

# Create a new Feature-Based Machining Sequence

## Objectives

Learn how to create a new Feature-Based Machining Sequence

## Overview

What is a Feature-Based Machining Sequence?

How to create a new Feature-Based Machining Sequence

## Feature-Based Machining Sequence

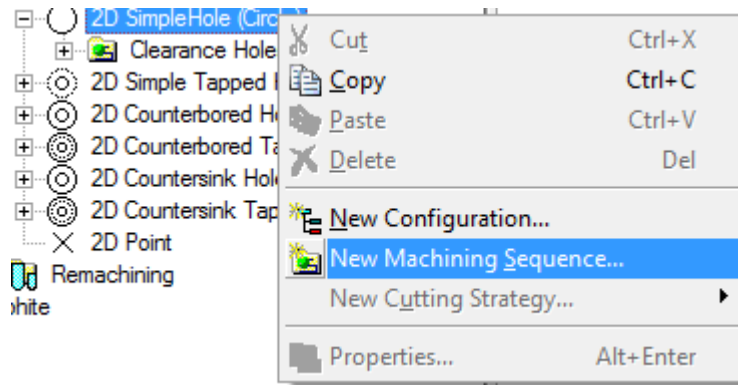
A Feature-Based Machining Sequence defines a series of drilling operations to be used on each feature found when Hole Making. New Feature-Based Machining Sequences must be added to the System database. These are found in the Feature-Based Machining node in the Configuration tree. In that node, we have all the hole features that Prospector can find:

- Simple Through Hole
- Simple Blind Hole
- Counterbored Through Hole
- Counterbored Blind Hole
- Countersink Through Hole
- Countersink Blind Hole
- 2D Simple Hole
- 2D Simple Tapped Hole
- 2D Counterbored Hole
- 2D Counterbored Tapped Hole
- 2D Countersink Hole
- 2D Countersink Tapped Hole
- 2D Point

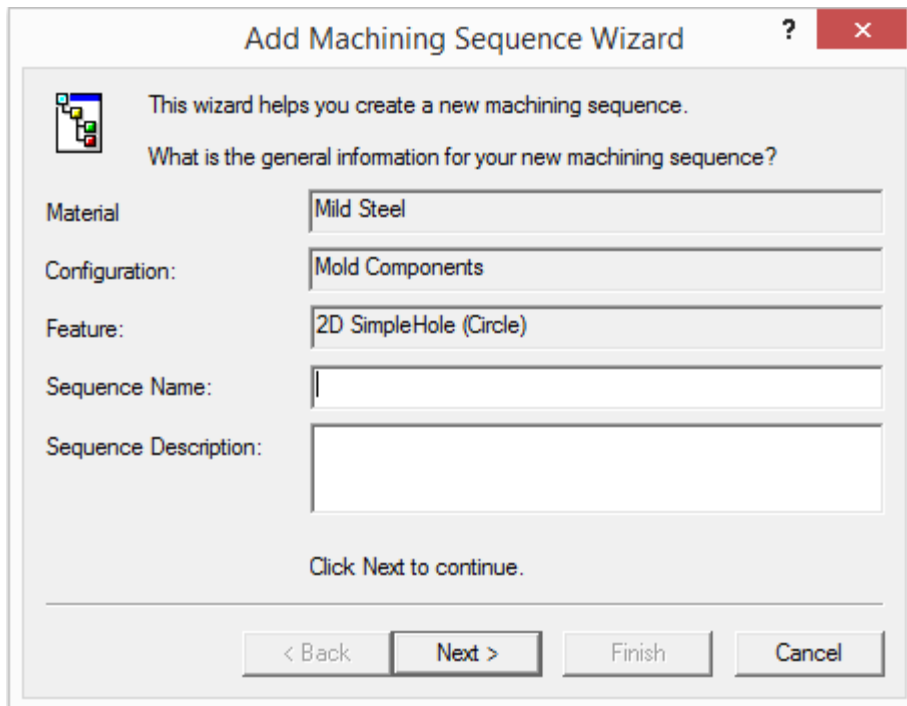
Each hole feature can have any number of Machining Sequences. We have included several examples in our database. There are two ways to add a Machining Sequence. Click on the Feature and under the Insert menu select "New Machining Sequence..." or right-click on the Feature and select "New Machining Sequence..." This will start up the New Machining Sequence Wizard.

### Example 1 - Let's add a Machining Sequence for 2D Simple Holes that also Bores the holes

New Feature-Based Machining Sequences have to be created in the System database, so make sure to load the System Database in Insight. Go to "Cutting Strategy Selection/Mild Steel/Mold Components/Feature-Based Machining." Right-click on "2D SimpleHoles" and select "New Machining Sequence...":



This starts up the Add Machining Sequence wizard.

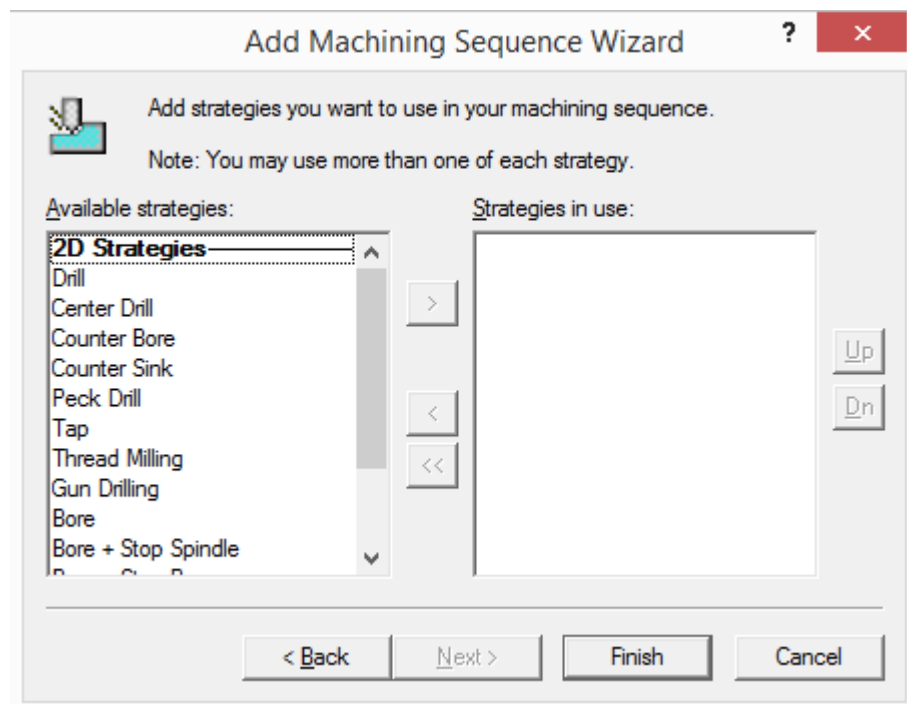


**Page 1 - Machining Sequence information**

It's telling use the Material, Configuration, and Feature. We need to supply the Name and a Description. Use:

Name: Bored Holes  
 Description: Holes that will be machined using a boring cycle.

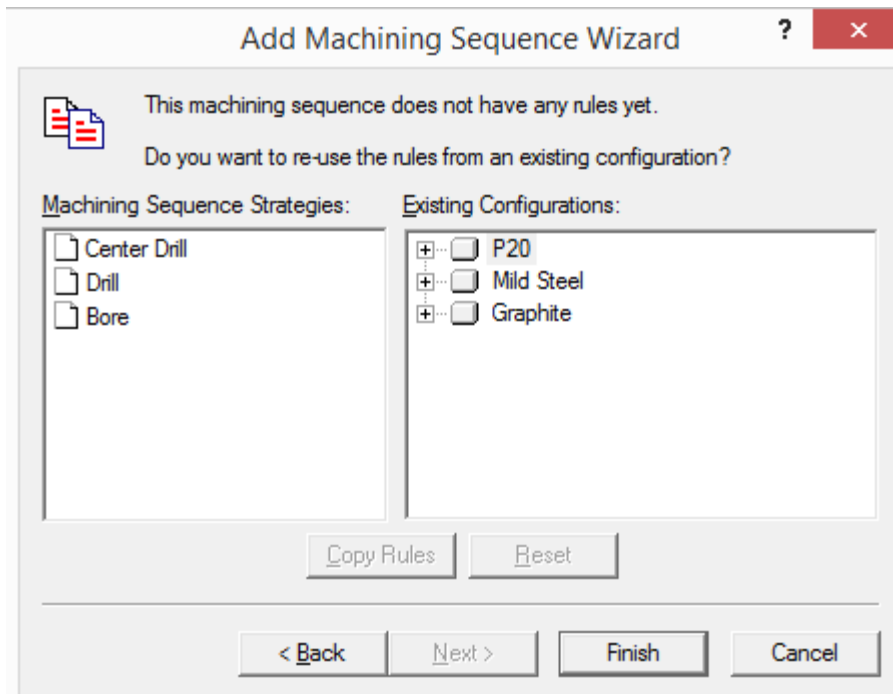
The Name and Description show up in Prospector, we'll see this later. Click Next.



**Page 2 - List of Drilling Strategies**

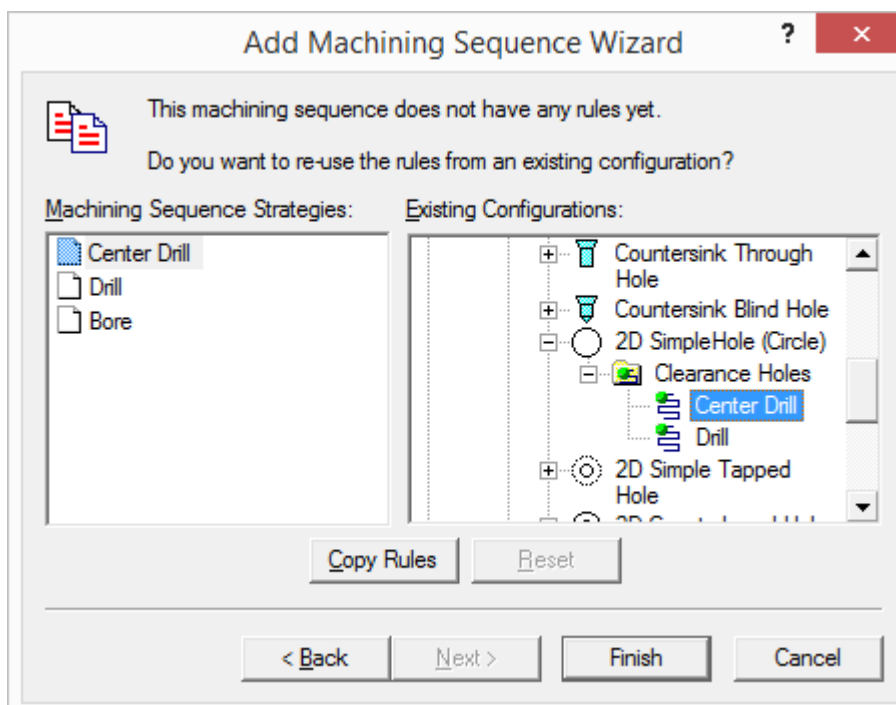
On the left, we have the available drilling Strategies. On the right, is the list of drilling Strategies to be used for this Machining Sequence. Double-clicking on a Strategy on the left will add it to the list. Double-clicking on a Strategy on the right will remove it from the list. Alternatively, you can use the > and < buttons to add and remove a highlighted Strategy. The << button will remove all Strategies. The Up and Dn buttons can be used to re-order the list by moving the selected Strategy up or down. You can add any number of Strategies. You can add multiples of the same Strategy. Add:

Center Drill  
 Drill  
 Bore

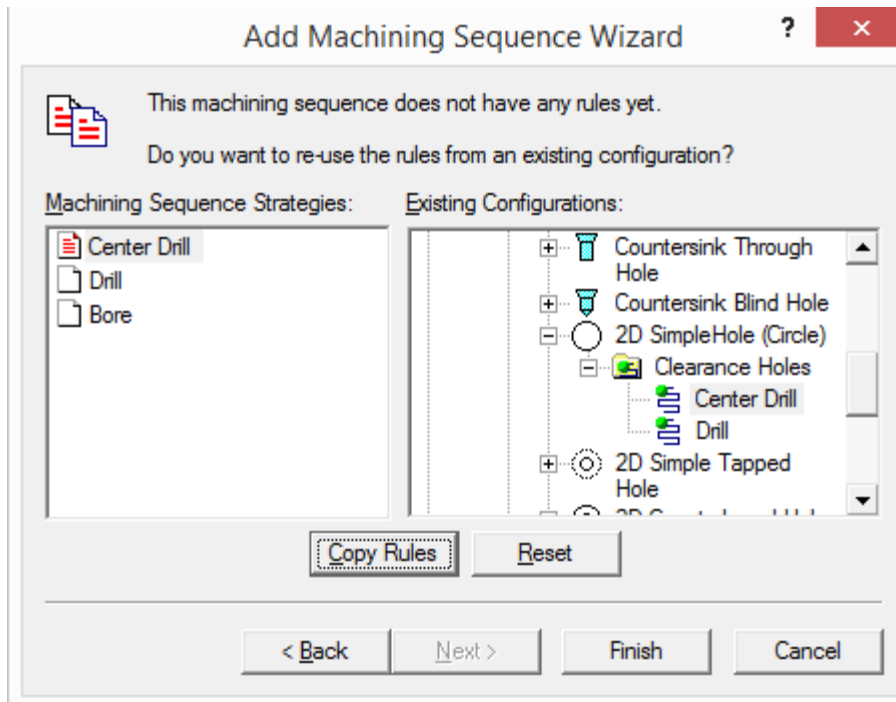


### Page 3 - Copy Rules

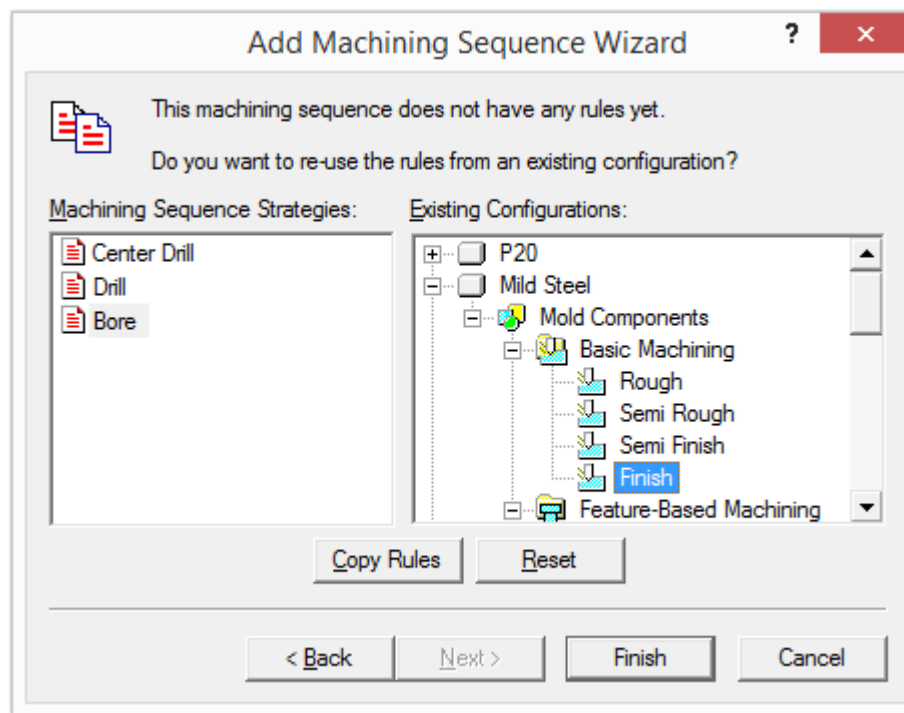
On the left, we have our list of drilling Strategies. On the right, we have a list of Configurations, just like the Configurations pane. We can copy the rules from any Configuration, either Basic Machining or Feature-Based Machining. For the first two, Center Drill and Drill, let's copy the rules from the other Machining for 2D Simple Holes. Click on Center Drill. Go to "Mild Steel/Mold Components/Feature-Based Machining/2D Simple Hole/Clearance Holes/Center Drill."



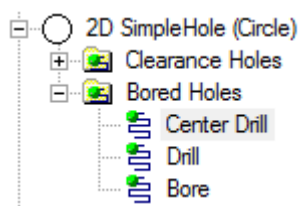
The Copy Rules button is now enabled, click it. The Strategy turns red indicating the Rules were copied. The Reset button is enabled if you want to delete the Rules.



Click on Drill and copy the Rules similarly. For Bore, we'll copy from Basic Machining. Click on Bore. Go to "Mild Steel/Mold Components/Basic Machining/Finish." Click "Copy Rules" to copy the Rules.



Click Finish to add the Machining Sequence. Our new Machining Sequence is added to the list:



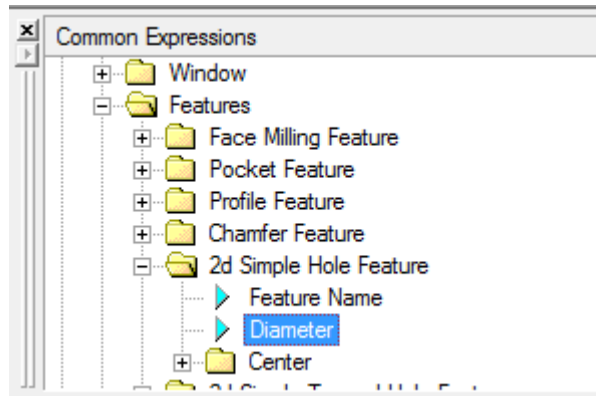
Now let's take a closer look at the rules we just copied, specifically, the tool selection Rule for the Drilling and Bore Strategies. For the Drill Program, the Diameter of the tool is set to the Diameter of the Feature - makes sense.

|                  |                                   |
|------------------|-----------------------------------|
| Drill Tool       |                                   |
| Name             |                                   |
| Diameter         | "2d Simple Hole Feature,Diameter" |
| Length           |                                   |
| Tool Point Angle |                                   |

For the Bore Program, the Diameter of the tool is a complicated Rule because we copied it from Basic Machining:

|                |  |
|----------------|--|
| Tool Selection | "Boring Bar"   |
| Tool Type      | "Boring Bar"   |
| Boring Bar     |  |
| Name           |  |
| Diameter       | <pre> If Not IsEmpty( "3d Simple Blind Hole Feature,Diameter") Then   "Bore,Tool Selection,Boring Bar,Diameter" = "3d Simple Blind Hole   Feature,Diameter" Else If Not IsEmpty( "3d Simple Through Hole Feature,Diameter") Then   "Bore,Tool Selection,Boring Bar,Diameter" = "3d Simple Blind Hole   Feature,Diameter" Else If Not IsEmpty( "2d Simple Hole Feature,Diameter") Then   "Bore,Tool Selection,Boring Bar,Diameter" = "2d Simple Hole   Feature,Diameter" Else   "Bore,Tool Selection,Boring Bar,Diameter" = .5 </pre> |
| Length         |  |
| Reamer         |  |

Let's changed the Diameter to be the Diameter of the Feature. Click in the cell and hit backspace to delete the Rule. Down in Common Expressions, Variables/Features/2d Simple Hole Feature, double-click on Diameter.



Now our Rule is set:

|                |                                   |
|----------------|-----------------------------------|
| Tool Selection | "Boring Bar"                      |
| Tool Type      | "Boring Bar"                      |
| Boring Bar     |                                   |
| Name           |                                   |
| Diameter       | "2d Simple Hole Feature,Diameter" |
| Length         |                                   |
| Reamer         |                                   |

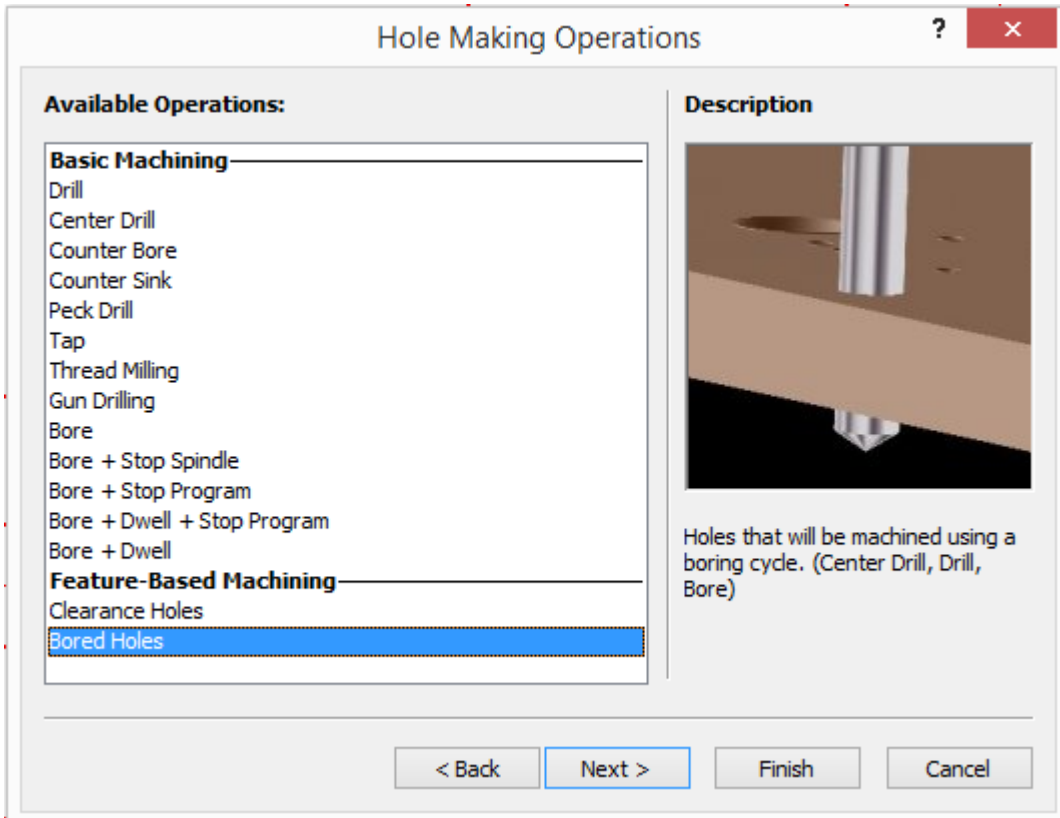
Let's also change the Rule for Depth. Again, it's a complicated Rule based on some of the 3D Features, since this is 2D, let's just use 1" for now. Click in the cell and hit backspace to delete the Rule. Type in 1.0 and hit enter:

|       |   |
|-------|---|
| Depth | 1 |
|-------|---|

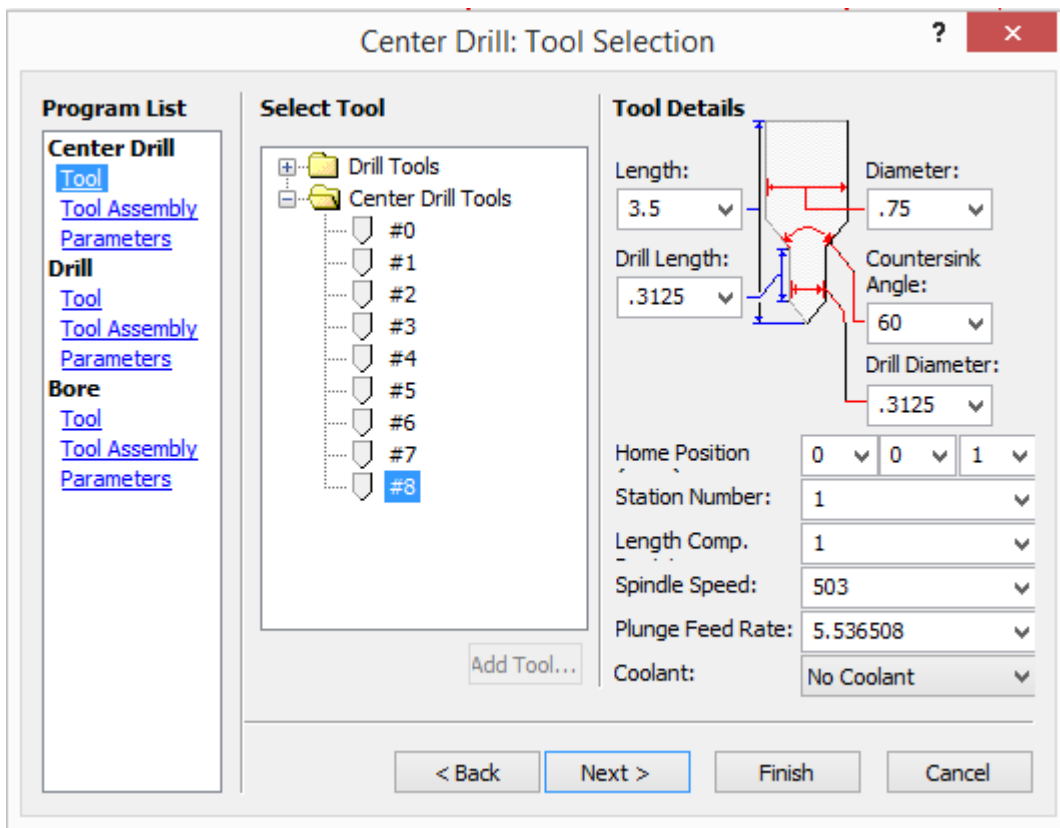
Let's test it out in Prospector. Save the Database.

**Prospector**

In Finish, new 2D Program, pick Hole Making. Find 2D Simple Holes with a diameter of 2.5. At the bottom of the list of drilling Strategies, you see the list of Feature-Based Machining Sequences for this feature.

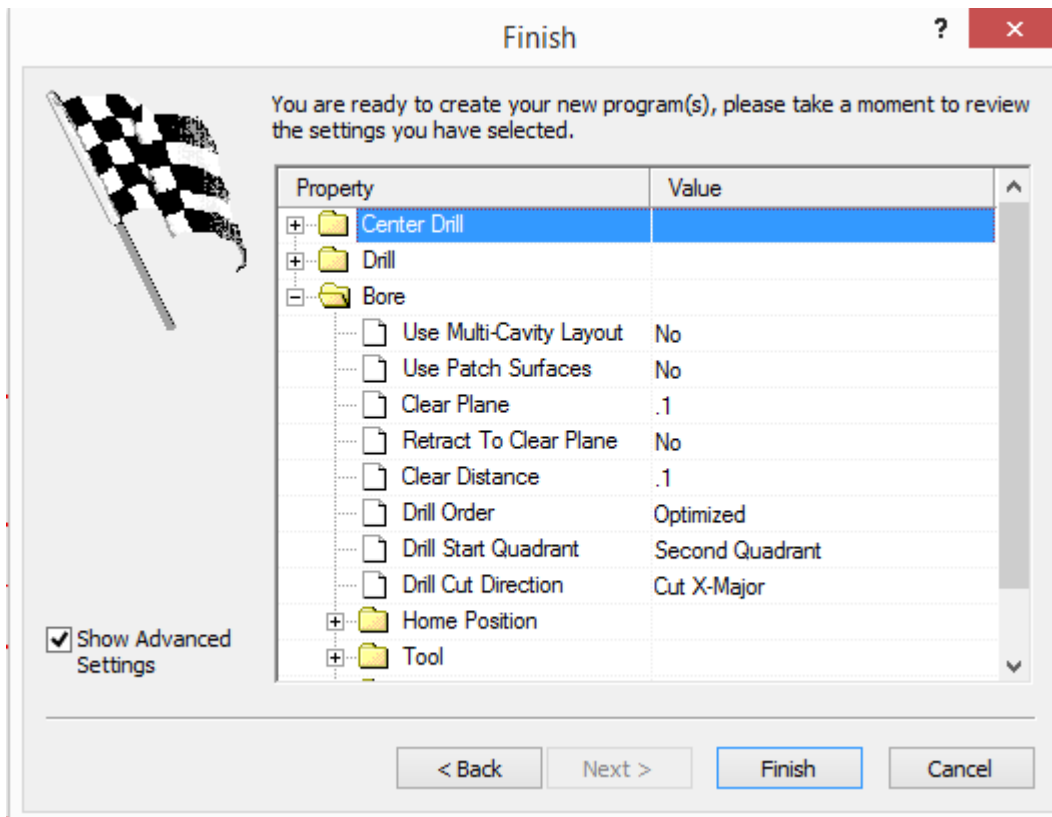


Select the name we used for the Machining Sequence - "Bored Holes." You'll see the description on the right along with the list of Strategies. Click Next. And now we can go through the rest of the wizard. Each Strategy in the list will have its own Tool, Tool Assembly, and Parameters page.

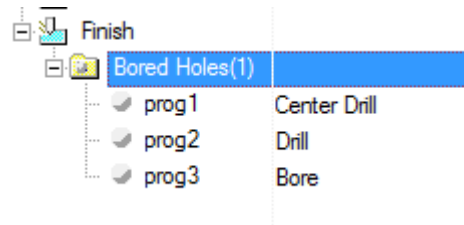


You can quickly jump around using the navigation control on the left. The last page contains a list of all the Strategies with all their

Properties available to edit.



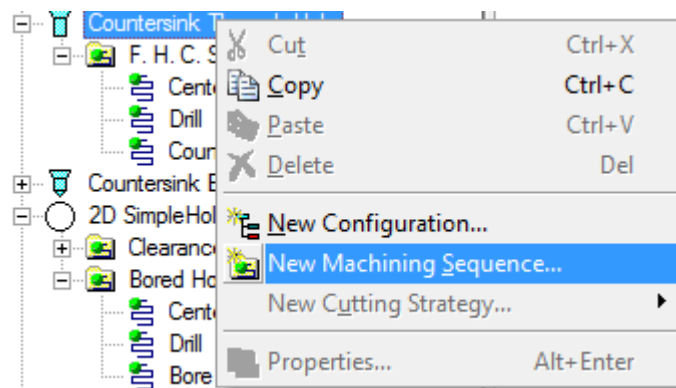
Click Finish. If you haven't used Machining Sequences or Remachining Sequences before, this might look new. Instead of a single Program being added to the tree, we have another level - the Program Group. This represents an instance of the Machining Sequence - it's named Bored Holes (1):



Under that level are the Programs, in order. You cannot change the order of a Program Group. You can Update the Group or you can Update individual Programs in the Group. You can view/change the Properties of the Group or Properties of individual Programs in the Group. Build will generate the Programs in order. Let's take a look at another example. This time for a 3D Feature.

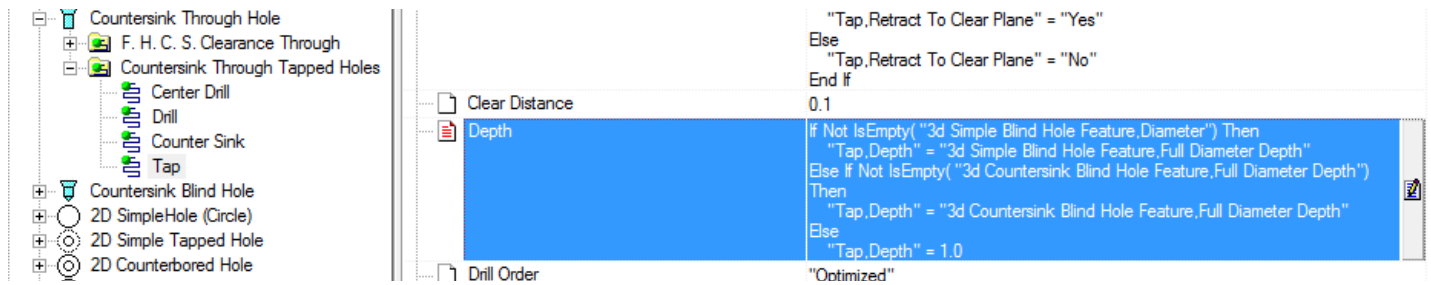
### Example 2 – Countersink Through Hole Machining Sequence with a Tapped Program

Back to Insight. In Cutting Strategy Selection/Mild Steel/Mold Components/Feature-Based Machining, right-click on Countersink Through Hole and select New Machining Sequence:

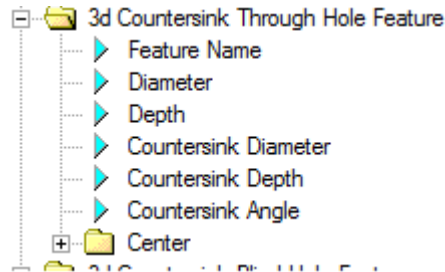


For the name, use "Countersink Through Tapped Holes" and Description "Countersink through holes that need to be tapped." Use

the following Strategies: Center Drill, Drill, Counter Sink, and Tap. For the first three, copy the Rules from the existing FHCS Machining Sequence. Copy the Tap Rules from Basic Machining/Finish. We'll edit the Tap Rules. First, let's change the Depth:



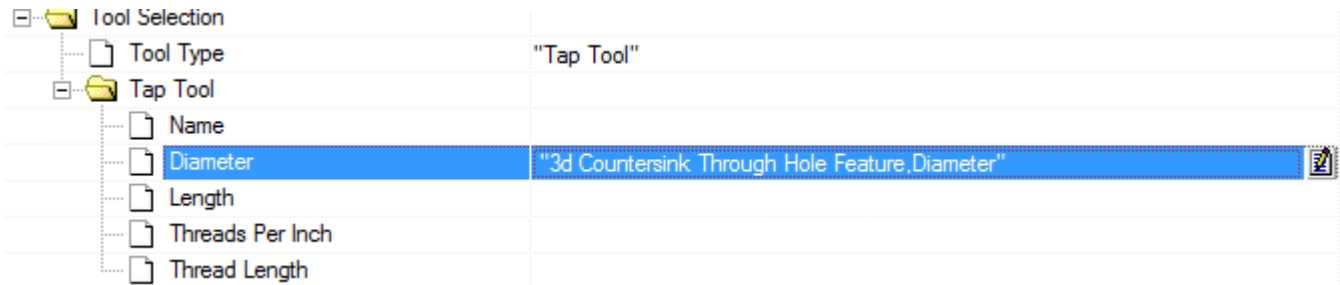
Click in the Rule for Depth and hit backspace to clear the Rule. I want to Tap down an inch, but not from the top of the hole, but from the bottom of the counter sink. Here's how I can do it. Down in the Common Expressions pane, find the "3d Countersink Through Hole Feature" and double click on Countersink Depth:



Now type a space, plus sign (+), another space and the number 1. Hit Enter to save the Rule:



And let's just use the Diameter of the Hole for tool selection. Go down to the Tool Selection area, open up the Tap Tool, clear the Rule for Diameter, double click Diameter down in the Common Expressions pane, and hit Enter. You should see this:

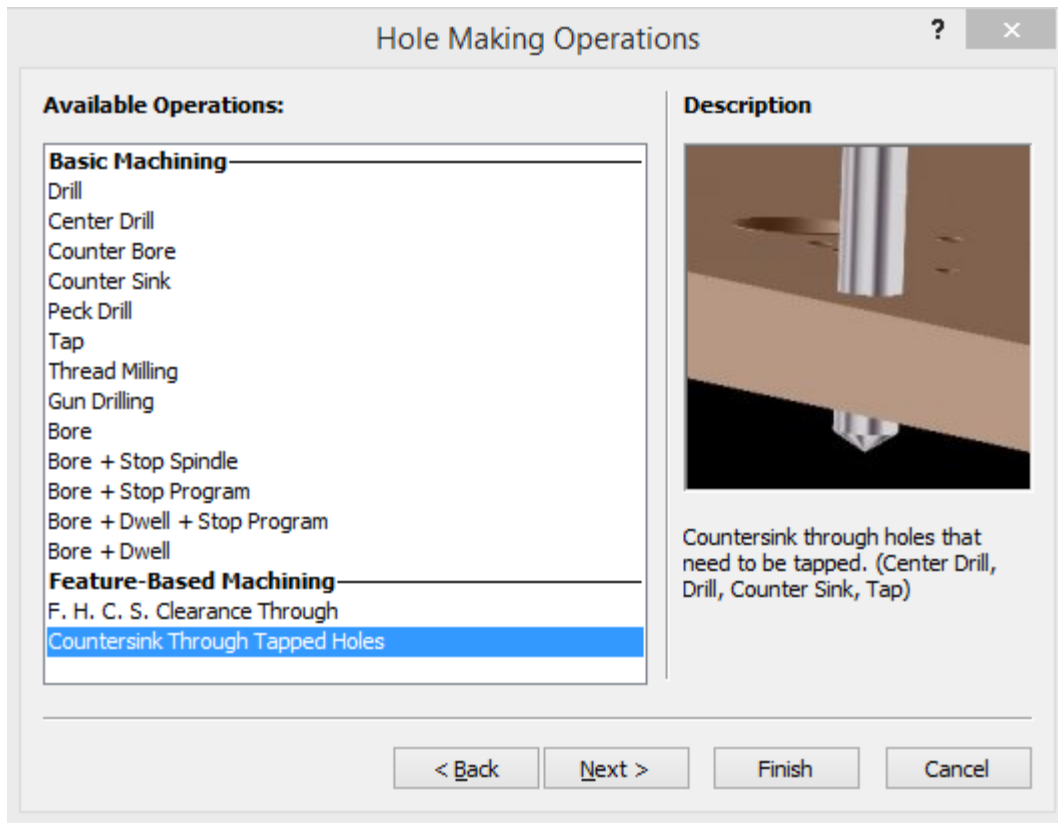


Save the database and return to Prospector to test out the new Machining Sequence.

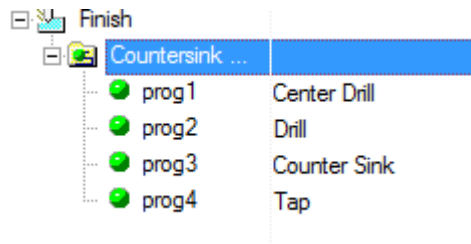
### Prospector

Open up the 3D Hole project. In Finish, new 2D Program, Hole Making. On the feature page, select Countersink Through Hole and click Find Now. Next to the Strategy page and we see our new Machining Sequence:

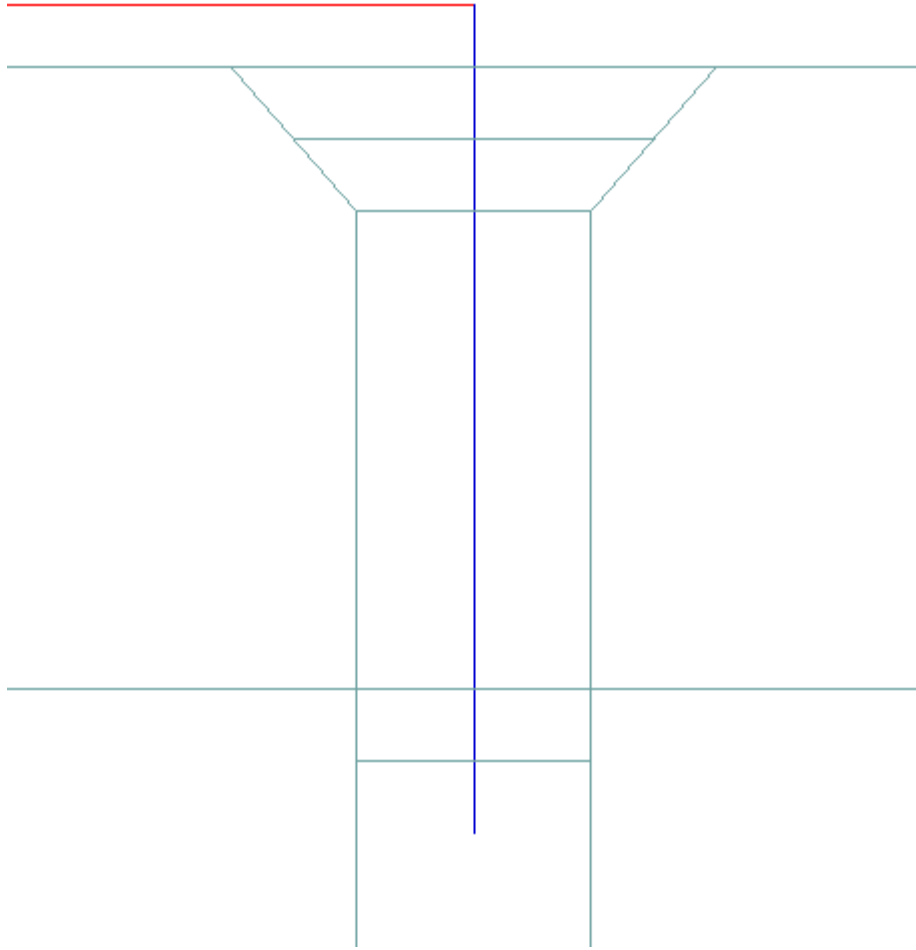




Select it and hit Finish. A new Program Group is added to the Project. This one has four programs:



Let's take a quick look at the Tap program:



Measuring from the bottom of the countersink to the end of the Tap, we see that the Z distance is one inch:

