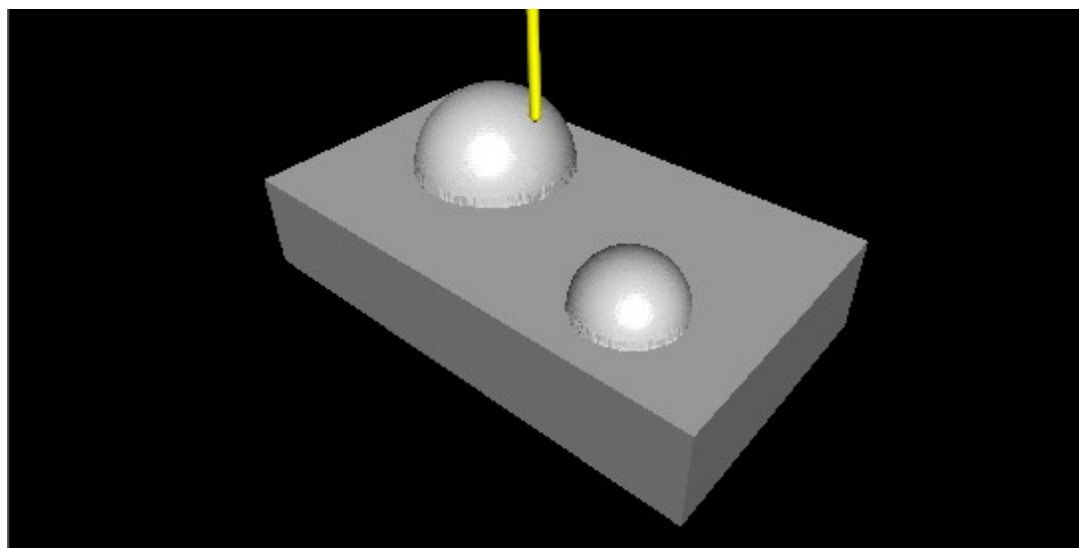
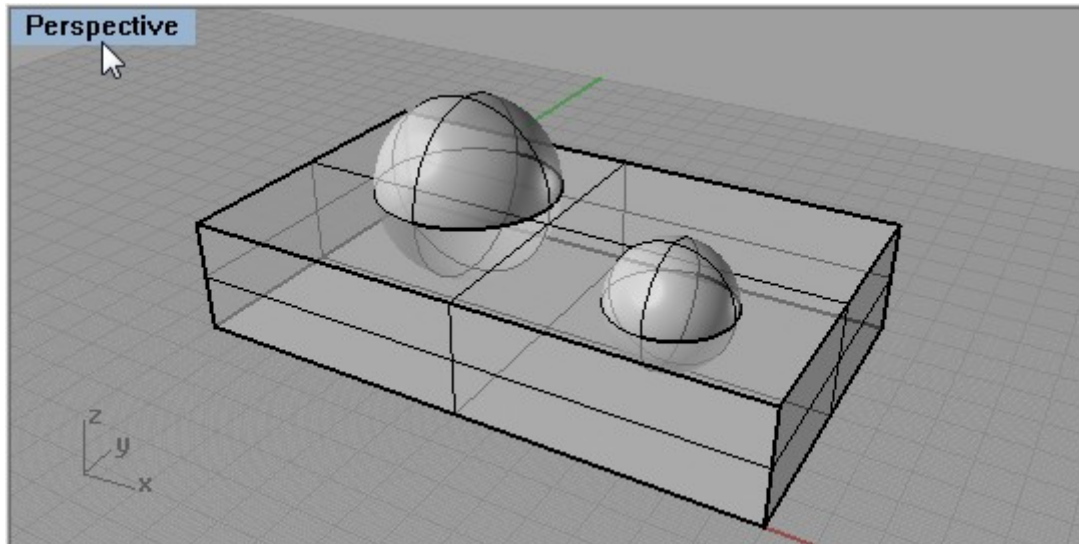


## MadCAM 4.2 Tutorial - 3D Basics

How to create a basic toolpath from a model.



## About this tutorial

MadCAM®, *Mould And Die Computer Aided Machining*, is a Rhinoceros® plugin for 3D machining. The MadCAM plugin gives the ability to model and generate toolpaths inside Rhino 3D, from contour milling to advanced milling strategies with 5 axis machines.

This tutorial is an aid for the user to quickly get knowledge about the functionality and strategies within MadCAM and learn some tech tips on the way.

Command functions that have their origin from Rhino are typed in *italic*: *OffsetSrf*

Toolbar and menu functions are also written in *italic*: *Solid-Box* -> *Diagonal*

Functions from MadCAM are typed in ***italic bold***: ***Create Box*** -> ***Select Region Curves***

## The Basics

When MadCAM is installed there will be a new toolbar available. All functions are available from this new toolbar and the sub menus within it.

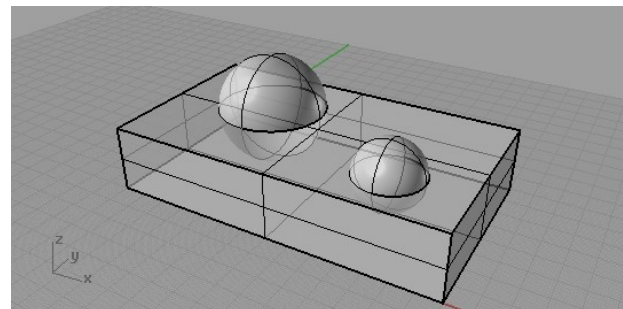


*It is wise to click on the help icon (the right most button in the toolbar) and read the chapter about MadCAM settings to get the most out of the plug-in.*



If the toolbar is missing, or if it's lost during work progress, it can be found by following the menu *Tools-Toolbar layout...* and then by choosing MadCAM in the collection list. When highlighted all available MadCAM toolbars are visible in the Toolbars list. The main menu is called MadCAM 4.2. Enable it and exit.

To continue with the tutorial, a model is needed. Create a simple model by making a box with two spheres on top. Let the spheres be halfway down in the box. The dimensions of the model is not important in this scenario. The example box to the right is 140 by 80 mm (~5.5 by 3.15 inches) in the base, but anything would do.



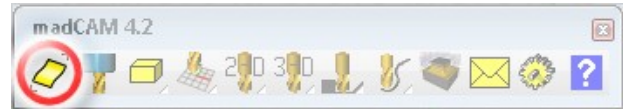
Note that the model position in Rhino's coordinate system is relative to the machine when the model is machined. If the box is made with one corner at Origo (position 0,0,0) that is the place for it in the machine as well.



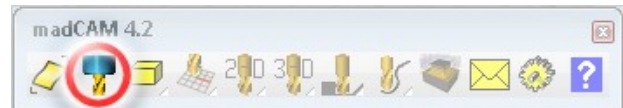
This model will be reused, saving it will save time in later tutorials.

## The quick version

Select the model, click **Select Surfaces...** and the model will be surrounded by a green wireframe which represents the stock.



Click on **Create / Load Cutter** and select a suitable cutter. Then click OK.

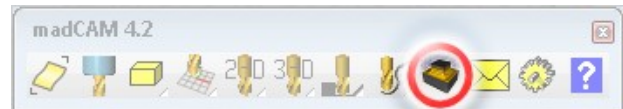


Click on **Create 3D Toolpath** -> **Roughing**. Leave the default values and click OK.

MadCAM generates a toolpath for roughing and adds the path to a new layer.



To see the result click on the **Simulate** button and run the simulation.



Either with the same cutter or with another one selected (via **Create/Load Cutter**), click on the **Create 3D Toolpath** -> **Z-level finishing** to create the first finishing toolpath. Use the default values and click OK.



MadCAM generates a new toolpath and adds the path to a new layer.

To see the result click on the **Simulate** button and run the simulation.

Either with the same cutter or with another one selected (via **Create/Load Cutter**), click on the **Create 3D Toolpath** -> **Planar Finishing**. Use the default values and click OK.



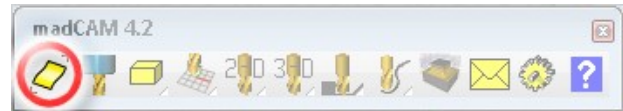
Once again a new layer is added with a new toolpath. Use **Simulate** to see the result.



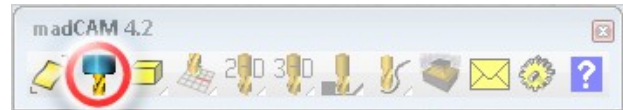
## The detailed version

### Roughing

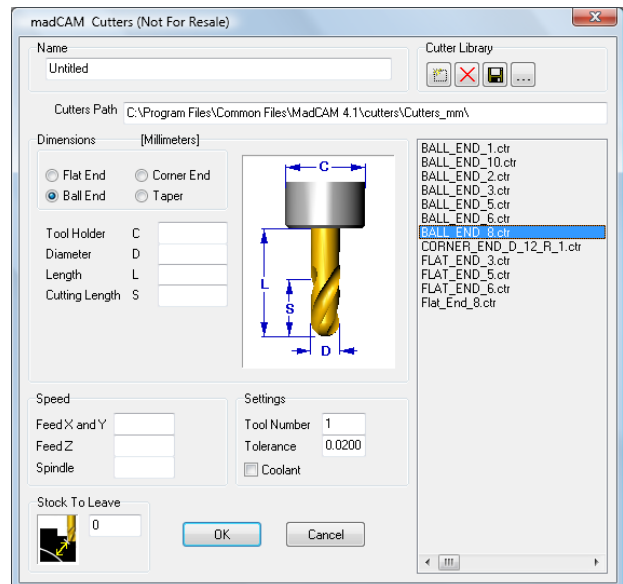
Select the model, click **Select Surfaces...** and the model will be surrounded by a green wireframe which represents the stock.



Click on **Create / Load Cutter** and select the tool that will be used when cutting the model.



The Cutters dialog will show up. This is the dialog to use when defining a new tool or when to change tool between tool paths. There are a number of predefined tools that can be used. To make a new tool definition, either edit an existing tool by choosing one in the right most column and save it under a new name, or create a new tool by clicking the **New Cutter** icon in the top right corner and set the correct values. Save it under a name that makes it easy to identify the tool afterwards.



To continue with the tutorial, choose the cutter named BALL\_END\_8 and click OK.

## More about Cutters

The Cutters dialog is your library of tools that can be used when generating tool paths. A tool can be used as is, modified or created directly in the dialog.

To create a new tool, click the **New Cutter** button in the upper right corner button row.

Specify what kind of tool type it is by choosing the profile (Flat/Corner/Ball/Taper), the input fields and the tool drawing will change to reflect the choice.

By following the drawing, set the dimensions in the input fields. Remember that the tool length and Tool Holder value will affect how deep Madcam can lower the tool when creating tool paths. Make sure they are correct otherwise the tool might hit the stock or the model. It's better to specify a too wide tool holder and shorter length than the real tool length to be on the safe side. If all dimensions are known, then filling in the correct values is the preferred way to go.

Speed and spindle settings are the standard settings for this tool, specify a "normal" value. If the tool needs to go with another speed, for example in a softer material, the feeds can be changed when the tool is selected (select tool, change feed, press OK).

**Stock To Leave** is very useful. If the value is set to a *positive* value the tool path will leave that thickness. If the value is set to 3 mm, all surfaces will have 3 mm material left when the milling is done. This is a good feature to use when leaving a thin layer of material after roughing to get a smooth surface when finishing.

If the value is *negative*, the tool will consume the model, there will be less material left than the Rhino model. If the value is set to -2 mm the tool path will make all surfaces with a 2 mm cut off. This is a very useful feature when making die-sinking graphite tools (always smaller than the expected cutout) or when making foam cores that will be wrapped with a laminate.

**Stock To Leave** can also be set directly in the Tool Path dialogs and will have the same effect.

Give the tool a descriptive name like the existing tools or similar. As long as it's easy to identify the tool by the name, it will be much easier to handle the tool library when it grows. By double clicking in the name field, the tool will be automatically named.

Now, don't forget to click **Save** before leaving the dialog, unless the tool is temporarily.

The model is defined and a tool is selected, now it's possible to start generating toolpaths. Click on **Create 3D toolpath** -> **Roughing** in the submenu.

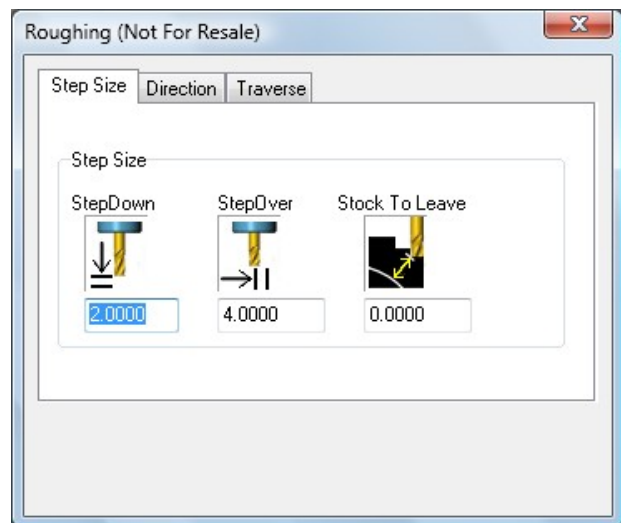


Roughing is the first cut when 3D-milling. Generally the roughing is made with a bigger cutter that can take heavier loads and deeper cuts. This is to shorten the milling time and to save the fine cutting tools.

In the Roughing dialog there are some options that can be changed depending on what material that is going to be cut and what type of tool and machine that will be used.

For example if the material is foam the **StepDown** that specifies how deep each cut should be, and **StepOver** that specifies the overlap for each cut, can be much higher than if the material is steel. With higher values, the milling time will be shorter. But if these values are changed without knowledge of the tool and machine capacity, serious damage can occur.

**Stock To Leave**, that also could be set in the Cutter dialog, specifies how close to the final model the tool should go. Stock To Leave can also be a negative value to have the tool cut off more than the real model. This is a very useful feature if the model is going to be built up with extra material after the milling, for example draped with glass fibre or covered with a skin material. It's also very useful in the finishing faces of the milling. If the model is a graphite tool for die sinking all tools from first roughing (smaller tool) to the last finishing (close to real size) can be made from one single model.



The **Direction** tab specifies how the cut should be done. Besides the different strategies the can make the roughing better (Pocketing is usually better for cavities than ZigZag) there can also be some time saving here as well. Depending on what the model looks like and how the machine handles direction changes the actual machine speed can increase.

Keep Pocketing as the Direction.



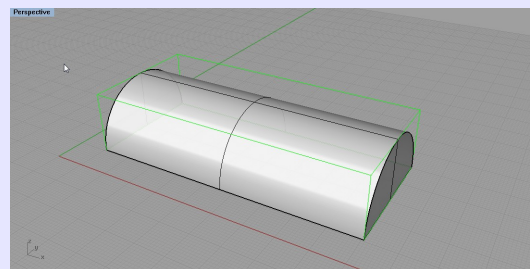
## More about Direction

Imagine a model that is a half cylinder. If this model is cut with direction set to ZigZagY the toolpath will only make short cuts before it hits the model and have to either lift or change direction.

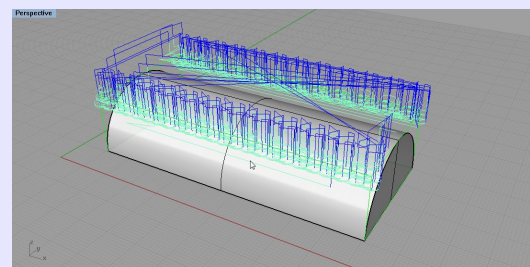
The second image shows the the result of a roughing with ZigZagY. The toolpath generator is forced to change direction very often and this causes speed loss, longer cutting path in total and more travel.

The third image is the result of a toolpath generated with Direction set to ZigZagX. Compared to the first roughing this strategy generates a shorter toolpath and few travels. If the machine is sensitive to direction changes this toolpath will also let the machine controller keep the feed high thanks to the straight long lines it cuts instead of many short like in the first example.

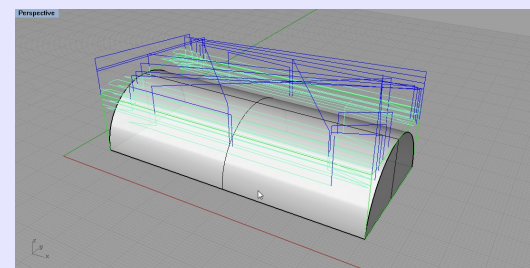
Experiment with the settings and compare the results, run the different toolpaths in the simulator (see below). Also notice if different strategies affects the real machining.



More about Direction: First image, the model itself



More about Direction, image 2: Roughing with ZigZagY.



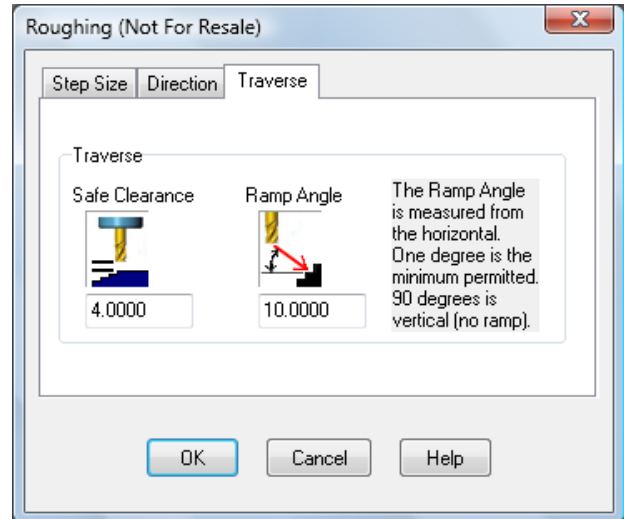
More about Direction, image 3: Roughing with ZigZagX

The **Traverse** tab has two settings, the **Safe Clearance** for rapid movement and if the tool should go into the material with an angle, **Ramp Angle**.

**Ramp Angle** gives the ability to let a big cutter with no drilling capabilities enter the stock material without precutting by smoothly get into the material sideways. If the tool in use is very good in vertical movements (such as most two fluted end mills) the ramp angle can be vertical.

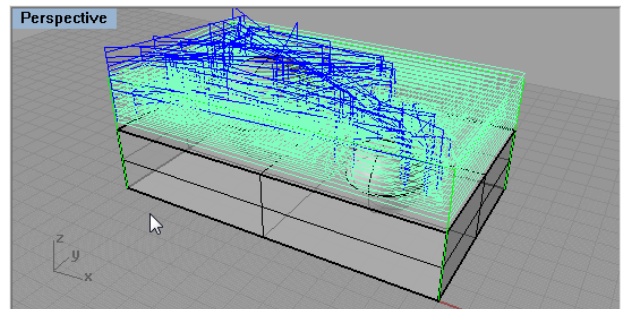
For the tutorial example, leave the default values.

Now it's time to generate the toolpath.



Press OK in the dialog and the toolpath generation begins. Depending on the available computer speed this will take more or less time. When the toolpath is ready it will show up in a new layer.

The cyan coloured lines is cutting toolpath and the dark blue lines represents rapid travel.



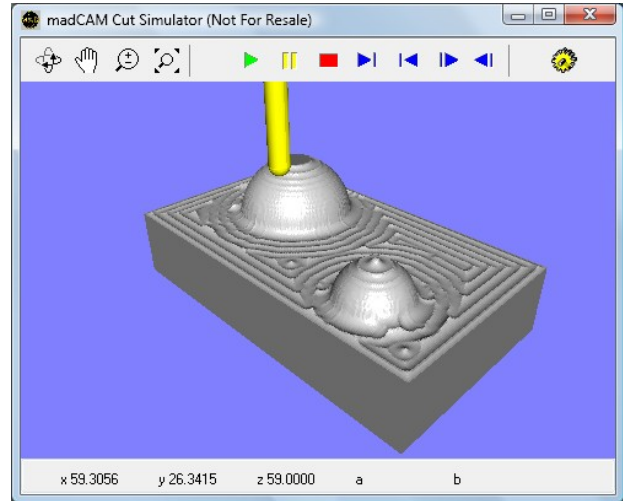
MadCAM has a built in simulator that makes it possible to see the cutting and what the model will look after the cutting. Click the **Simulate** icon in the toolbar and the simulator window will show up.



In the new window, the **MadCAM Cut Simulator**, click the play button to see the cutting. To fast forward to the end of the toolpath click the first blue arrow called **Next Toolpath** that will take the active toolpath to the end.

If something that happens in the toolpath needs extra watching or to get a deeper knowing how the cutting works, click the **Pause** icon and use the **Step Toolpath Curve** and **Back Toolpath Curve** buttons, the two rightmost blue arrows. By using these functions the toolpath can be watched step by step to see what's happening.

When the simulation is done, close the window.



## Finishing

The model has been enriched by two new layers. One is the stock model and the other one is the roughing toolpath. To simplify the tutorial the same cutter will be used. Otherwise, go back to the **Select Cutter** dialog and select the finishing tool.

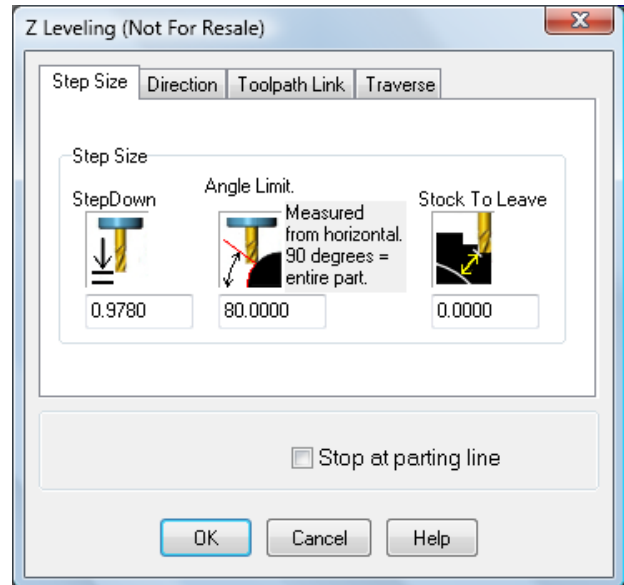
Now the finishing phase of the model will begin. Click the **Create 3D Toolpath** -> **Z-level finishing** icon.

The Z-level finishing generates a toolpath that only affects non-planar surfaces. It will create a constant depth cutting toolpath for all surfaces within the angular tolerance given in the dialog. The constant depth is very important when cutting in hard materials.

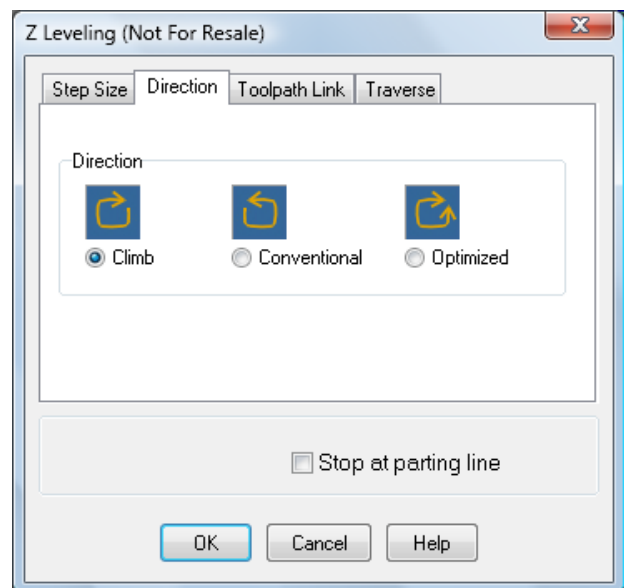


Note that when milling in soft materials the Z Leveling can be skipped and the whole finishing process can be done with **Planar Finishing** (see below). It's not always the fastest strategy to skip this step though. The two finishing steps should in most cases be combined.

**StepDown** defines the depth for each cut.  
**Angle Limit** is the value that defines how close to a “planar” source the tool should get. In this case there is 10 degrees left to vertical (90-80). As long as the angle of the surface is in between the given angle and horizontal it will be affected by the cutter. All the other surfaces, the horizontal areas and the surfaces that only have an angle at 10 degrees will be left untouched  
**Stock to leave** is the same as before.

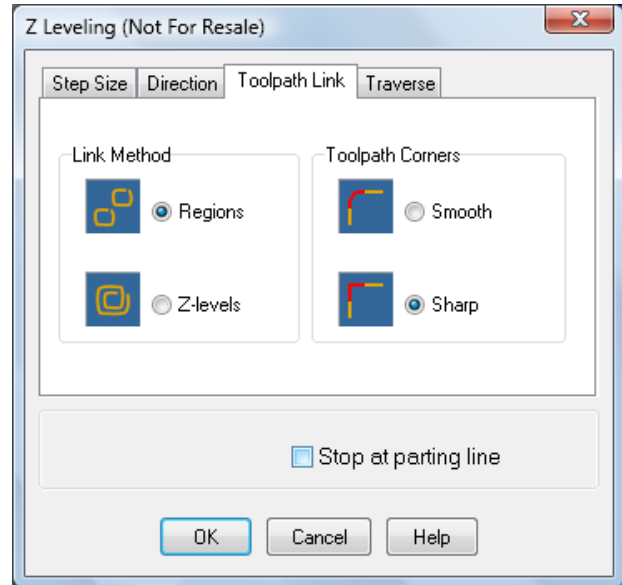


The **Direction** tab now looks different. The options are related to the cutter rotation. Depending on material, cutter type and machinery these can be changed to get the best surface, or to optimize speed.



The **Toolpath Link** tab options can optimize the toolpath to be faster and smoother. **Link Method** defines if the cutter should finish one depth before next depth or if the toolpath can stay in one area and finish many levels to minimize transport time. For the tutorial example, and most models, **Regions** is fastest.

To get smoothness in the toolpath, the corners can be allowed to be softened. For accuracy, leave the option at **Sharp**.



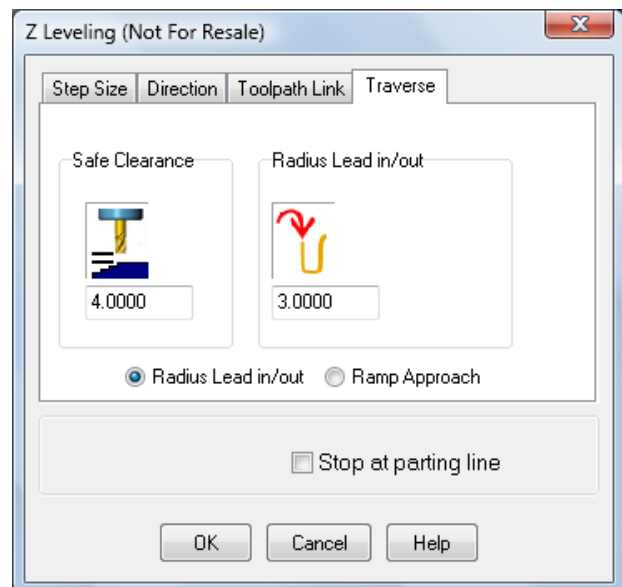
The last tab in the dialog is **Traverse** that now has a choice between radius or ramp lead in.

**Radius Lead in/out** will put the cutter at the right depth but in open air and smoothly hit the material with a radius.

**Ramp Approach** is the same as in roughing, the cutter will be lifted and enters the material with an angle.

Now click **OK** and the toolpath generation starts.

When the generating is done, click on the **Simulate** icon and click **Simulate** (the play button) to see the toolpath and how it affects the model.



Now it's time for the second step of finishing, the **Planar Finishing**. The planar differs from the Z-Leveling as it handles the flat or very little angled surfaces.

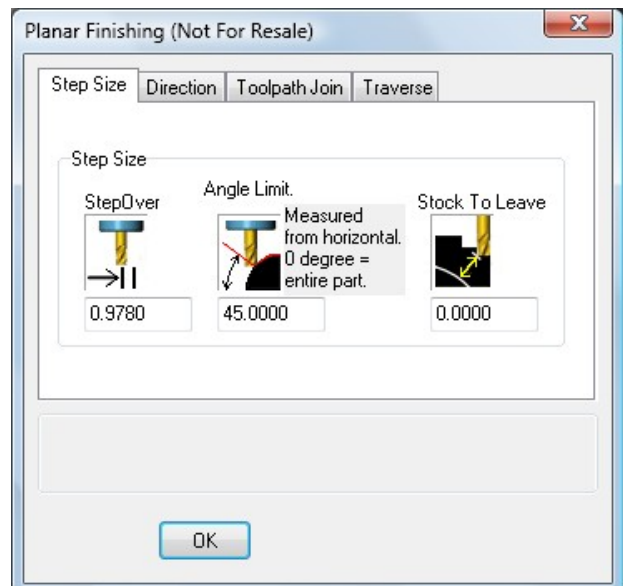
Once again, if another tool is supposed to be used for Planar Finishing (perhaps a flat end cutter for better surface), return to the **Create / Load Cutter** dialog and choose another cutter.



Click on the **Create 3D Toolpath**-icon and then **Planar Finishing**.

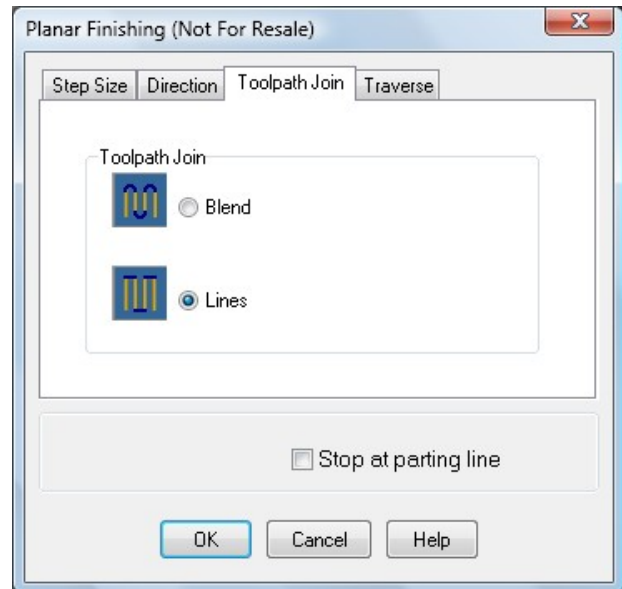


The dialog is very similar to the other 3D dialogs, but with the difference that the **Angle Limit** is opposite to Z-Level, it now calculates surfaces from 0 degrees to the given angle.



The **Toolpath Join** tab specifies how the toolpath should be joined. A **Blend** with a curve will create a smoother cutting with speed loss, and **Lines** will gain speed. This is once again machinery, tool and material specific.

Click OK and wait for the toolpath to be generated. One more time, click **Simulate** and check the toolpath in the simulator.



## Pencil tracing

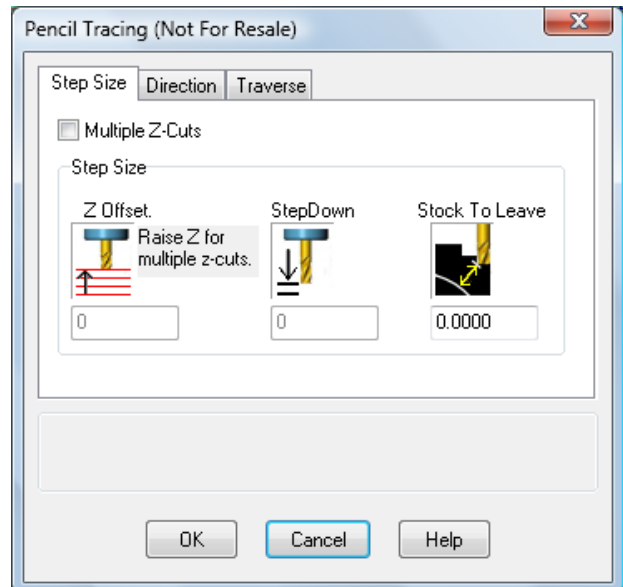
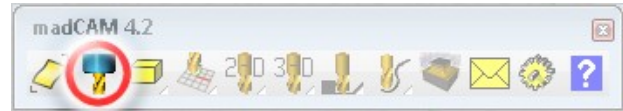
Now when the surfaces are milled to the model surfaces, there is still material left. As soon as a radius or inner corner has been smaller than a radius of 4 mm (as the 8 mm ball end has been used all the time) there will be material to cut. This is why **Pencil Tracing** exists.

Now, the 8 mm ball end mill must be replaced, otherwise the tracing has no effect. Go to the **Create / Load Cutter** dialog and load the FLAT\_END\_6 cutter by clicking on it in the list and then by clicking **OK**.

Click the **Create 3D Toolpath** -> **Pencil Tracing** icon.

To generate the toolpath just click **OK**, but to get a finer trace, enable **Multiple Z-Cuts** and set the value of 5 in **Z Offset** and 0.1 in **StepDown**. This means that the tracing should start 5 mm up from the tracing area and cut 0.1 mm at a time. Now, click **OK**.

When the toolpath is ready, check cutting in the simulator.



## Post processing

To make the toolpaths into machine code they need to be post processed. Click on the **Postprocess** icon.



The most important to do in this stage is to select a post processor description that works with the machine that is going to be used for the work. That is done in by choosing the a file that works with the machine with the file selector in the **Post Processor** field.

Click **Post Process >>** and choose a file name for the file. The processing starts and when it's ready it will yield a Total machining time calculated from the settings for the tool etc.

Load the file in the machine and execute.

