RhinoCAM

2½ Axis Machining

2025

2 & 3 Axis Setups

MecSoft Corpotation

Table of Contents

Introduction

In this tutorial you will learn how to define a Setup for 2½ Axis and 3 Axis Milling including how to define the Machine, Post, Setup, Stock, Alignment, Material and Work Zero. These steps are typical for all 2½ Axis & 3 Axis machining jobs.

Source Files for this Tutorial

Here are the links to download the source files used in this tutorial:

- AMS-MecSoft-CAM-2axis-Intro.zip

Guides & Videos in this Series

We also suggest that you complete the following companion tutorials and videos in this series:

- Tutorial: 2½ & 3 Axis Setups
- Tutorial: 2½ Axis Introduction
- Tutorial: 2½ Axis Advanced (AMS/Purchase Only)
- Tutorial: 2½ Axis Power Users (AMS/Purchase Only)
- Video: 2½ Axis Introduction
- Video: 2½ Axis Advanced (AMS/Purchase Only)
- Video: 2½ Axis Power Users (AMS/Purchase Only)

Other Supplemental Videos

We also suggest that you watch the supplemental videos on MecSoft.com:

- RhinoCAM-MILL product page
- VisualCAM-MILL product page

The Quick Start Guide

This tutorial assumes that you are familiar with how to load the MILL module and that you have previously completed the MILL Quick Start Guide. You can find this guide by selecting Learn … from the MecSoftCAM Main Menu.

In RhinoCAM:

The 2½ / 3 Axis Setup

The Setup for both 2½ Axis and 3 Axis machining jobs includes the Machine definition, Post, Stock, Alignment and Material. These occupy the first two panes on the left side of the Program tab shown here. While there are many different options available for defining these items, we are selecting the most common methods users will need. The procedure also includes a Work Zero. While optional, the Work Zero is a very common method used to define the Machine Coordinate System (MCS) origin in a location other than the World Coordinate System (WCS) origin.

At the end if this tutorial the Machining Job and display will look similar to this:

Machine

Here is the procedure for defining a Machine for both 2½ and 3 Axis Machining jobs. All the commands and dialogs can be accessed from the Program tab of the Machining Browser.

1. You will be working from the Program tab of the MILL module Machining Browser.

2. From the Program tab select Machine.

3. From the Machine Tool Setup dialog select Manual Definition and then select 3 Axis from the Number of Axis selection list. This is the correct selection for both 2½ Axis and 3 Axis machining.

4. There are General Parameters that affect 2½ Axis and 3 Axis machining. For example you can setup a tool change point and set the Translational Limits of your CNC machine. For most applications, the default values are acceptable.
5. Now look at your Machining Job tree and verify that the Machine is set to 3 Axis.

Post

Here is the procedure for defining your Post-Processor. This is required for all machining jobs. When installed, MecSoft CAM supports over 300 different CNC controllers. There is a Post definition for each one. If you are active on your Annual Maintenance Subscription (AMS) MecSoft will also assist you with customizing your Post-Processor.

1. You will be working from the Program tab of the MILL module Machining Browser.

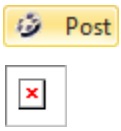
2. From the Program tab select Post.

3. This will display the Set Post-Processor Options dialog. Here you can select your Post as well as other Posting related options. Here is the short list:

   (a) You can select your post definition.
   (b) You can Edit the selected Post using the PPG (Post-Processor Generator).
   (c) You can tell the software what gcode editor program to display g-code files in.
   (d) You can set posting file naming conventions.
   (e) You can also add posted file extensions

4. For this exercise we will go to the Current Post-Processor list and select Fanuc0m as our defined post.

5. Now pick OK to accept the selections and close the dialog. Now look at your Machining Job tree and verify that the Post is set to Fanuc0m.



Setup 1

A Setup defines the orientation of the machine tool's XYZ axes. By default, when you start a new CAD file, a Setup 1 is created automatically that is aligned with the World Coordinate System (WCS) origin in the host CAD system. For most 2½ Axis and 3 Axis applications you do not need to change it. However, if needed here are the steps to edit the setup.

Know Your Product Configuration

The default Setup 1 can ONLY be modified in the Professional and Premium configurations of RhinoCAM. If you are running the Express, Standard or Expert configurations and need to machine in a different orientation relative to your Part, you will need to change the orientation of the part in RhinoCAM.

1. You will be working from the Program tab of the MILL module Machining Browser.

2. From the Program tab select the Setup menu and then select Setup.

3. This will display the Setup dialog. You can use this dialog to change the orientation of your selected Setup. **Note**: In most 2½ Axis and 3 Axis applications you will not need to change it.

4. If you are running in the Xpress, Standard or Expert configuration, you can change the orientation of your part from RhinoCAM

5. From the Program tab select the Setup menu and then select Orient Part.

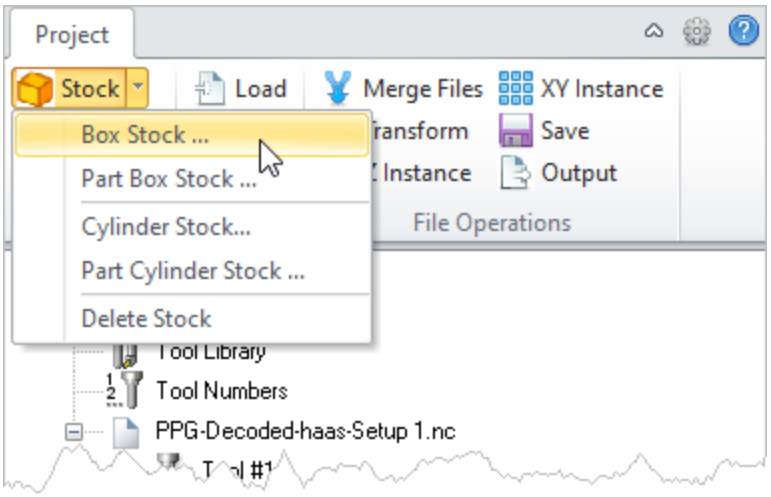
6. You can then use the Orient/Locate Part Geometry dialog to reposition the part for machining.

Box Stock

For most 2½ Axis and 3 Axis applications you will only need a simple Box Stock to generate toolpaths. Using this Stock method you can set the Length, Width and Height of the Stock model. Here are the basic steps to create a Box Stock.

1. You will be working from the Program tab of the MILL module Machining Browser.

2. From the Program tab select the Stock menu and then select Box Stock.



3. From here, select Copy Model Bounding Box to calculate and display the minimum dimensions of the stock box. Then make any adjustments to the Length, Width and Height as needed and pick OK. Our stock dimensions will be 15 x 3.5 x 0.75.

4. Once you enter the stock dimensions, pick OK to close the dialog. The stock model will display on the screen as shown below.

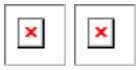
5. If the stock does not display, check to see that it is toggled ON. There is an icon for this locate on the toolbar at the base of the Machining Browser.

Align Stock

Once you have a Stock defined you can also align it in relation to the part. For this example we have a stock that is larger than our part so it will need to be aligned. Here are the basic steps to align your stock and part.

1. You will be working from the Program tab of the MILL module Machining Browser.

2. Select Align and then Align Stock from the menu.



3. This will display the Align Part and/or Part Geometry dialog. The Z dimension of our stock is larger than our part thickness. We did this so that we can machine it to size. For this purpose we will want the stock to be aligned flush to the bottom of the part so that the excess stock extends upward in the Z axis.

4. From the dialog we will set Z Alignment to Bottom and XY Alignment to Center and then pick OK.

Stock Material

Once you have a Stock defined you can also align it in relation to the part. For this example we have a stock that is larger than our part so it will need to be aligned. Here are the basic steps to align your stock and part.

1. You will be working from the Program tab of the MILL module Machining Browser.

2. From the Program tab select Material to display the Select Stock Material dialog shown below.

3. Under Material, drop down the menu and select ALUMINUM - 6061 and then pick OK to close the dialog.

4. The Material selection will only come into play if you select Load from File from the Feeds & Speeds tab of either a machining operation or a Tool definition. It is used to display suggested spindle speed and feed rate values.

Work Zero

By default the program zero point is aligned with Setup 1 which in turn be default is aligned with the World Coordinate System (WCS) origin in the host CAD system. You can use this Work Zero command to place the program zero to another location.

1. You will be working from the Program tab of the MILL module Machining Browser.

2. For this part we want the program zero to be located at the top left corner of the stock geometry. To do this we select Work Zero from the Program tab to display the dialog.

3. From the Work Zero dialog we select Set to Stock Box. Then we set Zero Face to Highest Z and Zero Position to South West and then pick Generate.
4. The Work Zero is created and placed in the desired location. The coordinates of toolpaths located under this Work Zero in the Machining Job will be calculated from this location.
5. The Work Zero is also displayed on the part. If you do not see the Work Zero triad, there is a button located at the bottom of the Machining Browser to toggle it on/off.

Index

- A -

- I -

- M -

- P -

- S -

- W -