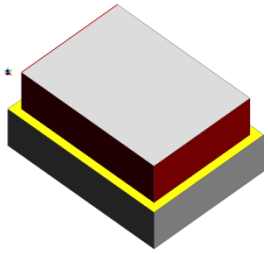
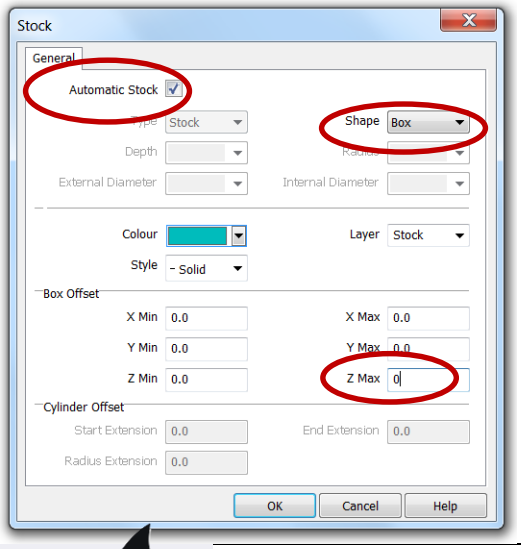
Island Profiling / Contouring



Definition

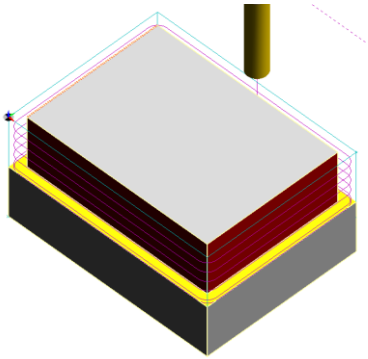
Profiling or contouring is the second most common milling process. It is the process of following along an edge or contour. Profiling or contouring can either follow the inside surface of a pocket or along the outside edge of an island. Or a combination of the two. As with roughing, profiling and contouring can be completed in multiple passes if the depth to be reached is too great to completed in one cycle.

We most commonly compare this type of operation to the process of weed eating or edging a lawn. Many times profiling or contouring is used as a finishing operation to cleanup or finish a surface left behind by a roughing or pocketing operation.



Create the stock for the part

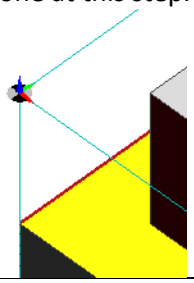
-Select AUTOMATIC STOCK
-Select a SHAPE of [BOX]
-For this activity we will leave all the stock box offsets at ZERO

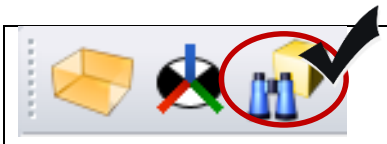


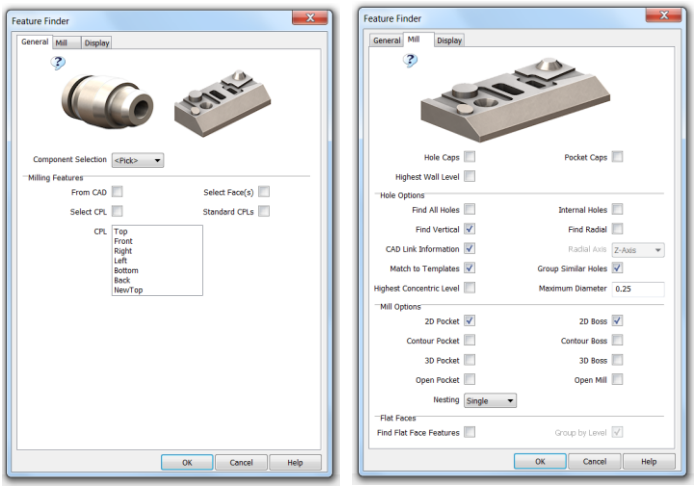
Evaluate and redefine the part reference zero PRZ or CPL

-For this example, the PRZ should not need to be reset. It has been created in Autodesk Inventor with the correct PRZ as we would like to assign it.

Nothing needs to be done at this step.





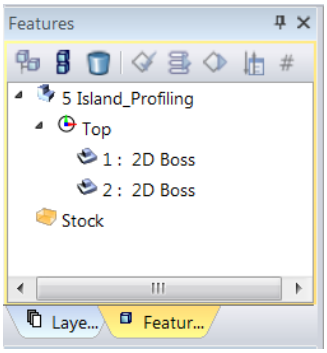


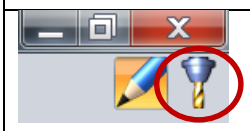
Use the feature finder to allow EdgeCAM to evaluate the 3D CAD model in order to break the model into manufacturable features

Hole Options
Find Vertical
CAD Link Information
Match to Template
Group Similar Holes
Maximum Diameter = The largest tool you have to be able to drill with (Example = 0.25)

Mill Options
2D Pocket
2D Boss

All features found will now be displayed on the features tab



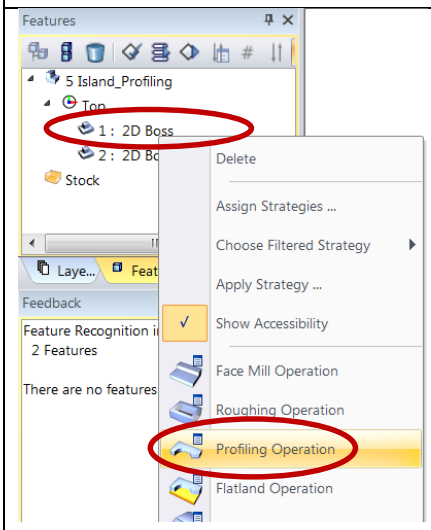


Leave DESIGN mode and enter MANUFACTURE mode

DISCIPLINE – [Mill]

MACHINE TOOL – Choose the correct POST for your machine [intelitek.mcp]

INITIAL CPL – Ensure the CPL is [Top]



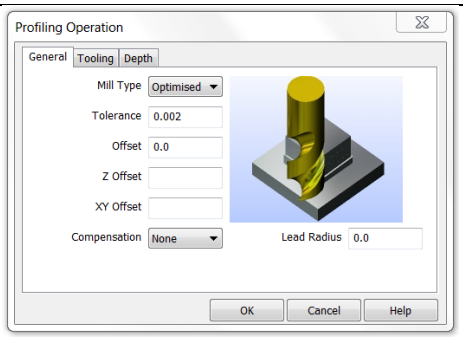
Switch from the SEQUENCE tab back to the FEATURES tab

-Select the 1st 2D BOSS from the features list.

Note: As you hover over the features, they should highlight the corresponding feature on the 3D model.

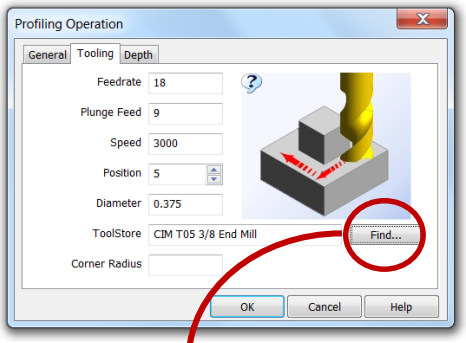
Right click on the first 2D BOSS and select PROFILING Operation from the list

At this time we do not need to define any boundaries for this operation. Select either select the check mark in the top right corner, select enter on the keyboard, or right click to select DONE



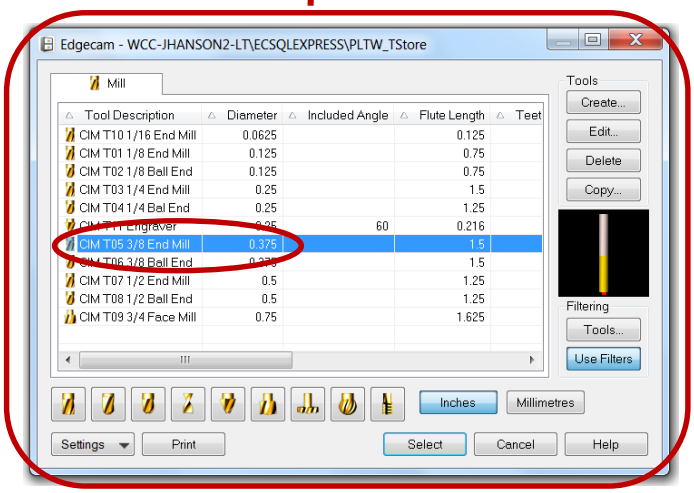
GENERAL

-MILL TYPE – Either select [CLIMB]
-% STEPOVER – [50%]
-OFFSET – [0] (Used to leave material on all axis (X, Y and Z). A finishing operation can then be used to remove the offset material
-Z OFFSET – [0] (Used to leave material on only the Z Plane of the surface defined.)
-XY OFFSET – [0] (Used when an offset along only the X and Y axis are desired.)
-TOLERANCE – [0.001 to 0.005]
-COMPENSATION – [None]



TOOLING

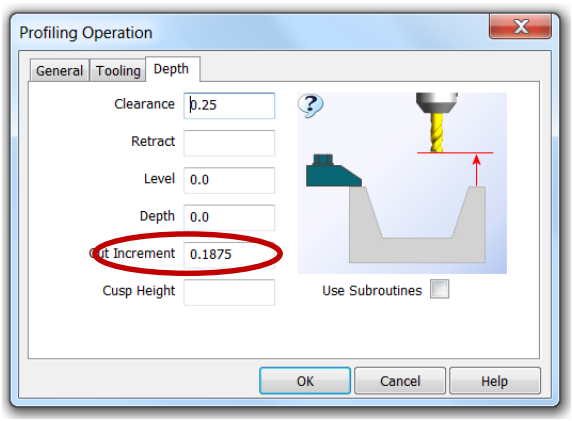
-1ST Choose the FIND box (choose the largest end mill tool you have available in your library or have available at you machine)
-2nd Ensure the FEEDRATE, PLUNGE RATE, and SPEED are correct for the material and the tool you are using.

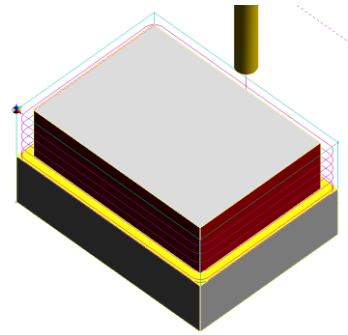


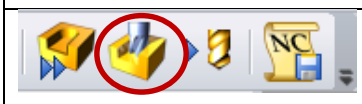
DEPTH

-CLEARANCE – [0.25] - Retract point above the PRZ
-LEVEL – [0] – Measured from the top of the feature selected
-DEPTH – [0] - Measured from the bottom of the feature selected
-CUT INCREMENT – [0.1875] - determine a safe number for the tool you are using to go from the level to the depth. If the Cut Increment is smaller than depth, the facing operation will produce multiple passes to go from the level to the depth.

For prototyping material, use approximately ½ to 1/3 the bit diameter as the cut increment

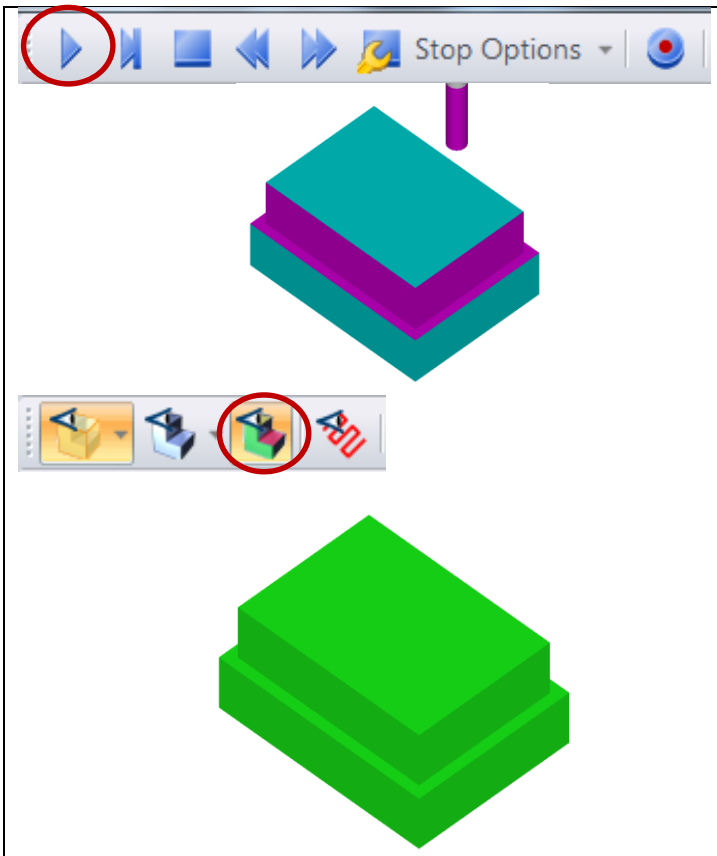






Simulate the machining process

Once the simulator opens, Select PLAY

	Ensure the simulator produce the results desired Select the COMPARE tool – this option will compare the original 3D model and the results of the simulated operations. At this time since we raised the stock above the 3D model, set the facing operation to only cut the amount raised from the 3D model, the compare should show as all green GREEN – Simulation and CAD model areas that Match **BLUE** – Areas not removed or remaining from the stock to produce the 3D model **RED** – Areas removed from the stock that have cut into the 3D model Close the simulator to return to EdgeCAM
	**Note: If EdgeCAM is running in HOME WORK mode, producing the NC code is not available* Select NC tool from the tool bar to produce the NC code using the POST Processor selected earlier