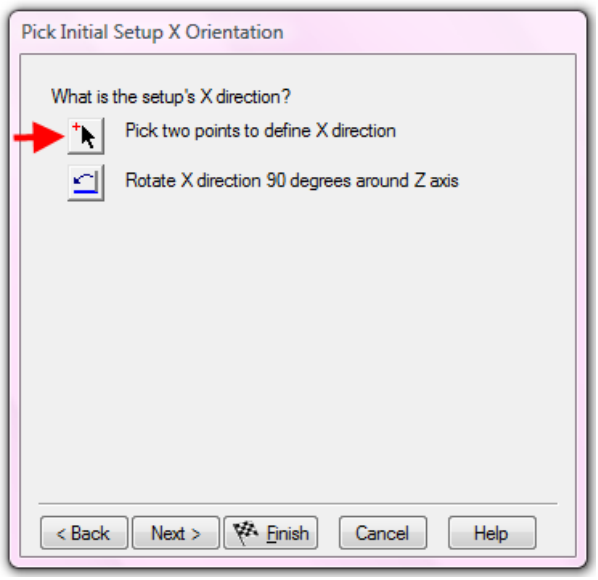
- Click **Next**

- Click on the **horizontal surface** at the bottom of the pocket



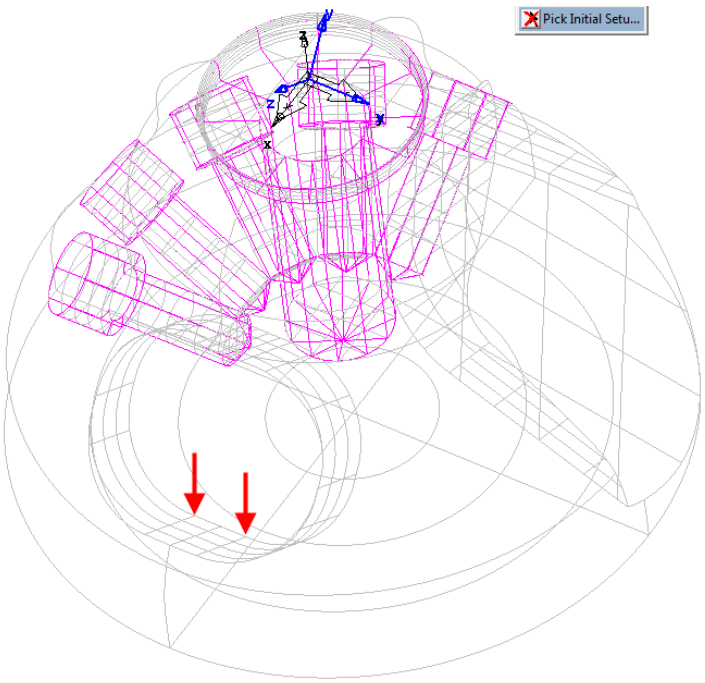
This will set the Z –Axis of the new setup so that it is aligned perpendicular to the bottom of the pocket.

- Click **Pick two points to define X direction**

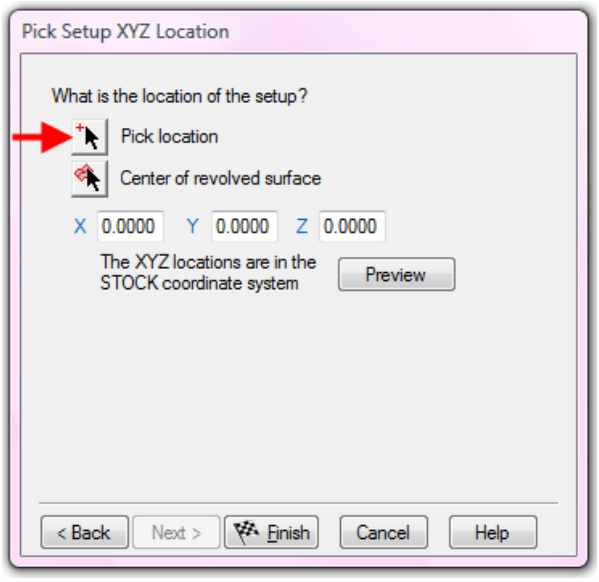


This option allows us to align the setup X-Axis. This is not strictly necessary as all coordinates will eventually be written out relative to setup1. However it may be useful if we wish to add extra geometry or a hole pattern later.

- Click on the two points indicated on the bottom edge of the pocket. Work from Left to Right as this will define the direction

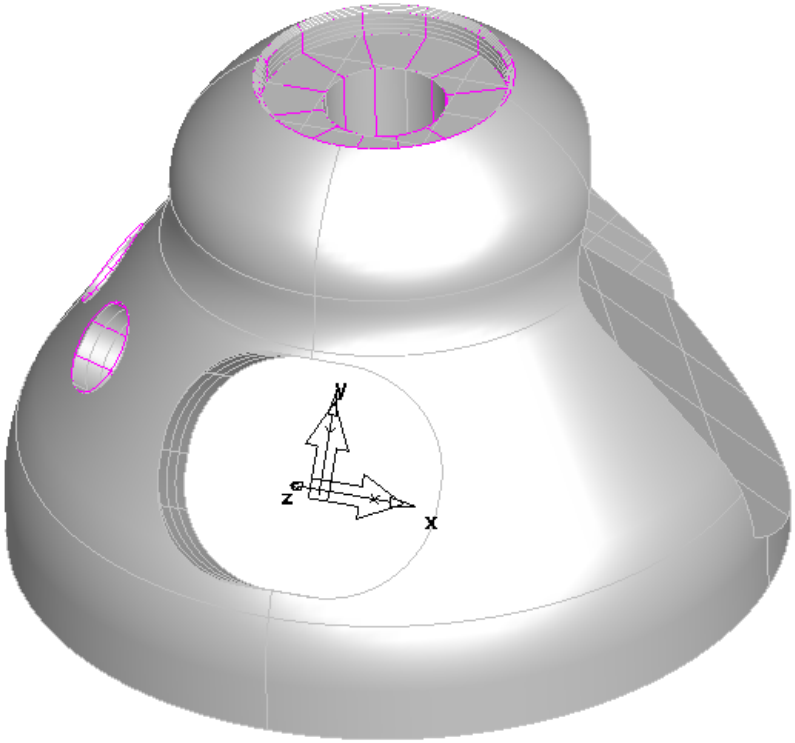


- Click on **Pick location**



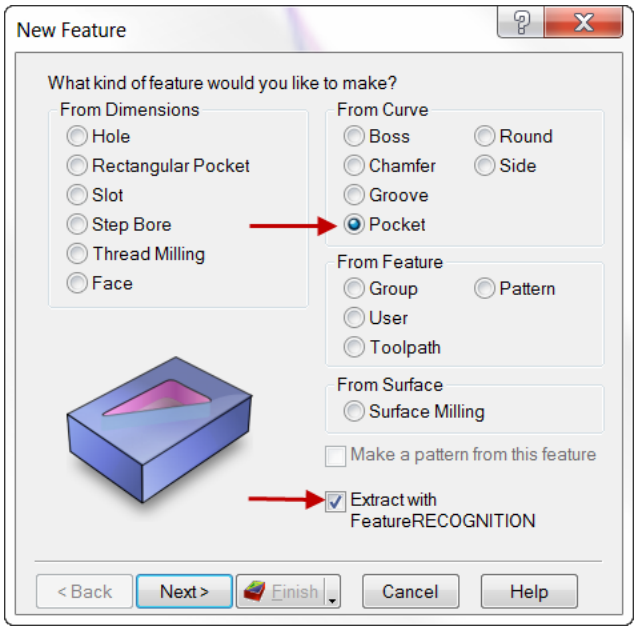
- Click on the centre of the pocket

Again the location is not critical; it is useful though to position the setup by the feature so you can see which setup and feature belong together.



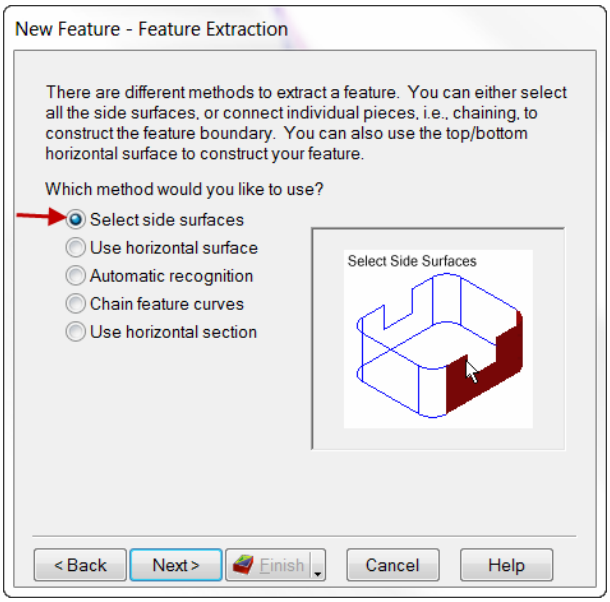
The finished setup. We are now ready to go ahead and create the pocket feature.

- Open the **New feature wizard**
- Check **Pocket** and **Extract with FeatureRecognition**

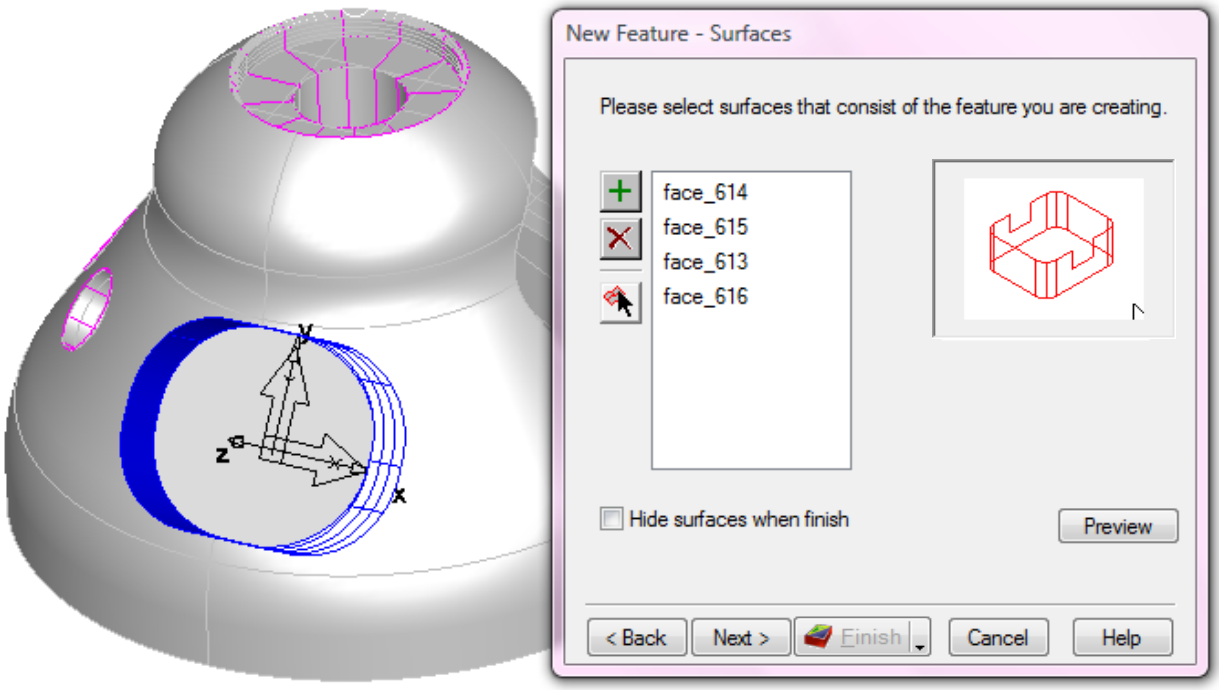


- Click **Next**

- Check **Select side surfaces**

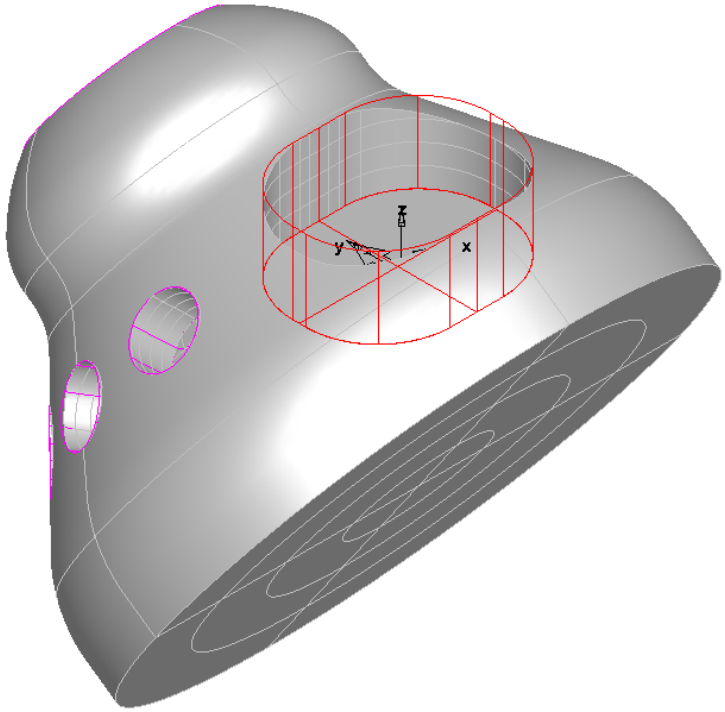


- Click **Next**
- Select the surfaces that make up the side of the pocket and add them to the feature



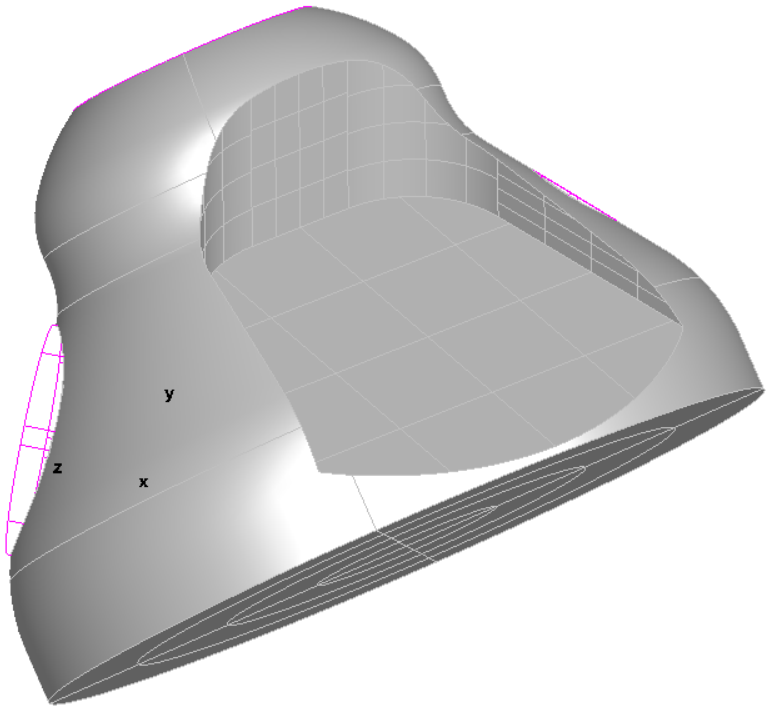
- Click **Next**

- Click **Next**
- Click **Finish**
- Take an **Isometric view**

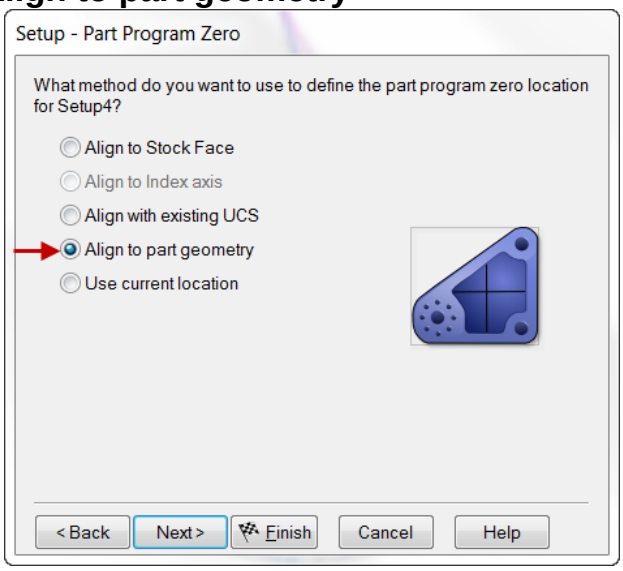


The pocket has been created. We will now create a setup for the side feature.

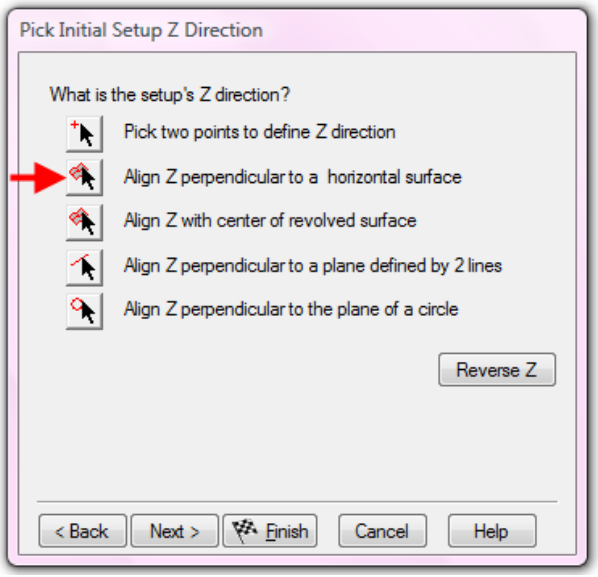
- Rotate the view so that the side is visible



- Double click on **setup1** to open its properties
- Select **New**
- Click **Next**
- Check **Align to part geometry**

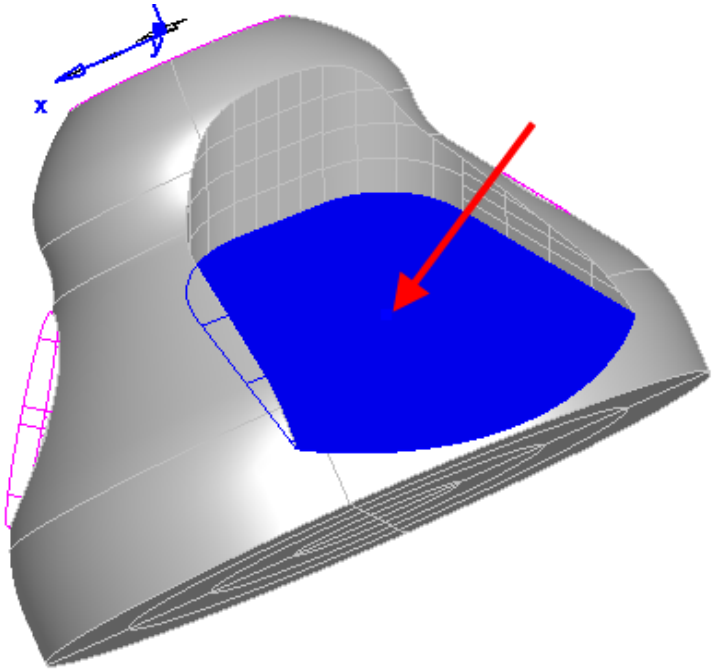


- Click **Next**
- Click **Align Z perpendicular to a horizontal surface**

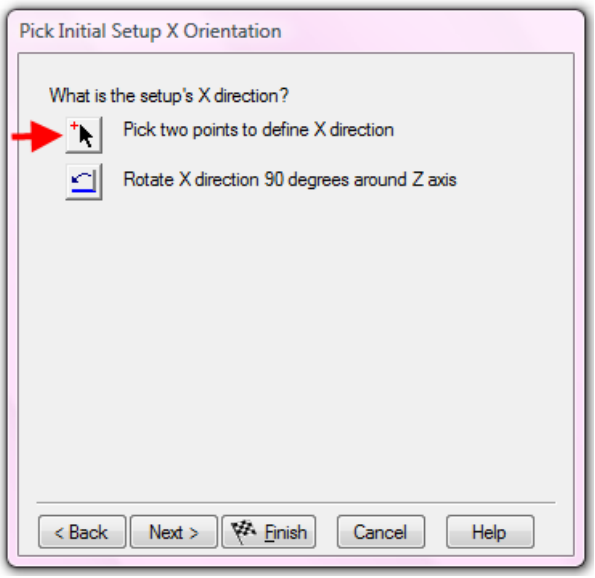


- Click **Next**

- Click on the **horizontal surface** at the bottom of the Side



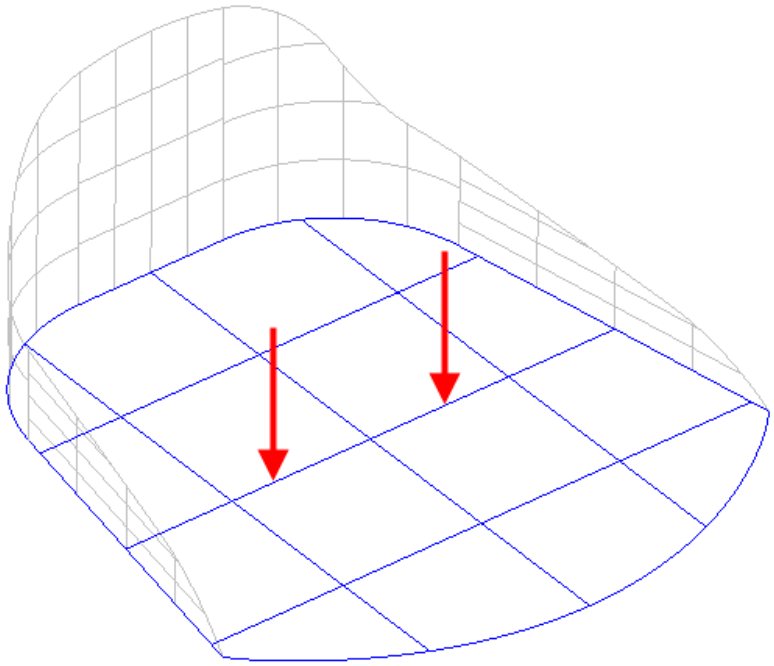
- Click **Pick two points to define X direction**



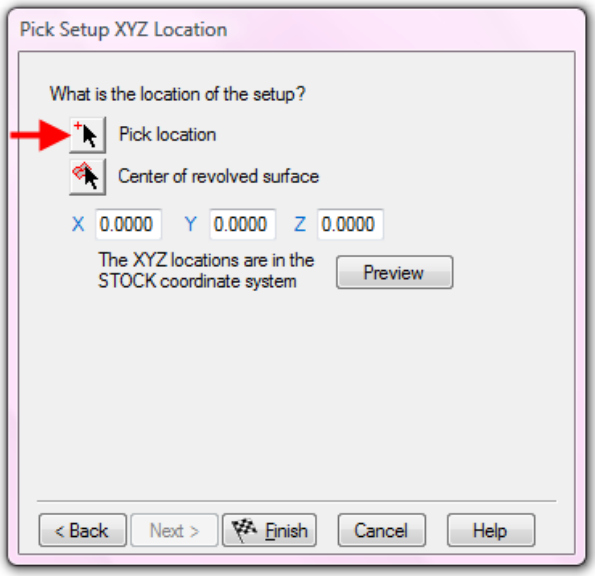
This option allows us to align the setup X-Axis. This is not strictly necessary as all coordinates will eventually be written out relative to setup1. However it may be useful if we wish to add extra geometry or a hole pattern later.

- Turn off shading

- Click on the two points indicated centre isoline of the surface.
  Work from Left to Right as this will define the direction

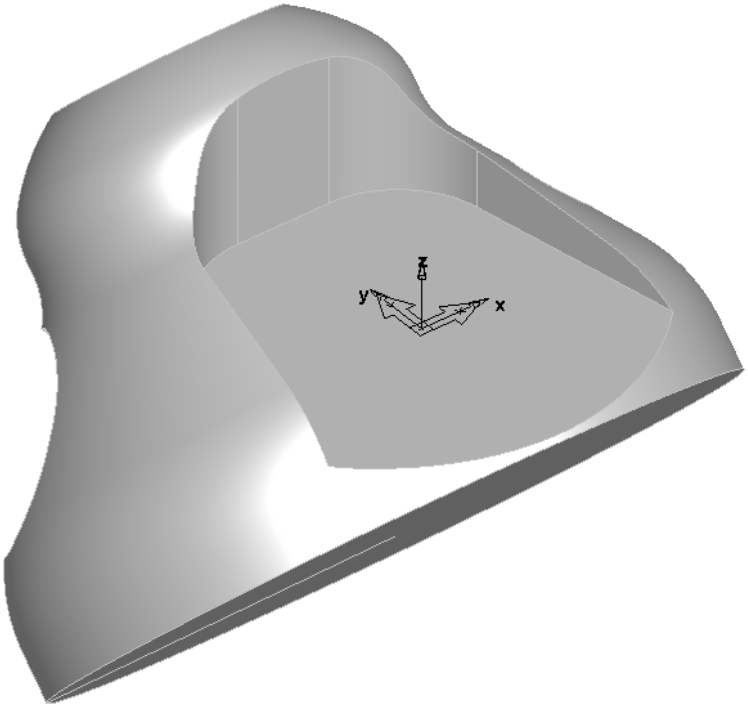


- Click on **Pick location**



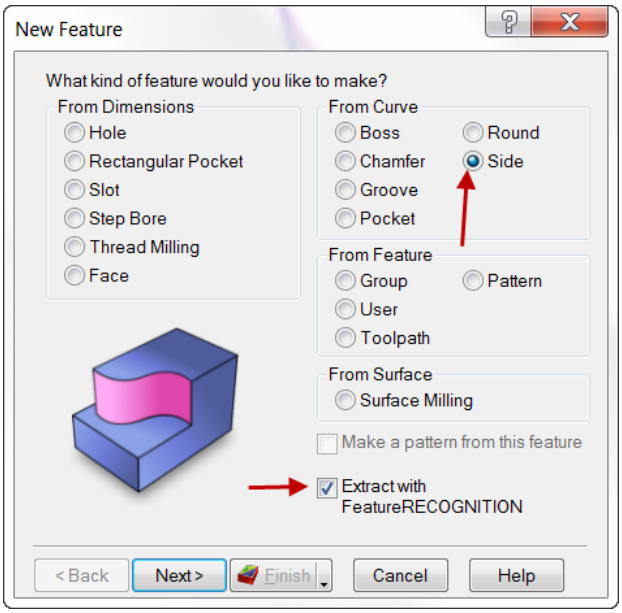
- Click on the centre of the surface

Again the location is not critical; it is useful though to position the setup by the feature so you can see which setup and feature belong together.



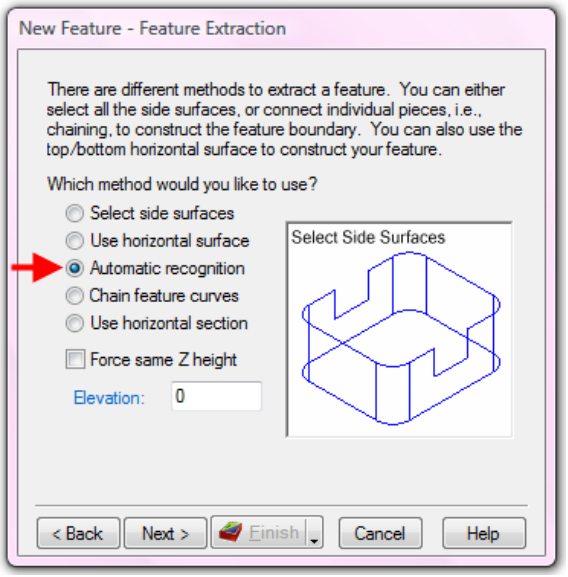
We are now ready to create the side feature, this time we will use Automatic Feature Recognition (AFR). This method looks along the setup Z-Axis and identifies any features that are aligned with it.

- Open the **New feature wizard**
- Check **Side** and **Extract with FeatureRecognition**

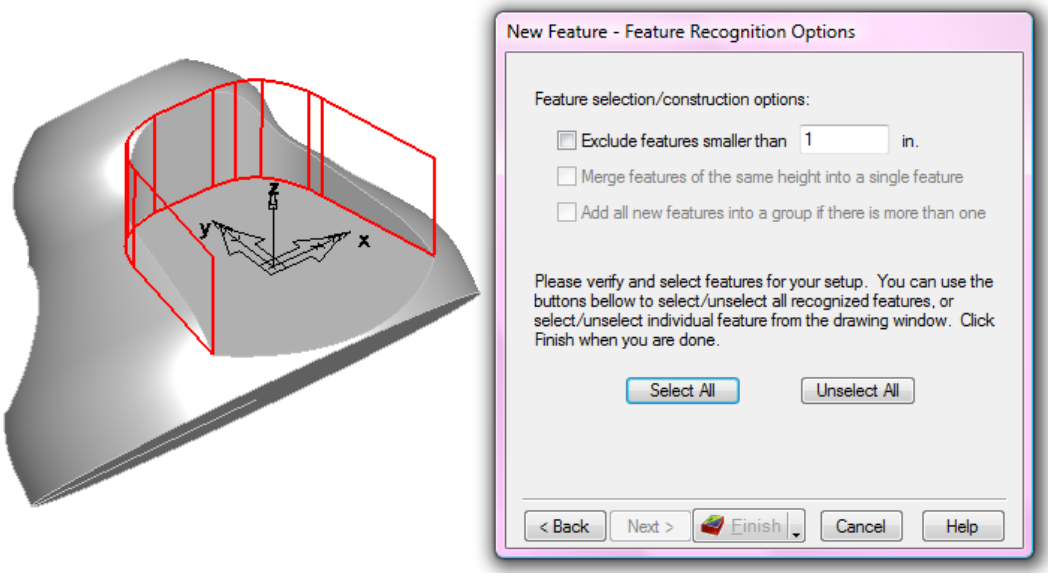


- Click **Next**

- Check **Automatic recognition**



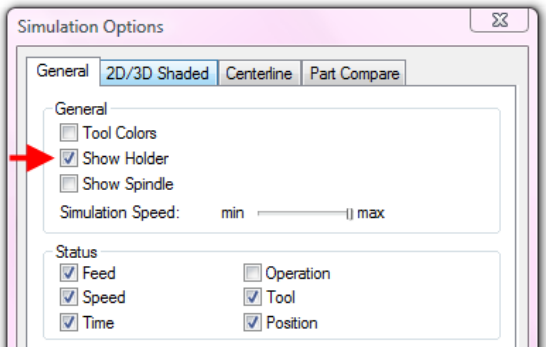
- Click **Next**
- Click **Select all** and then **Finish**



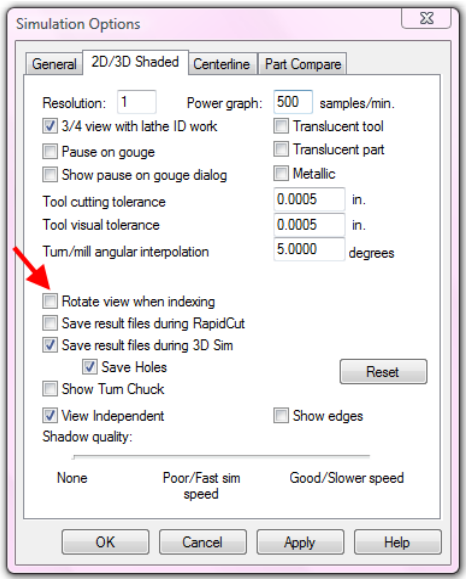
The part is now fully programmed using 3+2 machining. We will finish up by running a 3D simulation so we can see the motion of the tool around the part.

- Select **setup1**
- From the **Options** menu select **Simulation**

- On the **General tab** check **Show Holder**



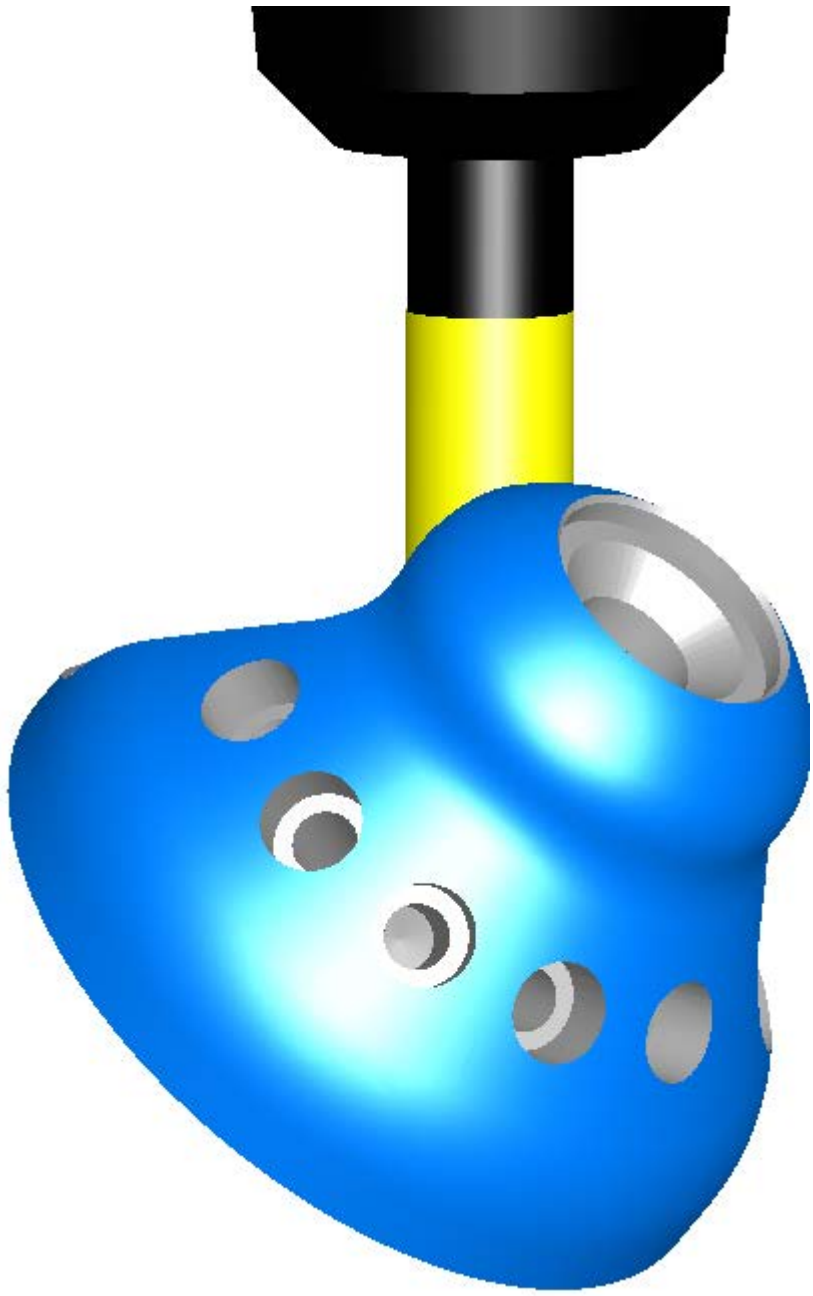
- On the **2D/3D Shaded tab** uncheck **Rotate view when indexing**



If this option is checked then the tool will stay still during the simulation and the part will move. We have unchecked it so the part will appear to stay still while the tool moves.

- Click **Apply** and then **OK**

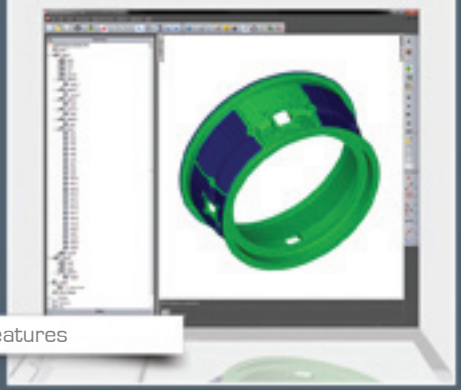
- Slow down the simulation speed
- Run a **3D simulation**



Note how the tool apparently moves around the part. depending upon the post processor and machine tool these movements will be translated into XYZ linear moves together with Head/Head, Head/Table or Table/Table rotations.

FeatureCAM 2012

FeatureCAM 2012



new features

Delcam TV

www.delcam.tv

FeatureCAM Learning Zone

www.delcam.tv/lz

FeatureCAM Website

www.featurecam.com

THE QUEEN'S AWARDS FOR ENTERPRISE: INNOVATION 2010

THE QUEEN'S AWARDS FOR ENTERPRISE: INTERNATIONAL TRADE 2011

Powering your productivity

Delcam Headquarters | Small Heath Business Park | Birmingham | B10 0HJ | UK
+44 (0)121 766 5544 | marketing@delcam.com | www.delcam.com

To contact your local reseller, visit **www.delcam.com/resellers**

© Copyright Delcam plc 2011. All other trademarks are the property of their respective owners.