Using the Menubar
Various software options can be configured using the drop down menus, positioned in the top left corner of the QuickCAM 3D window.
Click here for more...

Stage One - Select 3D or Image.
Here, you decide whether to load in a pre-designed 3D model or an image file which will use the lightest and darkest parts in the picture to create a 3D model.
Click here for more...

Stage Two - Orientate Model.
You use this screen to orientate your 3D model, so that the cutting tool will be pointing down on the face you want to machine. The 3D model can also be rotated to make best possible use of the available space in your CNC machine.
Click here for more...

Stage Three - Billet Definition.
In this stage, you enter the size of the billet. This is the block of "raw" material that you will eventually place into your CNC machine. In most cases, this will be larger than the size of your 3D model.
Click here for more...

Stage Four - Position & Size.
Both your 3D model (loaded in stage one) and your billet (defined in stage three) are shown together in this screen. You can scale and position your 3D model so it can be machined from a particular area of your billet.
Click here for more...

Stage Five- Tooling Setup.
The type of cutting tool you intend to use is defined in this stage. Options here also help define whether roughing passes will be needed and the level of detail and finish quality that can be achieved.
Click here for more...

Stage Six- Cutting Plane.
This stage allows to define how far down the 3D model you will cut. This can be useful when you want to machine left and right sides, or just a chosen area of your design.
Click here for more...

Stage Seven - Machining Strategy.
You define the feedrate and spindle speeds that will be used here. The raster cutting method used can also be defined - this is the direction the cutter zig-zags across the billet, in stepped movements, in order to machine your design.
Click here for more...

Stage Eight - 3D Simulation.
Simulating the process allows you to see exactly how your CNC machine will manufacture your design. Examine the simulated model for problem areas, then if necessary, step back to previous screens to apply new settings. This can help avoid expensive mistakes and save valuable machining time.
Click here for more...

Stage Nine - CNC File Output.

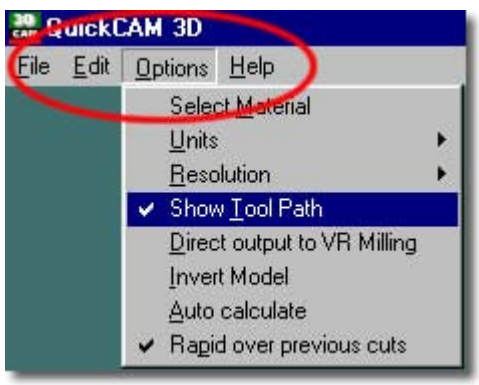
You can decide where to place the datum position (the zero point of your CNC file) - you will have to find and set the same position on the billet used in your CNC machine. Finally, create your Denford CNC file or post process your design for manufacture on a specific brand of CNC machine.
Click here for more...

Back to top of page Next >

QuickCAM 3D Menubar Options

Next >

Software menus are displayed under the titlebar, at the top of the QuickCAM 3D software window. To display a software menu, position your screen cursor over the menu title and left button click - the list of options in the chosen menu will drop down.



Availability of the options in each menu changes depending on the stage reached through the QuickCAM 3D process.

Menu - File

Open Project
Opens up a previously saved QuickCAM project. QuickCAM project files (*.Q3D) store all the settings used to create a part from an STL model, including a record of where the file is, billet size, model scale etc. Note, that the original STL model used, must still be available before re-opening a project.

Save Project
Once you are happy with the output from QuickCAM 3D, you can save all the settings you used, into a project file. QuickCAM 3D always saves the current settings for the next time you use it. However, it is useful to save project, for example, that are done for different machines or material types- where feeds and speeds will be different.

Save STL
Save your 3D model as a Stereolithograpy file (*.STL), a format commonly used by many 3D software packages, such as Pro/DESKTOP.

Auto Run

Once you have re-opened a project, you can simply click this button to automatically re-create the toolpath output - without having to go through all the seperate stages. You may find this feature useful if you have many similar models to machine with the same billet size, cutting strategy etc. In this case, you would do the first model toolpath correctly, save the project, then simply call each new model the same filename as the first (ie, overwrite the STL file each time).

Exit
Closes down the QuickCAM 3D software.

Menu - Edit

Roughing Tool
Allows you to quickly edit the roughing tool details. The roughing tool parameters are only ever used if your machining strategy means that the model cannot be cut with one pass of the tool (ie, multiple depth passes are required).

Menu - Options

Select Material
Displays the "Material Selector" window, allowing you to define the type of material used when rendering the simulation of your machined part.
Click here for help on how to select a material or import your own image maps

Units
Swap between Metric (millimetres) or Imperial (inches).
A tickmark shows which option is currently active.

Resolution
This new feature of QuickCAM 3D is to improve surface finish of detailed models. There are three resolutions available:

> **High** - For very fine / small models where detail is required. Using this option for a large model will result in longer calculation times.
> **Medium** - The default setting for most models - gives a good quality finish without the long calculation times.
> **Low** - Would only be necessary to use this on very large models, or very slow PC's. Will give a coarse finish that may need further finishing.

Show Tool Path
On Stage Seven - Machining Strategy, the screen automatically shows the path taken by the cutting tool across the billet after clicking the [Calculate...] button.
Click the option title to turn the feature on or off. A tickmark is displayed when the option is on.

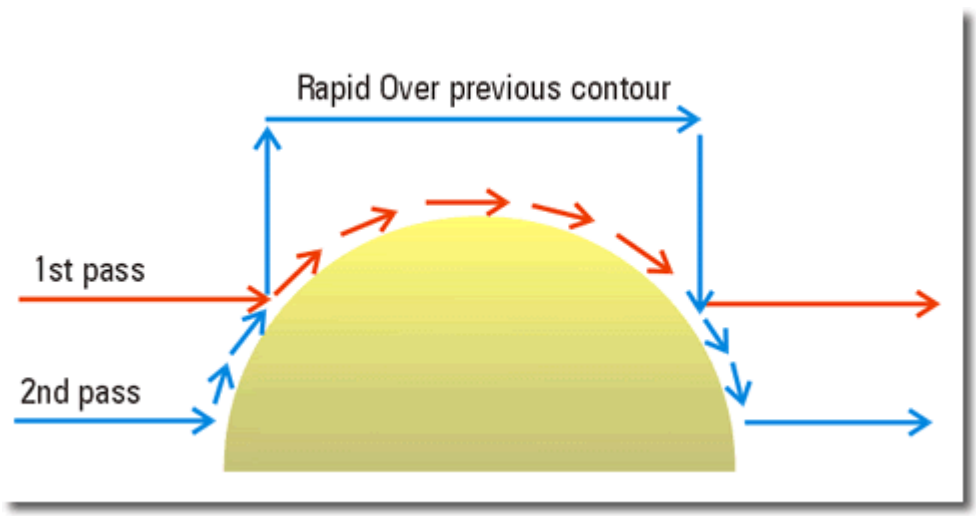
Direct Output to VR Milling
On Stage Nine - CNC File Output, Denford VR CNC Milling software (if previously installed) will automatically start and load your new CNC file. Note - Requires Denford VR CNC Milling Version 2.14 or higher. On versions lower than this only the software will start automatically.
Click the option title to turn the feature on or off. A tickmark is displayed when the option is on.

Invert Model

Available only when at the machining strategy stage. Clicking this, will invert the model and toolpaths as though you were making a mould of the model.

Auto Calculate
This option simply changes the next button on the machining strategy stage, into a calculate button when the toolpath is not known. It is then changed back into a Next button.
Used by advanced users that know the settings are correct, and quickly want to progress to the final stages

Rapid over previous cuts
If this option is selected, and your machining strategy requires some roughing passes to be made, then the toolpath generated will automatically cause the machine to rapid over parts of the model that have already been cut (by the previous passes). The final finishing pass will still run over the whole of the contour:



Menu - Help

Contents
Opens the QuickCAM 3D helpfile.
This helpfile is context sensitive - pressing your [F1] function key will automatically open the helpfile on the page corresponding to the software stage, window or menu being used.

About
Displays QuickCAM 3D version number and product details.

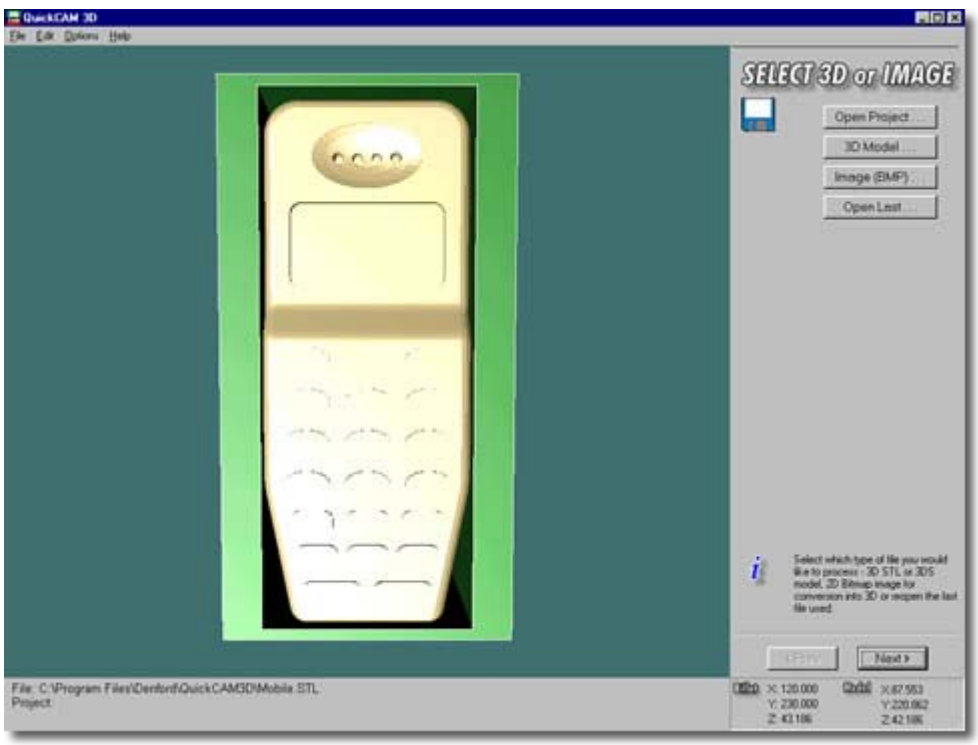
Click here for an quick overview of each stage of the process

Back to top of page Next >

QuickCAM 3D
Stage One - Select 3D or Image

Overview

Here, you decide whether to load in a pre-designed 3D model (*.STL or *.3DS formats which most CAD systems will export) or an image file (including the popular *.BMP and *.JPG formats) which will use the lightest and darkest parts in the picture to create a 3D model.



Opening a project

Click Open Project . . . to quickly open a previously saved project.

QuickCAM project files (*.Q3D) store all the settings used to create a part from an STL model, including a record of where the file is, billet size, model scale etc.
Note, that the original STL model used, must still be available before re-opening a project.

Opening a 3D Model

Click 3D Model . . . to open a previously saved 3D image. Formats available include Stereolithograpy files (*.STL) most commonly used with Pro/DESKTOP CAD software and 3D Studio (*.3DS) files.

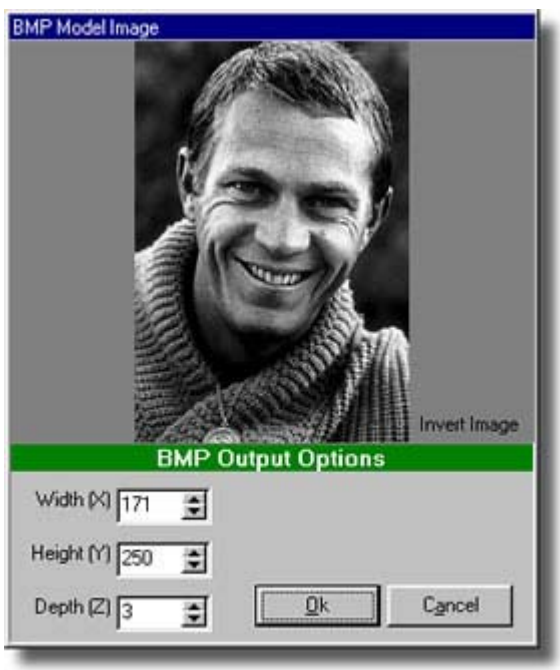
3D models will open with their top face pointing directly towards you. The top face of your 3D model is defined in your CAD software package. If your model opens and displays the side or bottom of your model,

don't worry - we can orientate the view in the next stage.

Opening a 2D Image

Click Image (BMP)... to open a previously saved 2D image. Formats available include Joint Photographic Expert Group images (*.JPEG or *.JPG), Device Independent Bitmaps (*.DIB), Bitmaps (*.BMP), Icons (*.ICO), Enhanced Metafiles (*.EMF) and Windows Metafiles (*.WMF).

When you open a 2D image, QuickCAM 3D will automatically apply a depth value to your image, effectively turning it into a 3D model. The lightest tones in your image are assigned with the greatest depth of cut, whilst the darkest tones are assigned with the least depth of cut (zero).



When you open an image, a dialogue window is displayed allowing you to define:

1. The width (X) and height (Y) of the image model, stated in millimetres or inches, depending on your "Options|Units" setting. X and Y are linked - i.e. if you alter one value, the other will automatically change to keep the photo in proportion. Change the value that best helps you fit the image to your proposed design or workpiece.
2. The depth (Z) of the image model, stated in millimetres or inches, depending on your "Options|Units" setting. This value is depth of cut not the thickness of your workpiece. In the example shown above, this value has been set to "3mm" - the CNC file we produce could be used with 4mm thickness polystyrene, to produce a 3D photograph.
3. The [Invert Image] button allows you to reverse the tones, so you can use photographic negatives to produce a 3D model.

Opening the last file used

Click Open Last... to open the file you last used in QuickCAM 3D.

Moving between screens

Click < Prev to move back to the last QuickCAM 3D stage viewed.

Click Next > to move forward to the next stage in the QuickCAM 3D process.

[Click here for an quick overview of each stage of the process]

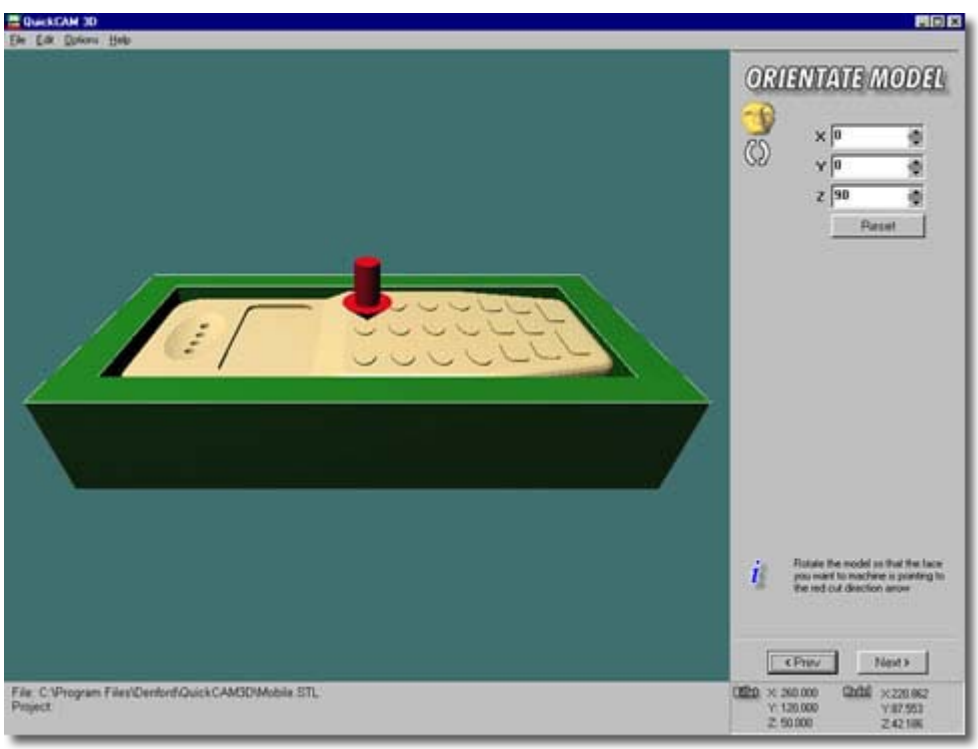
Back to top of page < Previous - Next >

QuickCAM 3D
Stage Two - Orientate Model

< Previous - Next >

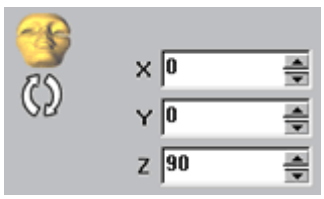
Overview

You use this screen to orientate your 3D model, so that the cutting tool will be pointing down on the face you want to machine. The 3D model can also be rotated to make best possible use of the available space in your CNC machine.



Configuring the 3D model face to be cut

A large red arrow will appear, pointing to one of the faces on your 3D model. This arrow indicates the cutting direction (the Z axis of the CNC machine). You must orientate this red arrow so it points directly at the face you want to cut.



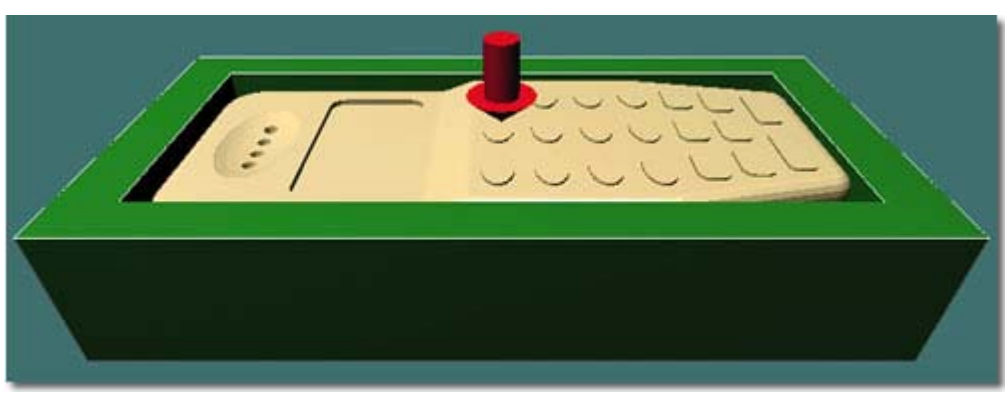
The red arrow can be moved around the faces using the [Up] and [Down] nudge buttons, positioned on the far right of each X, Y, Z readout panel. Each click of a nudge button will step the red arrow 90 degrees across the selected axis.

If you need to position the red arrow towards an angled face, you can manually type an angle value into the required X, Y or Z readout box.

Configuring the 3D model to fit in your CNC machine

When the red arrow is pointing directly at the face you want to cut, you can orientate the 3D model so it makes the most efficient use of the space in your CNC machine.

The best way of achieving this is by adjusting the view of the 3D model, so you can see the top of the face you want to cut and also the end or side face. On your screen, the red arrow should be pointing down - imagine this to be the cutting tool.
Click here for help on how to adjust the view of the 3D model



Using the [Up] and [Down] nudge buttons, positioned on the far right of each X, Y, Z readout panel, rotate the 3D model in 90º steps to best fit inside your CNC machine - imagine the view you are seeing is the view through the front window of your CNC machine. Usually, you want to align the longest side of the 3D model with the longest axis of your CNC machine (most commonly the X axis).

Moving between screens

Click < Prev to move back to the last QuickCAM 3D stage viewed.

Click Next > to move forward to the next stage in the QuickCAM 3D process.

[Click here for an quick overview of each stage of the process]

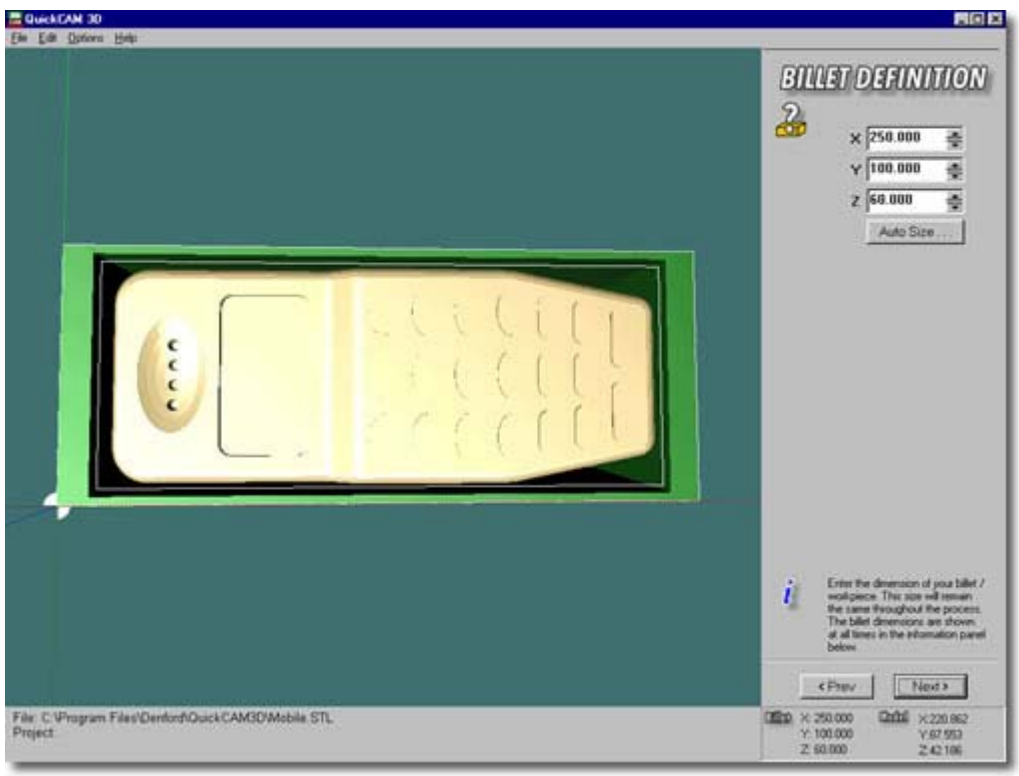
Back to top of page < Previous - Next >

QuickCAM 3D
Stage Three - Billet Definition

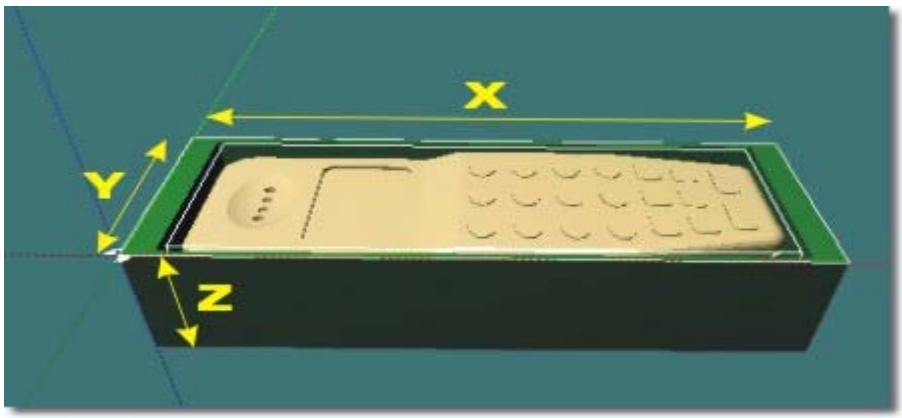
< Previous - Next >

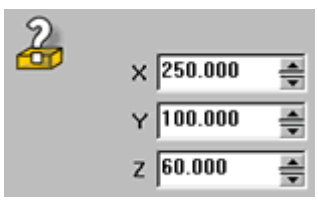
Overview

In this stage, you enter the size of the billet. This is the block of "raw" material that you will eventually place into your CNC machine. In most cases, this will be larger than the size of your 3D model.

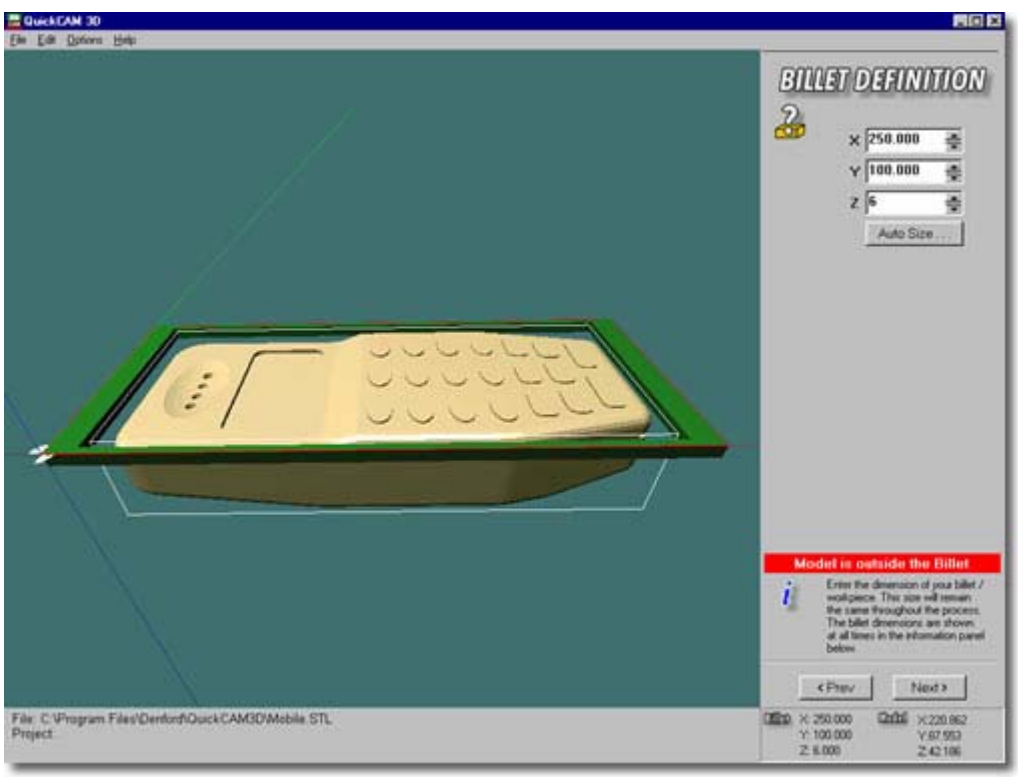


Entering the dimensions of your billet





You enter the dimensions of your billet using the X, Y, Z readout panel, in the top right corner of the screen. Click in each readout box and enter the required dimension value. The billet dimensions are stated in millimetres or inches, depending on your "Options|Units" setting. Alternatively, clicking [Up] and [Down] nudge buttons, positioned on the far right of each X, Y, Z readout box, steps the selected dimension up or down by one unit.

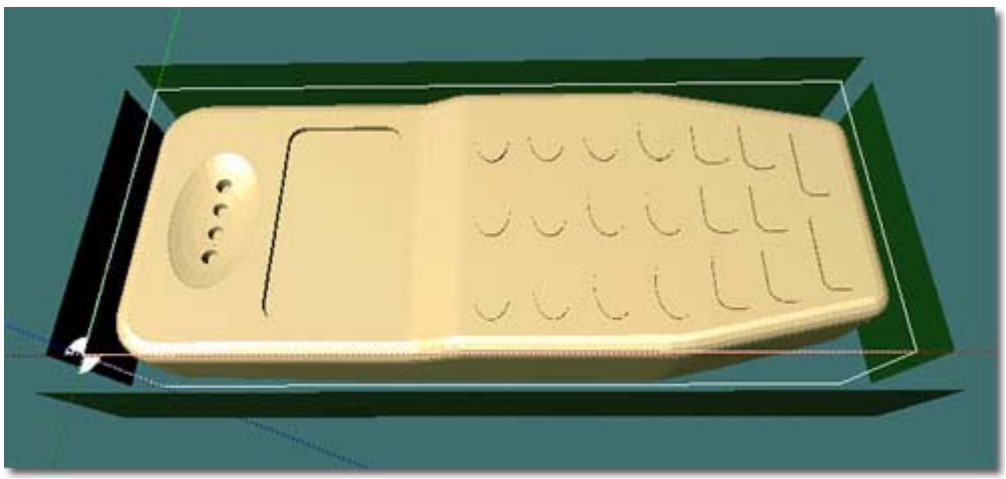


As each dimension is entered, the green block representing the billet will be updated on the view of the 3D model. Notice that a warning message appears whenever you enter a dimension that is smaller than 3D model size.

Note the white lines define the boundary edges of the 3D model itself.

Don't worry if the 3D model is positioned in the wrong area of your billet - this can be changed in the next stage.

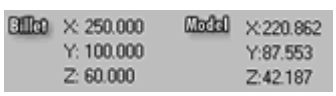
Autosizing the billet



Clicking the Auto Size . . . button will automatically create a billet size identical to the boundary size of your 3D model. You can still use the [Up] and [Down] nudge buttons to change these billet dimensions.

NB, you will notice that the side walls of the billet in this image, are set out away from the white bounds of the model. This is because a border size has been entered (see cutting plane for more details) The border size, in effect, makes a clearance area around the model.

Billet & 3D model dimensions readout display



Note that for future reference, the dimensions of both the billet and the 3D model are shown on the readout panel, located in the bottom right corner of the QuickCAM 3D software window.

Moving between screens

Click < Prev to move back to the last QuickCAM 3D stage viewed.

Click Next > to move forward to the next stage in the QuickCAM 3D process.

Click here for an quick overview of each stage of the process

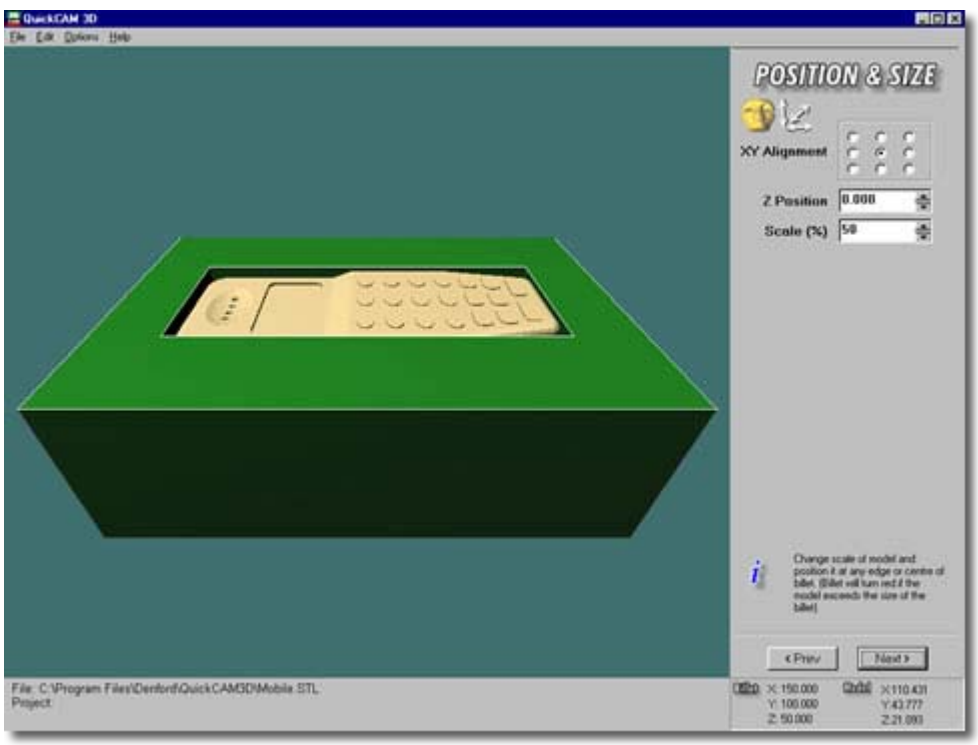
Back to top of page

< Previous - Next >

QuickCAM 3D
Stage Four - Position & Size

Overview

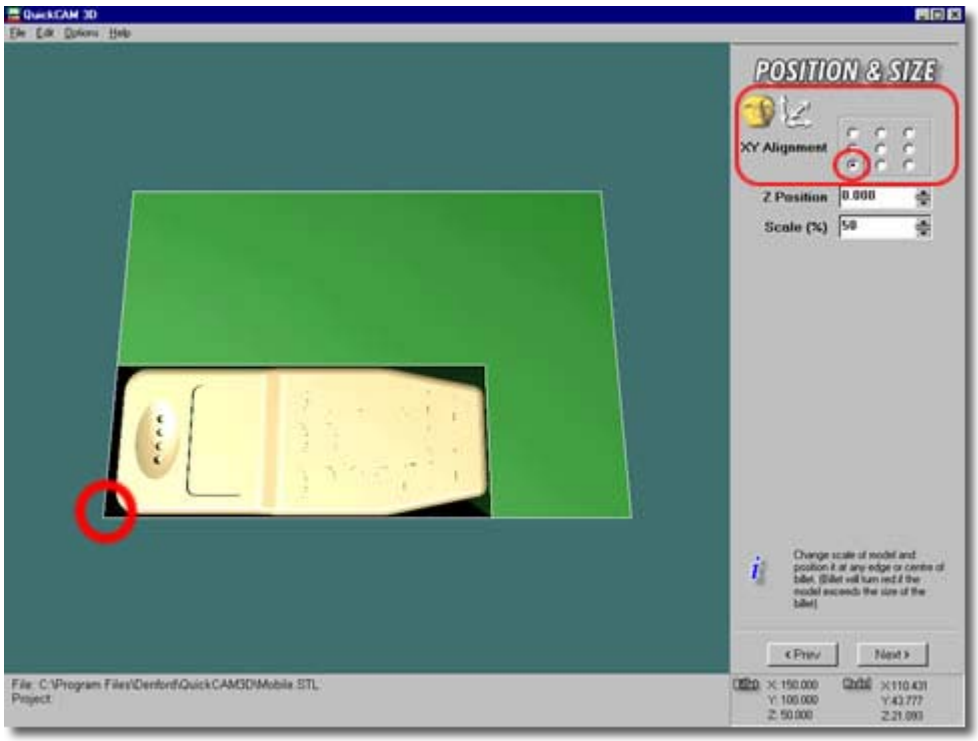
Both your 3D model (loaded in stage one) and your billet (defined in stage three) are shown together in this screen. You can scale and position your 3D model so it can be machined inside a particular area of your billet.



Setting the X and Y alignment of the model

The XY alignment panel, circled red , allows you some control over where your model is positioned within the top plane - the X and Y edges of the billet.

The model can be positioned in nine predefined locations, indicated by the white circular markers on the XY Alignment panel. The bottom row aligns with the front X edge of the billet, the top row with the back X edge of the billet, the right row aligns with the right Y edge of the billet and the left row aligns with the left Y edge of the billet. By default, the model is positioned within the centre of the top plane - indicated by the black dot.

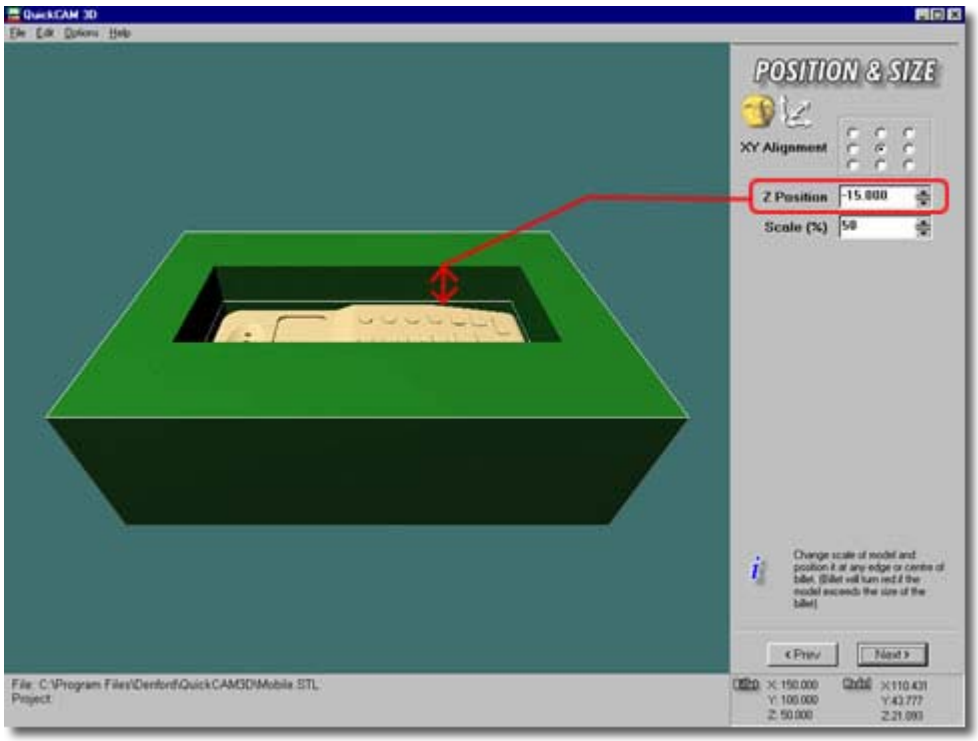


To reposition the model in the billet, click inside the appropriate circular white marker. The black dot will transfer to the chosen marker and the model will shift position within the billet. In the example above, the model has been shifted to align with the front X and left Y edges - the front, left corner of the billet.

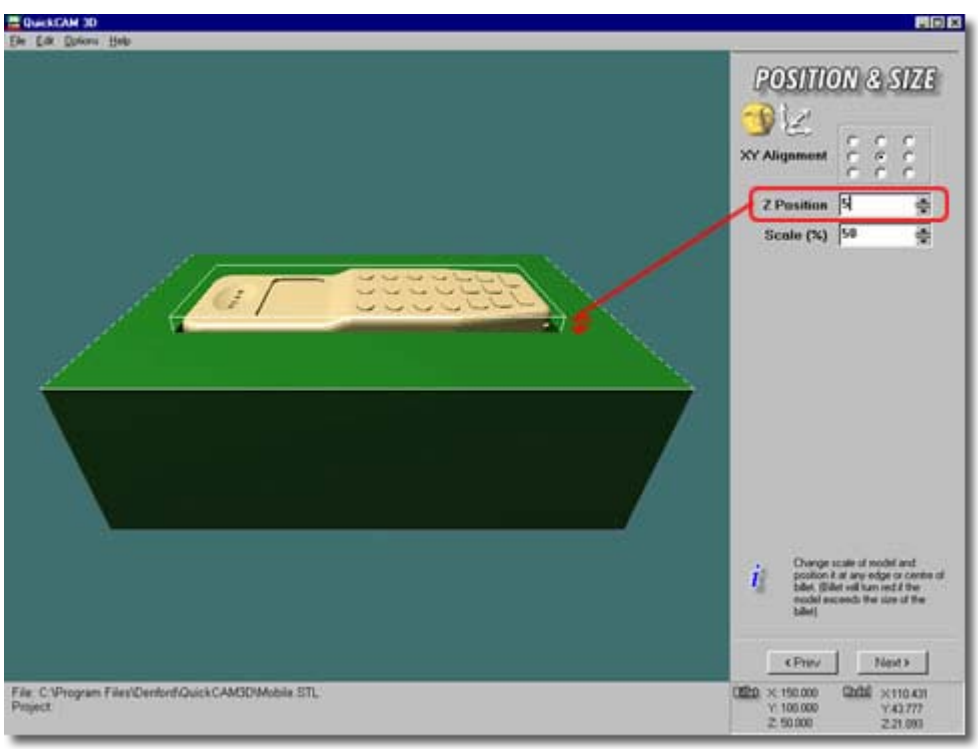
Setting the Z alignment of the model

The Z position panel allows you to define how high your model will be positioned inside the billet. By default, this value is set to zero - this corresponds to the top surface of the billet. In this situation, the top surface of your model is usually in line with the top surface of the billet.

Here the Z Position has been set to -15.0 mm:

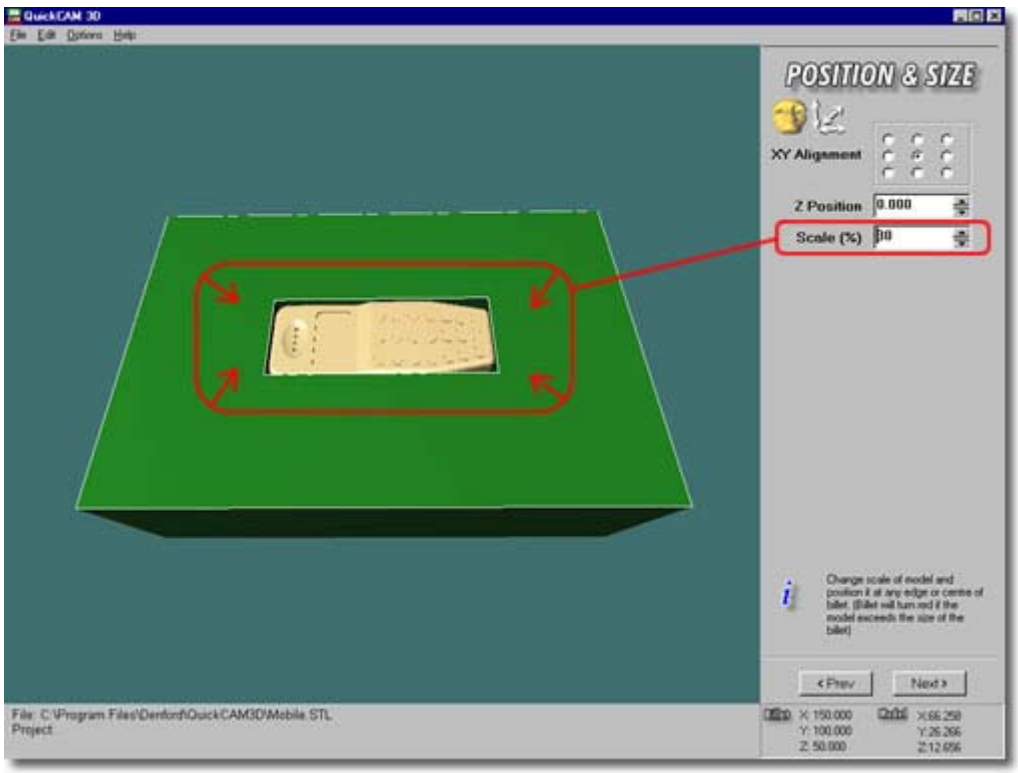


The model is raised or lowered in the billet by over typing with the required value, or by using the [up or down arrow] nudge buttons. As the value changes, the model position will update in the main graphic panel. Positive values raise the model whilst negative values lower the model.



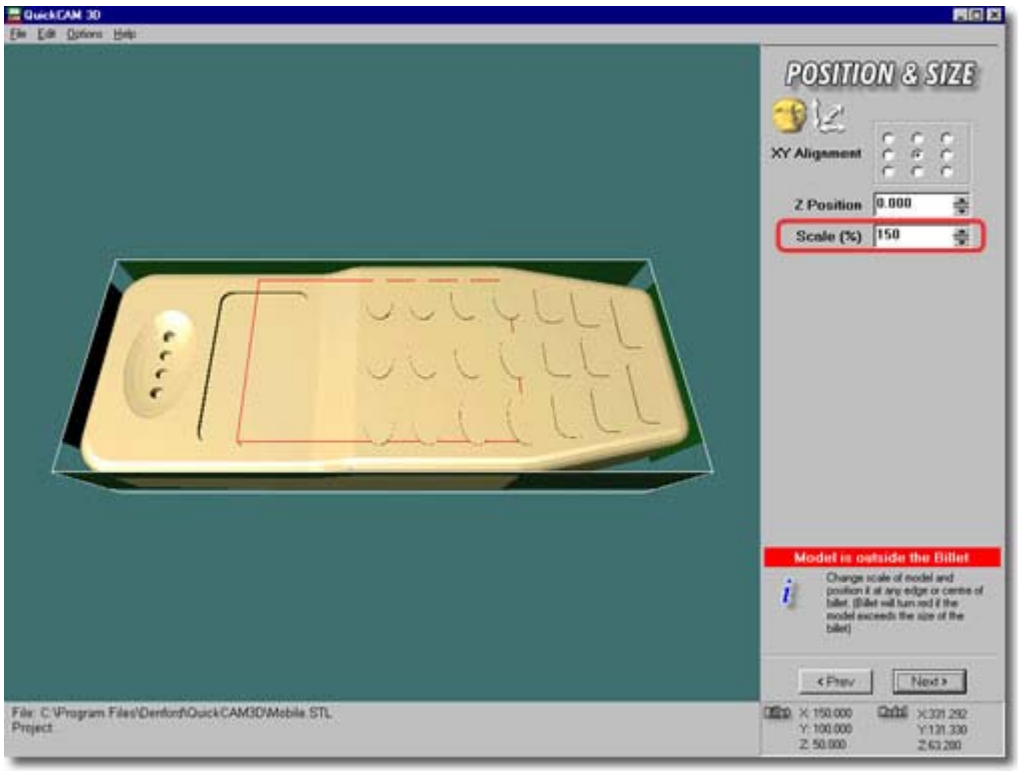
Parts of the model not contained within the boundary of your billet are highlighted. In the example above, the control panel and buttons on the mobile phone model are positioned too high - they are breaking through the upper boundary of the billet.

Setting the scale of the model



Use the Scale (%) panel to proportionally increase or decrease the size of your model within the billet. In the example above, the model has been scaled to 30% the original size.

You can set your scale by over typing with the required value, or by using the [up or down arrow] nudge buttons. 100% scale uses your original model dimensions. The model in the main graphic panel will update according to the scale value entered.



When increasing your model scale note that a warning panel will be displayed when your model size exceeds the size of your billet. In the example above, the ends of the model protrude from the billet.

Click here for an quick overview of each stage of the process

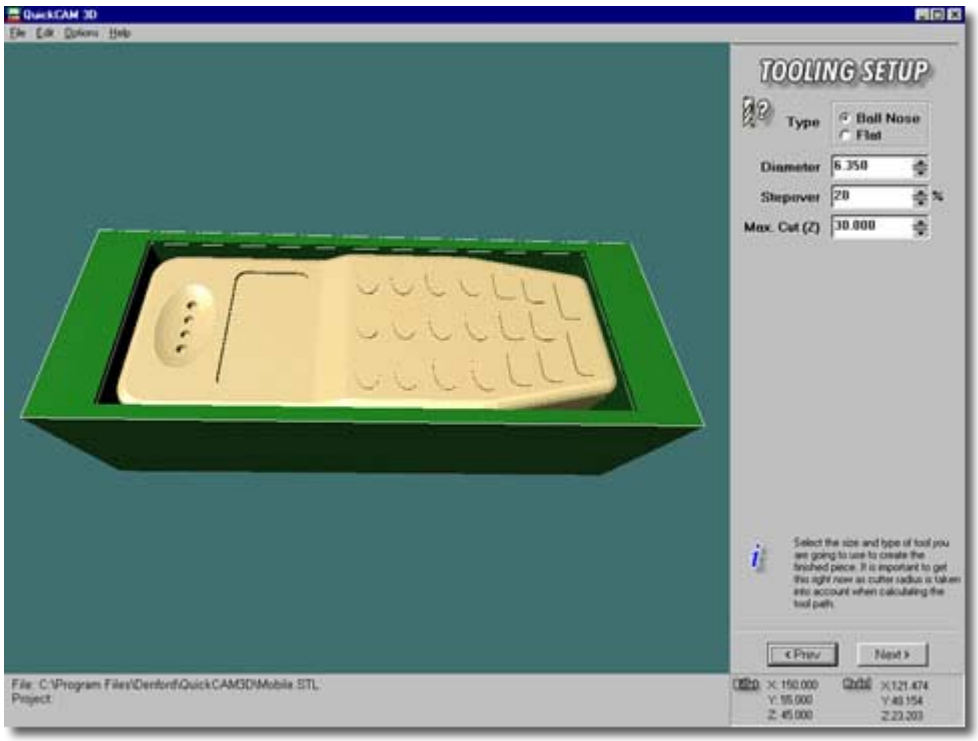
Back to top of page < Previous - Next >

QuickCAM 3D
Stage Five - Tooling

< Previous - Next >

Overview

The type of cutting tool you intend to use is defined in this stage. Options here also help define whether roughing passes will be needed and the level of detail and finish quality that can be achieved.



Tooling Parameter: Type

The 'Type' panel allows you to choose either a ball nose or a flat (slot) cutter. This can influence both the shape and accuracy of your final model.

In the example shown at the top of this page, we have chosen to use a ball nose cutter - the ball nose cutter is ideal for machining the main body of the phone but as a trade off, a small fillet will be added to all the buttons, at the point where they meet the main top surface of the phone casing. To obtain a crisp line at this point, we would have to use a flat cutter.

Click in the appropriate circular white marker to select the chosen option. The current option is shown using a black dot.

Tooling Parameter: Diameter

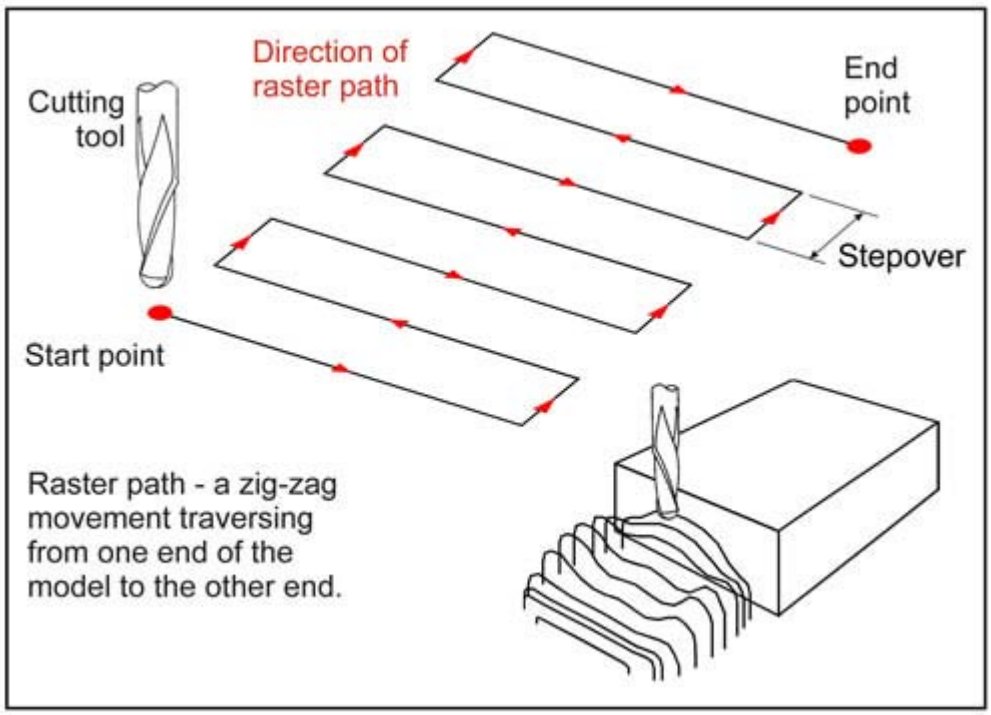
Enter the diameter of the cutting tool in the 'Diameter' panel.

Remember that the diameter value you enter will depend on the units being used in the QuickCAM 3D software. In the example shown at the top of this page, the software is set to metric (millimetres) but we want to use a 1/4" ball nose cutter. We must therefore enter the diameter value as 6.35, not 0.25.

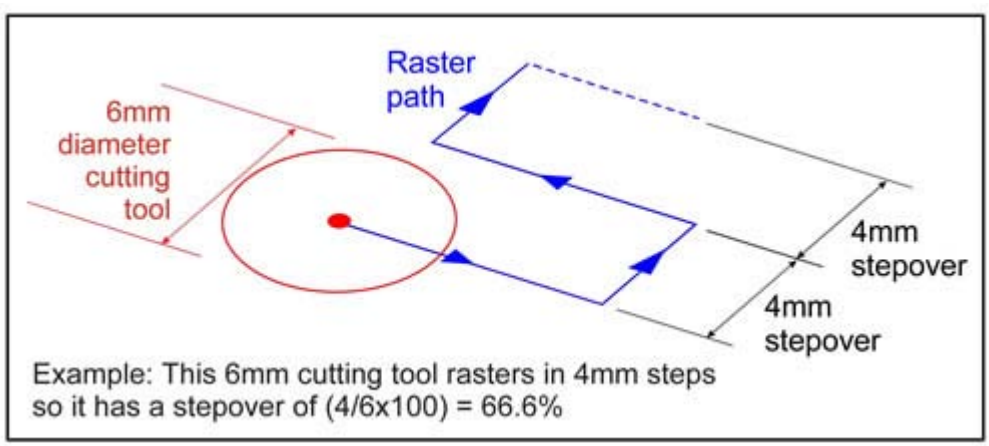
The current units of measurement being used can be checked by clicking the "Option|Units" menu. This option can be changed by over typing with the required value, or by using the [up or down arrow] nudge buttons.

Tooling Parameter: Stepover

Your 3D model is produced using a raster path - a series of parallel zig-zag lines that sweep across the surface of the billet, as shown in the diagram below.



The value entered in the 'Stepover' panel defines how much the tool moves before starting the next line on the raster path, or zig-zag. This value is calculated as a percentage of the diameter of the tool, as shown in the diagram below.



For the example shown in the screenshot at the top of this page, a 20% stepover on a 6.35mm (1/4") ball nose would be...
6.35mm x 20/100 = 1.27mm.
In other words, the tool would cut down the first raster zig-zag line, then move 1.27mm along before cutting back along the second zig-zag line.

Generally, choosing a large stepover will allow the model to be machined very quickly, at the expense of detail and surface finish. Choosing small stepover will allow the model to be machined with a high degree of accuracy but at the expense of long machining times. A good compromise is around the 25% mark.

The current units of measurement being used can be checked by clicking the "Option|Units" menu. This option can be changed by over typing with the required value, or by using the [up or down arrow] nudge buttons.

Tooling Parameter: Max. Cut (Z)

The 'Max. Cut (Z)' panel defines the maximum depth of cut that can be achieved by the tool in one pass. This will depend upon the material you intend to machine and in the case of softer materials, the flute length of the tool.

The current units of measurement being used can be checked by clicking the "Option|Units" menu. This option can be changed by over typing with the required value, or by using the [up or down arrow] nudge buttons.

Click here for an quick overview of each stage of the process

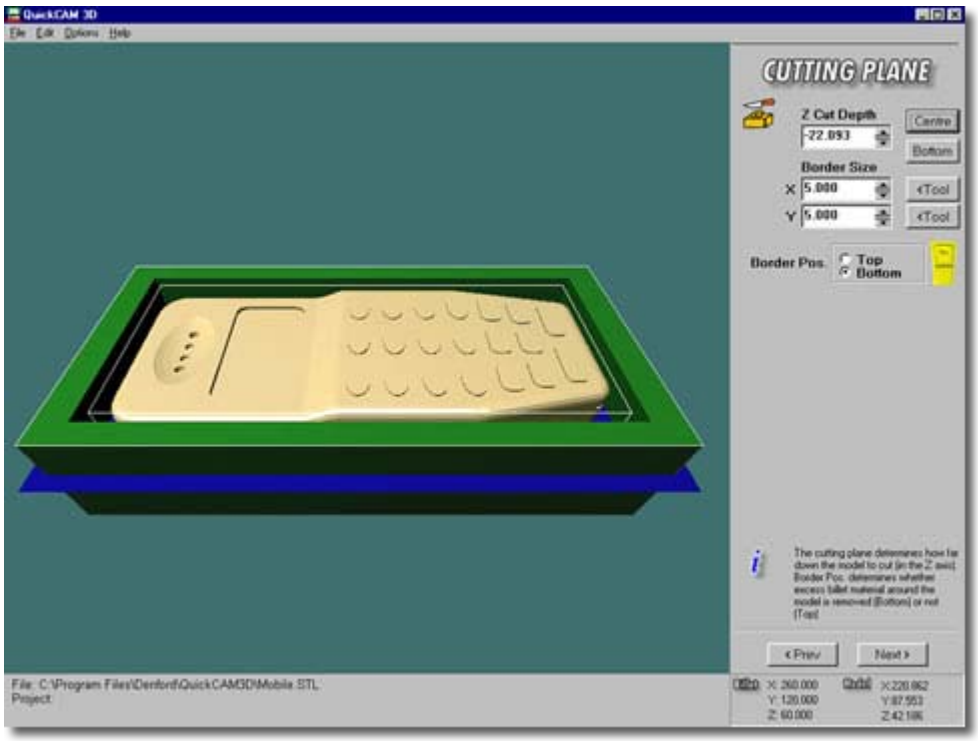
Back to top of page < Previous - Next >

QuickCAM 3D
Stage Six - Cutting Plane

< Previous - Next >

Overview

This stage allows to define how far down the 3D model you will cut. This can be useful when you want to machine left and right sides, or just a chosen area of your design.



The cutting plane, shown by the blue rectangular slice through the billet and model, defines the maximum distance you want to cut into the billet. Any part of the 3D model below this cutting plane will be ignored when your CNC file is written. In other words, all the parts of your model below this cutting plane will not be machined.

In the example shown above, only the top half (the front) of the mobile phone will be cut, so the cutting plane has been positioned through the centre of the 3D model. Since the phone is asymmetrical, it will be easier and quicker to machine a "front" and a "back", then glue both halves together to form the prototype model.

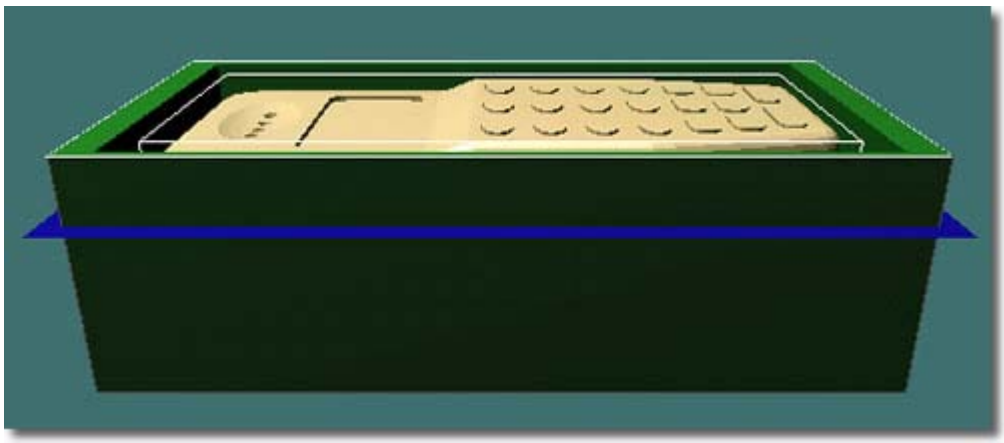
The exact position of the cutting plane is shown in the "Z position" readout panel. The "Z position" readout is relative to the top surface of the billet, which is automatically configured as the "zero" plane. In other words, when the cutting plane is positioned below the top surface of the billet, any values will be negative.

When you first move to the "Cutting Plane" screen, the cutting plane is positioned through the exact centre of the 3D model, with the bottom border position active.

Setting the position of the cutting plane

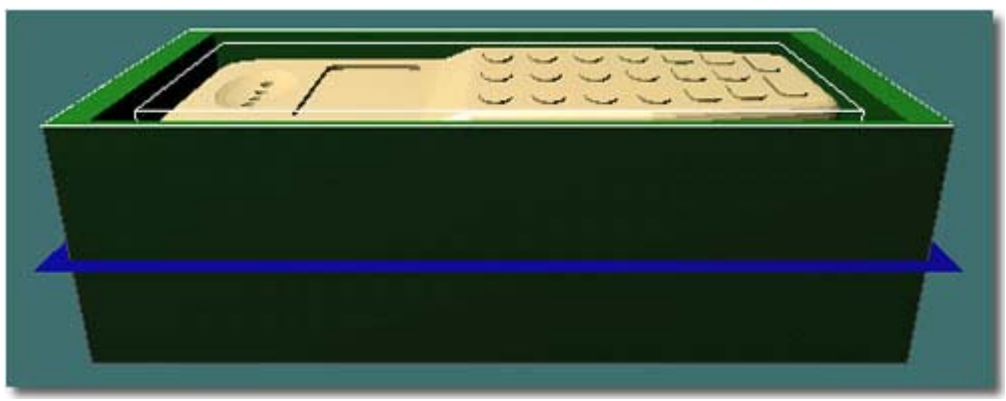
Clicking 

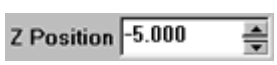
will position the cutting plane exactly through the centre of the 3D model, as shown below. Notice that this operation applies to the centre of the model, not the billet.



Clicking 

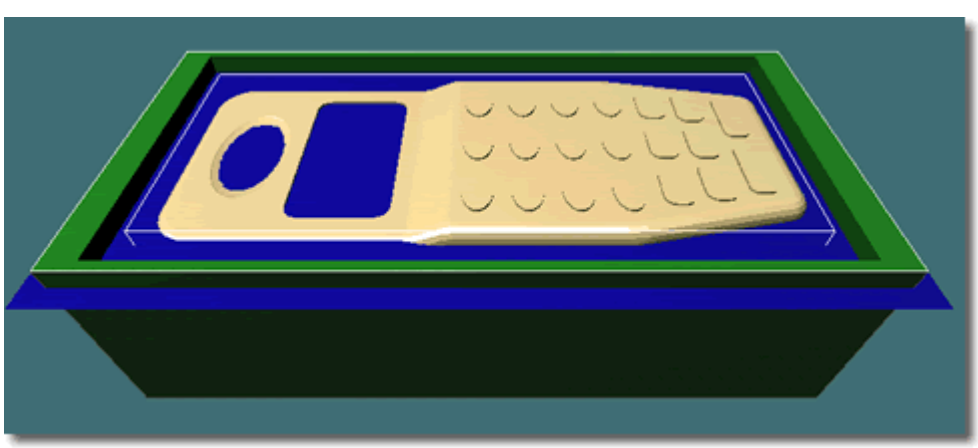
will position the cutting plane exactly in line with the bottom surface of your 3D model, as shown below. Notice that this operation applies to the bottom of the model, not the billet.



The 

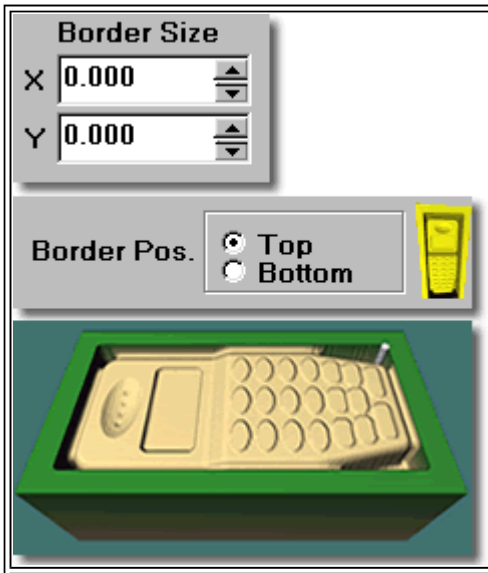
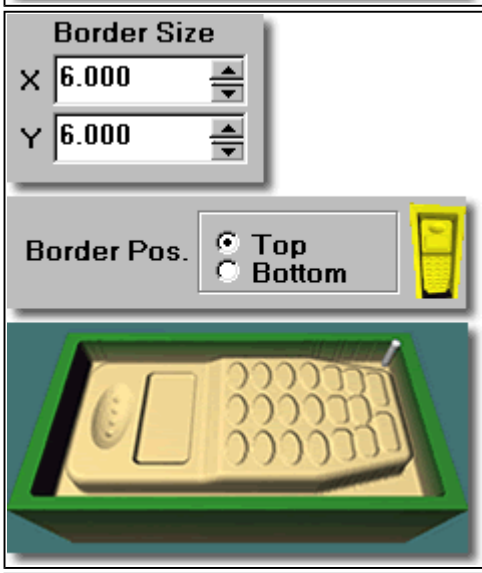
panel allows you to manually define the position of the cutting plane, either by over typing with the required value, or by using the [up or down arrow] nudge buttons. As the value changes, the position of the cutting plane will update in the main graphic panel, as shown below.

Remember that the Model data panel, located in the bottom right corner of your screen, contains all the dimensions of your 3D model. This information, particularly the Z value, may be useful if you've previously lowered the model using the Position screen (in Stage Four), or if you want to configure the cutting plane in line with a specific feature on your model.

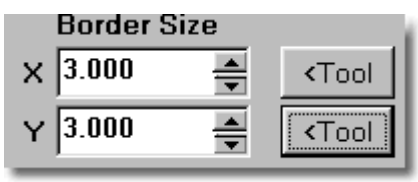


Setting border position & border clearance size

Border Pos. (Position) defines what happens to areas of the billet surrounding the model. Click in the appropriate circular white marker to select the chosen option. The current option is shown using a black dot. Also shown is the effect of changing Border Size values. This border size is the clearance amount between the model and the billet, to allow machining to occur around the model, without having to machine the whole surface of your billet.

	Here the border is set to the top with no border size (clearance). Because there is no border clearance, the tool is unable to cut all the way around the model.
	Here the border is set to the top with 6mm border size (clearance) in both X and Y axes. Because the tool diameter is 6mm, it is now able to clear all around the mobile phone model.
	If the border position is set to the bottom, the border size (clearance) has no effect, as the whole of the billet surface is cleared anyway.

When setting border size, you may quickly press either of the **<Tool** buttons. The current tool radius will then be entered into the appropriate editor.



Click here for an quick overview of each stage of the process

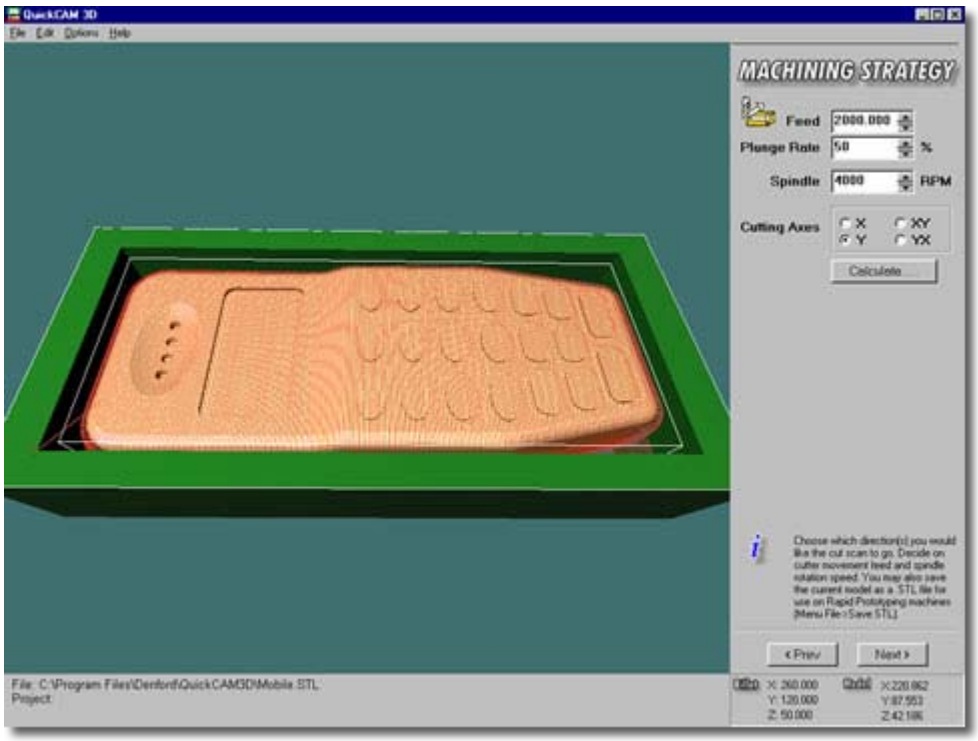
Back to top of page
< Previous - Next >

QuickCAM 3D
Stage Seven - Machining Strategy

< Previous - Next >

Overview

You define the feedrate and spindle speeds that will be used here. The raster cutting method used can also be defined - this is the direction the cutter zig-zags across the billet, in stepped movements, in order to machine your design.



Machining Strategy Parameter: Feed

The 'Feed' panel allows you to enter a feedrate appropriate to the type/size of cutting tool and type of material being cut. These values are commonly available from your local tool and billet material suppliers.

This option can be changed by over typing with the required value, or by using the [up or down arrow] nudge buttons. Metric feedrates must be entered in mm/min. Imperial feedrates must be entered in inches/min. The current units of measurement being used can be checked by clicking the "Option|Units" menu.

A list of feedrates when using 2 - 6mm diameter cutting tools with common materials on Denford CNC machines can be accessed by clicking here.

If you want to calculate your own spindle speeds, click here for more information

If you want to calculate your own feedrates, click here for more information

Machining Strategy Parameter: Plunge Rate %

When cutting harder materials such as aluminium or hardwood, it is often required to cut down (plunge) into the material at a slower rate than it is possible to cut normally. This very much depends on the type of tool used, material and speeds/feeds selected.

For this reason, the plunge rate parameter has been introduced to give the user more control of machining conditions. The plunge rate is used on two occasions:-

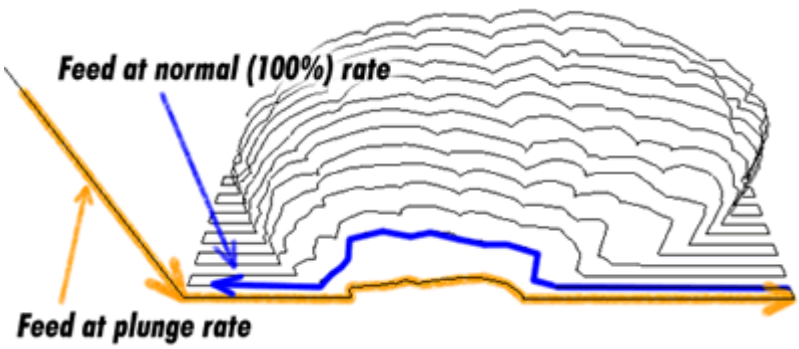
1. Whenever the toolpath requires the Z axis to feed down into the billet.

2. On the very first raster scan of any part.

For example, if you had entered **2000**mm/min into the feedrate parameter (see above), and enter **40**% into the plunge rate, then any plunge moves would be set to run at the following feedrate:

2000*(**40**/100)=800mm/min

This picture shows when plunge rate is active, after the first pass, feedrate continues at the normal speed for the rest of the raster scan:



Machining Strategy Parameter: Spindle

Enter a spindle speed appropriate to the type/size of cutting tool and type of material being cut into the 'Spindle' panel. These values are commonly available from your local tool and billet material suppliers.

This option can be changed by over typing with the required value, or by using the [up or down arrow] nudge buttons. Spindle speeds must be entered in RPM (rotations per minute).

A list of spindle speeds when using 2 - 6mm diameter cutting tools with common materials on Denford CNC machines can be accessed by clicking here.

Machining Strategy Parameter: Cutting Axes

The 'Cutting Axes' panel allows four different types of raster tool path to be selected:

1. X will produce a raster tool path that only cuts parallel to the X axis.
2. XY will produce a raster tool path that cuts parallel to the X axis first, followed by a second raster tool path that cuts parallel to the Y axis.
3. Y will produce a raster tool path that only cuts parallel to the Y axis.
4. YX will produce a raster tool path that cuts parallel to the Y axis first, followed by a second raster tool path that cuts parallel to the X axis.

Click in the appropriate circular white marker to select the chosen option. The current option is shown using a black dot.

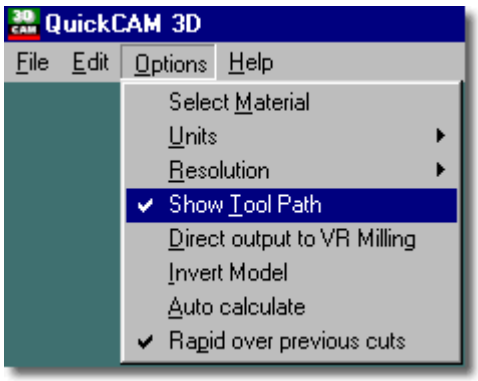
Calculating the Machining Strategy

Clicking the Calculate . . . button will calculate the tool path according the options configured in this screen

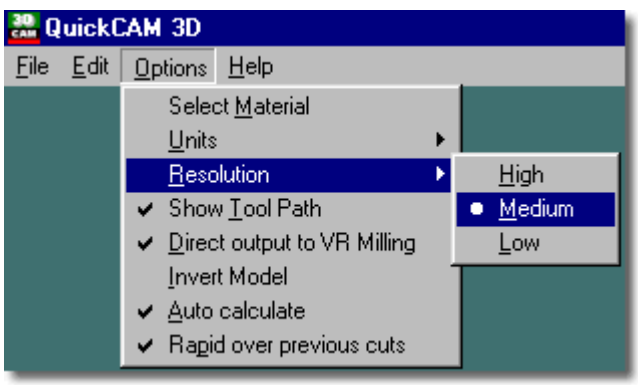
and the previous stages of the process.

Displaying the Raster Tool Path

The raster tool path is depicted on-screen using red lines, as shown in the example at the top of this page.



For most types of model, it is best to select medium resolution (default) for the calculation. This option gives a good finish to the calculated toolpath, without taking too long to process. However, you may find you want a better quality finish on the finished part. For this you would select **High** resolution. Select **Low** for very large models, or ones that take a long time to calculate:



NB, changing the resolution setting will automatically re-calculate the toolpath for you if you are in this stage of the process.

In the "Options" menu, if "Show Tool Path" is checked on (shown by a tickmark symbol) the raster tool path will be plotted on-screen as it is calculated.

If this option is checked off, the raster path will appear when the computer has finished calculating the raster tool path.

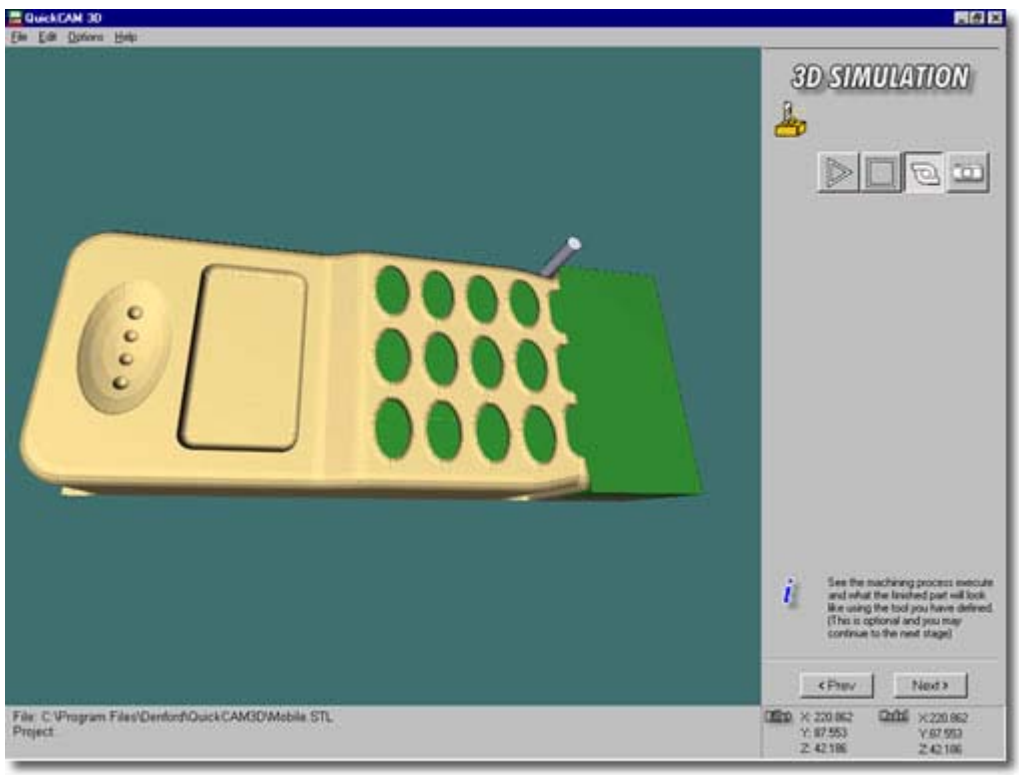
Click here for an quick overview of each stage of the process

Back to top of page < Previous - Next >

QuickCAM 3D
Stage Eight - 3D Simulation

Overview

Simulating the process allows you to see exactly how your CNC machine will manufacture your design. Examine the simulated model for problem areas, then if necessary, step back to previous screens to apply new settings. This can help avoid expensive mistakes and save valuable machining time.



Playing the 3D Simulation

Click the [Triangle] button, first from the left, to begin playing the machining simulation.

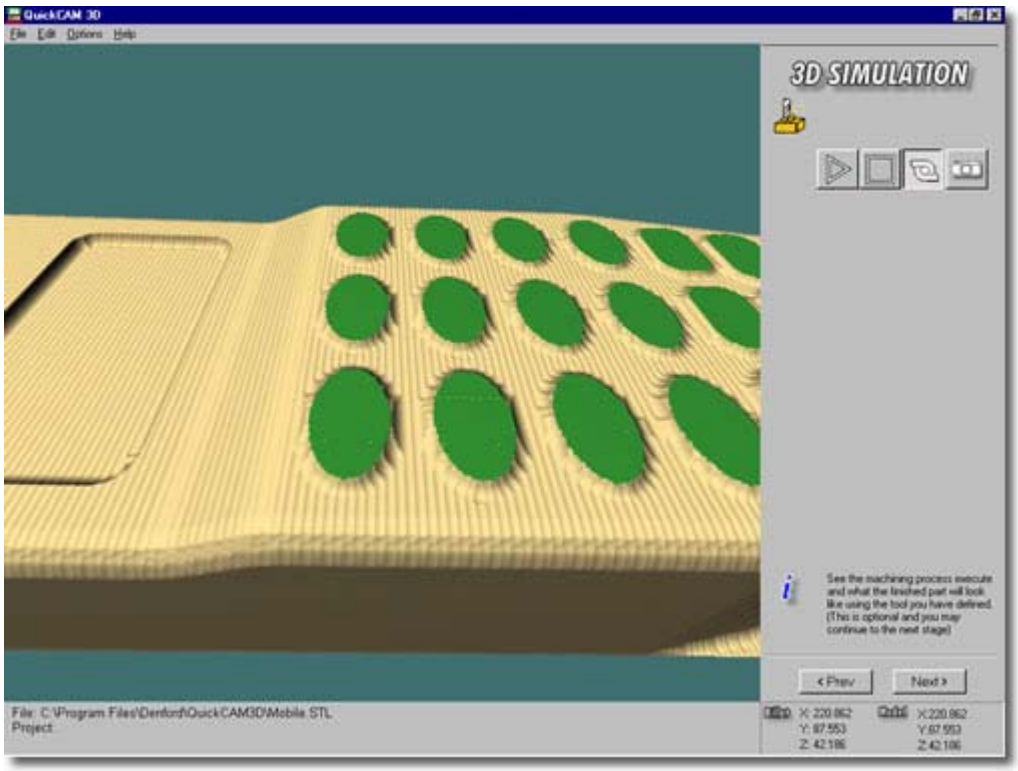
Click the [Square] button, second from the left, to stop the machining simulation.

Click the [Turbo] button, third from the left, to increase the speed of the 3D simulation.

A detailed graphic depicting your simulated part is displayed at the end of the sequence, as shown above.

Using the results from the 3D Simulation

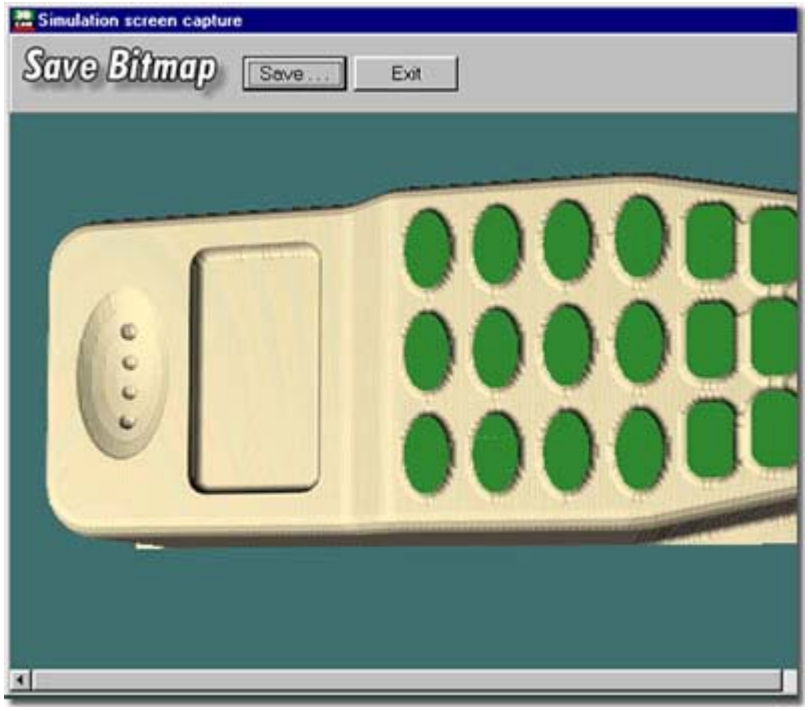
Remember, the 3D simulation provides an exact representation of what you would be cut from a billet on a real CNC machine. So if the simulated model has a rough surface finish, your real machined model will have the same rough finish!



You can zoom into the model to view specific parts in detail. In the example shown above, although the buttons are fairly well defined we could trace back to previous stages, change the tool stepover to a smaller (finer) percentage and run the simulation again to help remove the rounded edges. The green colours show areas of the billet that have not been machined.

Saving a Snapshot of the 3D Simulation

To capture a snapshot of your simulated part, click the [Camera] button, forth from the left, then save the file in an appropriate folder. Snapshots are saved in Bitmap (*.BMP) format.



Snapshots are useful for showing how your final model may appear in a variety of different materials, or showing how your machining strategies have developed. For example, if you are unhappy with the surface finish, you could take a snapshot, then change some of the parameters in previous screens and resimulate the model, building up a portfolio to explain how you overcame these problems.

Click here for an quick overview of each stage of the process

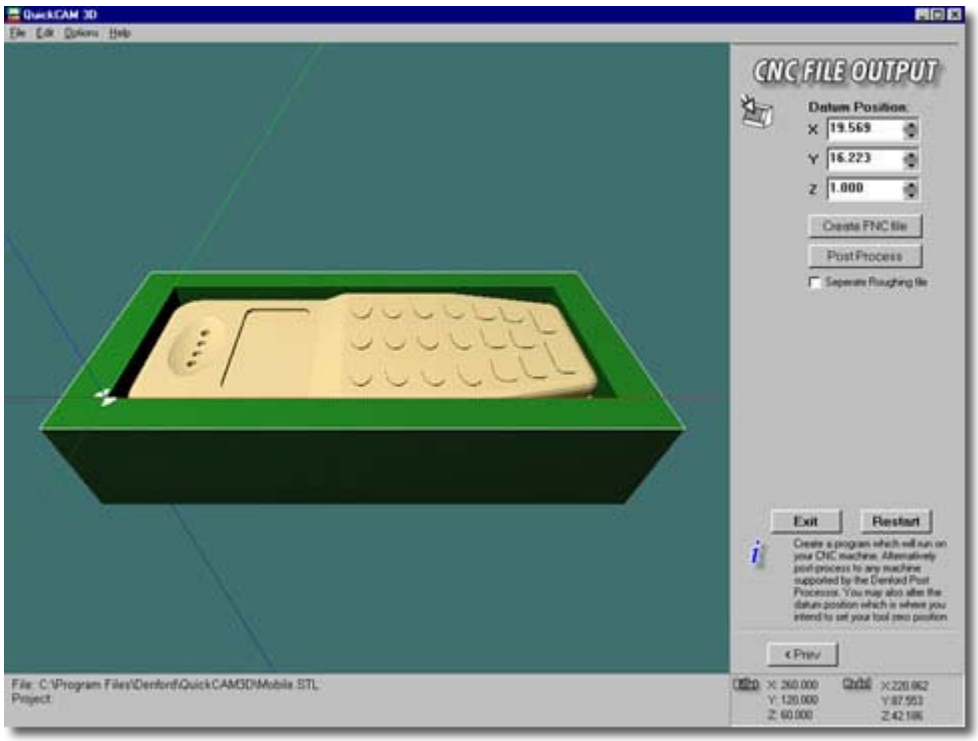
Back to top of page
< Previous - Next >

QuickCAM 3D
Stage Nine - CNC File Output

< Previous

Overview

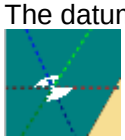
You can decide where to place the datum position (the zero point of your CNC file) - you will have to find and set the same position on the billet used in your CNC machine. Finally, create your Denford CNC file or post process your design for manufacture on a specific brand of CNC machine.



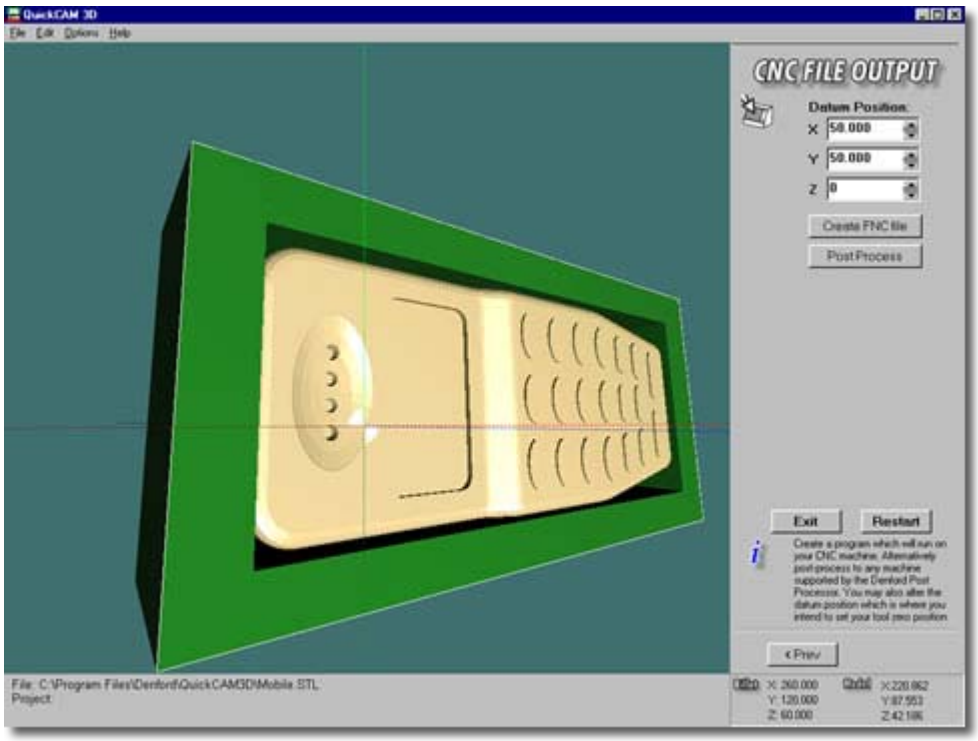
Configuring the Datum Position

The datum is the zero point of your CNC program. The default datum is positioned in the front left upper corner of the billet, as shown in the example at the top of this page.

The datum symbol looks like this...



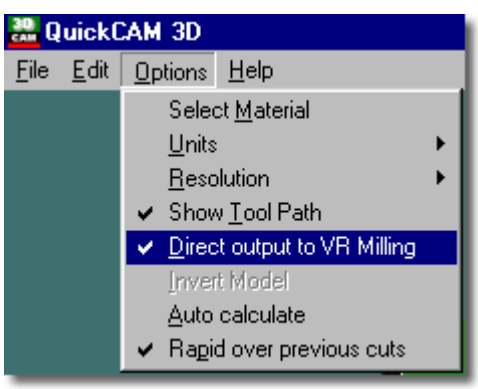
Using the 'X', 'Y' and 'Z' readout panels, values can be entered to alter the position this datum. Positions can be changed by over typing with the required value, or by using the [up or down arrow] nudge buttons. Any datum values must be entered using the current units of measurement - checked by clicking the "Option|Units" menu.



For example, you may want to move the datum to align with a specific part of the billet or a known position on a jig or fixture that you will be using on the CNC machine. Remember, the datum position you set here will need to be found on the CNC machine, when you configure your CNC machine offsets. In the example above, we have moved the datum so the zero position of the program will lie along the centreline of the Y axis, 50mm inside the left edge of the billet.

Creating an FNC file

Click the Create FNC file button to process your data and tool path into a CNC program suitable for use with a Denford CNC machine. CNC programs are saved using the file extension ".FNC".



In the "Options" menu, if "Direct Output to VR Milling" is checked on (shown by a tickmark symbol) the Denford VR CNC Milling software will be automatically started and the processed CNC program will be automatically loaded into the editor.

Please note that when using Denford VR CNC Milling software version 2.13 or before, only the VR Milling software will load - the CNC program must be manually loaded into the editor.

Post processing a file

Click the Post Process button to process your data and tool path instructions into a CNC program suitable for use with a specific Denford CNC machine. Select the name of the machine from the available list, select the area to save the file and click [OK] to post process.

Post processors supporting popular CNC machine manufacturers are also available as optional extras to the main QuickCAM 3D software package.

Click here for an quick overview of each stage of the process

Back to top of page < Previous