SolidCAM Quick Start Guide

This tutorial demonstartes an overview of the basic process of generating toolpaths for CNC Milling using SolidCAM.



Open the part

- We will be making this part. Notice two things:
- Stock material was created within the part as a seprate solid body. This can be done by performing a boss/extrude and deslecting "Merge Result"
- The view state of the stock material is transparent

Machining Assembly

Create a new assembly that contains the part and stock model as well as any fixtures. In this case, the MiniMill fixture plate. Loacate your part where you would like it on the fixture plate.

Create a fixture plate for each machining orientation, keeping your part in a single position.



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Machining Orientation Coordinate System Now, create a machining coordinate system for each machning orientation. Reference Geometry > Coordinate System

- For this part, the Coordinate System was created at the upper left of the part with the z origin at the top of the fixture plate
- Always create the origin at a point that can be re-found.
- Another, possibly better, option for the coordinate system for this part would be the upper left of the fixture plate itself.

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Create new SolidCAM Part In the main menu, select:

SolidCAM > New Milling Part

In the menu shown at left, select the options shown.

• These cannot be changed.

Define Initial Parameters

Select the machine that you will be using.

- The menu on the left of the screen will now appear. This is where you will define the initial parameters for the part.
- These can be modified later.



Select a coordinate system

In the "Define CoordinateSys Options" box select "Select Coordinate System". In the box below, select the Coordinate System(s) that you wish to use.

- Select all coordinate systems that you will want to use. When generating toolpaths, you will select the specific coordinate system that you want to use.
- You can also create coordinate systems from edges and surfaces, but it is much easer to define the Coordinate System in SolidWorks and then select it in CAM.

Return to the Inital Parameters menu by pressing the green check.

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Define Coordinate System Levels

Selecting the button corresponding to each level allows the level to be selected in CAD. Alternatively, the numbers can be entered manually.

These are the levels that the tool travels to in different cases. As a general rule, the levels are listed in descending order.

- Tool Start Level height the tool moves to before starting an operation. This should be above the maximum level of the stock.
- Clearance Level height the tool moves to during rapid movements. This should be above the maximum level of the stock.
- Part Upper Level The top of the part.
- Part Lower Level The bottom of the part.
- Tool Z-level used in 4+ axis milling.

Select Stock and Target Material

In the Initial Parameters menu, click the button that reads "stock"

A menu menu is brought up on the left side of the screen.

Select "CAD Selection"



CAD Selection

Navigate to the part in the SolidWorks Feature Tree and select the stock material under Solid Bodies

Select "Resume" to exit CAD selection

Select the green check mark to exit Part Selection

Use the same process to select the "target" - your part.

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Finish Inital Parameters

Press the green check to finish defining the Inital Parameters

• The default "Settings" in the Part Settings box do not need to be modified.



Main SolidCAM Page

- This is the main SolidCAM page.
- There is a similar "Feature Tree" to what you are familiar with in SolidWorks.
- The Inital Parameters that you just defined can be edited by double clicking the Parameters in the top of the Feature Tree.

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Editing Tool Parameters (2) Select "OK" to return to the main window.

The highlighted values are incorrect



Selecting a Fixture

In the main SolidCAM window, souble click the "Fixtures" menu and use "CAD Selection" to either directly select the fixture or select it in the SolidWorks Feature Tree.

Press the green check to return to the main menu.

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Create Milling Operation

In the SolidCAM menu on the left right-click on the operations folder and select: Add Milling Operation > 3D Milling...

- This is the broadest machining operation and the one that you will use 90% of the time.
- Other milling operations will not be covered in this tutorial.

Define Geometry

Adding a new milling operation opens this window and starts you in the geometry tab.

Name your operation to indicate what tool it will use and if it a roughing or finishing pass.

You can "Define" the working area but that is not necessary for the roughing pass of this part.

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Select Operation Tool

The next tab is the Tool tab. Press the select button and choose the tool that you want to use.



These are usually fine from the level you defined when defining the coordinate system.

• Levels can be used to focus on sections of a part or split up a large file.

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Define Technology

For this part, we start with a rough pass with the parameters shown on the left.

When parameters are defined, select "Save and Calculate" and then select "Simulate". This will simulate the machie motion and show you a preview of the part after this toolpath is run.

- This is where things get fun! This tutorial won't go into too much detail as the best way to learn is through practice and experimentation. SolidCAM also does a good job explaining the different parameters within the interface.
- You have the choice to perform a Rough, Semi-Finish, or Finish pass. All three give you different options. Usually you will not want to do all three with the same tool. It is also good to note that it is not necessary to perform all three. That is completely up to you.

Link, Tolerance, and Motion Control will not be covered. They are unnecessary for most 3D.

Simulation

Selecting the "Simulate" button in the above window will open the simulation window.

Choose a simulator and set any options and then press "Play"

Simulation is essential to observe what the part looks like after each milling operation and identify areas of the part that need further refinement through additional operations. After roughing, this part clearly needs refinment.

I primarily use three simulators:

- SolidVerify is the highest quality simulator that does not require major setup. With very complicated surfaces, it can have a tendency to crash but it is generally reliable.
- **RapidVerify** works well but does not always render the most accurate results.
- Rest Material shows the material that is not part of your part after a milling operation.

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Second Roughing and Finishing Pass Follow the same procedure as before to create a new Milling Operation (Add Milling Operation > 3D Milling...)

Name your operation as before.

This time, we will edit the working area. Select the "Define" buttion in the Working Area box.

- Both operations will be performed with the same tool so they can be combined into a single milling operation.
- A second roughing pass is performed to remove more material before going to a finish pass. This will lead to a better surface finish but is not necessary in all cases.

Working Area

Select the "Cut only the Rest material" checkbox. This will force SolidCAM to identify the material that was removed by the previous pass.

The working area can be constrained to a surface or feature by selecting the "Working area" checkbox and then selecting "Define". This will allow you to use CAD selection to either directly select areas or use the SolidWorks Feature Tree.

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		Et al.	Explore other options through simulation.

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he drop-down menu in the "Finish" box, a finishing option. We will use "Constant his pass. The tool will stay at a constant id countour the part topography at Z levels.

open the Constant Z Finish parameter ١.

and define the appropriate parameters.

BD Milling Operation window, select "Save Iculate" and then "Simulate"

- ble Step Down allows for scallop height defined. It tends to be faster that tant step down and results in a more orm surface finish when compared to tant Step Down.
- ag open contour strategy will reduce movements and decrease machining

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Modifying Geometry

Simulating the part in Rest Material highlights an unmachinable region. Upon further inspection of the part, it is noted that this corner was not filleted!

Exit out of the simulation and navigate to the SolidWorks Feature Tree. Right click on the part and select "Edit Part". Add the fillet and save.

Close the part and return to SolidCAM.

The update was made but it is not reflected in

Right-click on the CAM-Part menu in the SolidCAM Feature Tree.

Select Synchronize.

It still does not update. WTF!

The trick is to double click on the CoorSys Manager and edit MAC-1 and the rest of your coordinate systems.

In the coordinate system edit window, right click in the selection box and select "Unselect All".

Select "Define" and reselect your coordinate

Asterixes should now appear next to all of your milling operations indicating that geometry has changed and the operation needs to be recalculated.

Right click on your operations and select "Recalculate". The new toolpaths should reflect the updated geometry.

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Final Simulation

Final simulation using Rest Material shows the finished part with material remining in holes only. Surfaces are nicely finished.

Saving

Save by selecting: SolidCAM > Save As and giving your part a revision name.

SolidCAM will close your file and open the new saved-as model.

• Having this revision history can be very useful.

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Exporting G-Code

Right-click on a Milling Operation in the SolidCAM Feature Tree and select GCode > Generate.

Name your file the same name that you gave the Milling Operation earlier.

Editing GCode

SolidCAM will open the GCode in Notepad so you can preview it before running on the mill.

The GCode that SolidCAM is not perfect. YOU MUST REPLACE ALL INSTANCES OF G53, G55, G56, G57, G58, AND G59 WITH G54.

Save your GCode to a safe place and copy it to a USB memory drive. We will use this drive to upload the code to the mill.