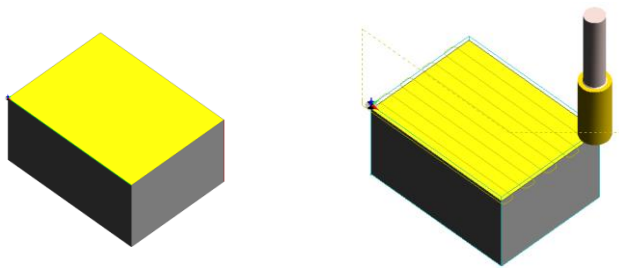# Facing

Definition

Face milling is the act of milling a surface perpendicular to the axis of rotation of the cutter. A face mill is a cutter specifically designed for this purpose.

www.CNCCookBook.com

Face milling is the most common milling operation and can be performed using a wide range of different tools. Cutters with a 45º lead angle are most frequently used, but round insert cutters, square shoulder cutters and side and face mills are also used for certain conditions.

http://www.sandvik.coromant.com

Application checklist and hints

Roll into cut
Consider machine tool stability, spindle size and type (vertical or horizontal) and available power.
Consider maximum chip thickness when positioning the cutter for optimum feed.
Position the cutter off center to produce the thinnest chip at exit.
Apply climb milling for favorable chip formation, i.e. thick to thin chip.
Program the cutter to roll into the cut and reduce the feed to obtain a smooth entry.
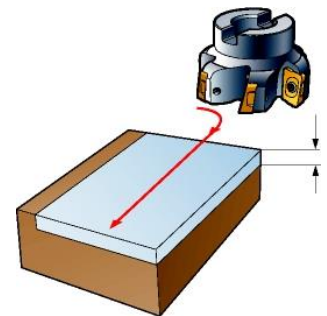
Keep cutter constantly engaged.
Avoid entries and exits through tool path programming.
Frequent entering and exiting the workpiece should be avoided if possible. It can create unfavorable stresses on the cutting edge, or cause dwelling and chatter tendencies. It is recommended that you program a tool path that keeps the milling cutter in full contact, rather than performing several parallel passes. When changing direction, include a small radial tool path to keep the cutter moving and constantly engaged.
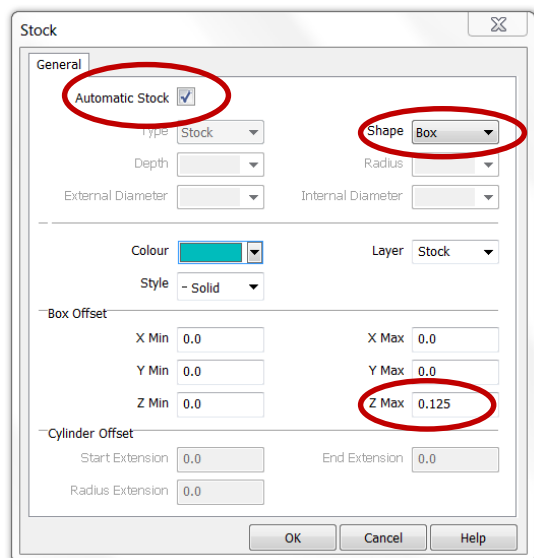Use a cutter diameter that is 20 to 50% larger than the workpiece.

http://www.sandvik.coromant.com
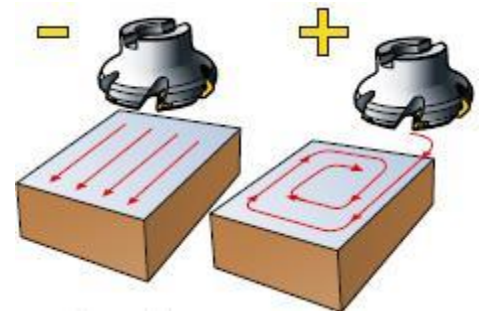
Face milling is most often completed with an end mill, a shell mill, or a fly mill
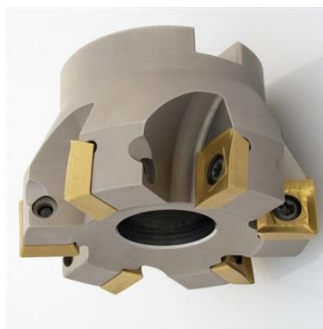
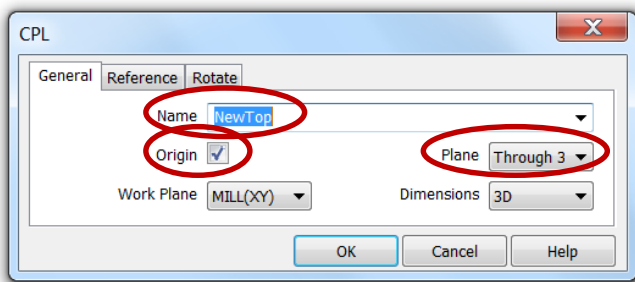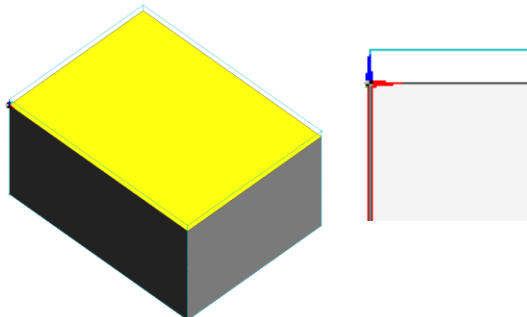*Create the stock for the part*

-Select AUTOMATIC STOCK
-Select a SHAPE of [BOX]
-Determine how much material is to be removed from the stock before the 3D model starts
BOX OFFSET - For this example we will set the stock up an additional 1/8 of an inch [0.125] to the Z MAX

After the stock is create, you now see additional material above the top surface


Evaluate and redefine the part reference zero PRZ or CPL

-Determine a NAME for the new PRZ/CPL [NewTop]
-Select ORIGIN to be able to reset the home Z
-Choose [THROUGH 3 POINTS] for the plane definition

To reassign the PRZ, through 3 points, you must select the following points in the correct order

*Ensure EdgeCAM represents your selection with a red box (Endpoint) or red X (Midpoint) before making the selection*

#1 – Select what will be the new origin
#2 – Select a point along the new X positive (either the corner or the midpoint of the Positive X Axis
#3 – Select a point along the new Y positive (either the corner or the midpoint of the Positive Y Axis
#4 – Select the origin corner
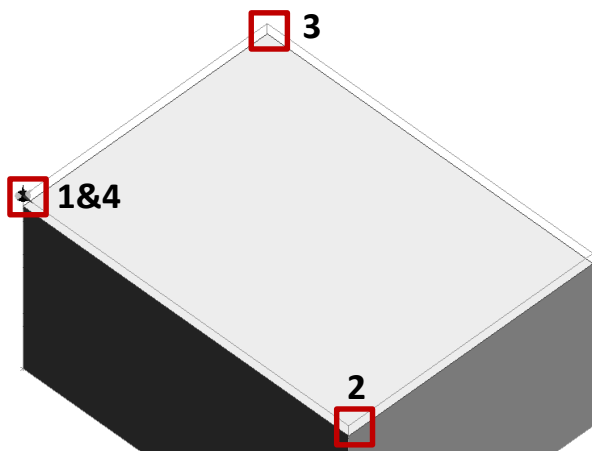
XYZ = RGB (Red / Green / Blue)
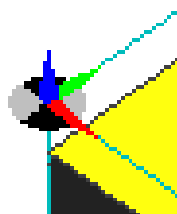
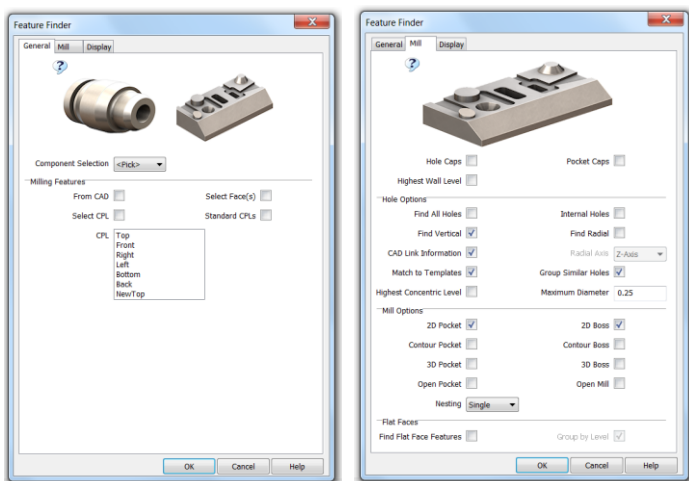Each point should now be pointing in the positive direction along the axis of the origin point
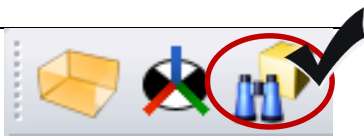
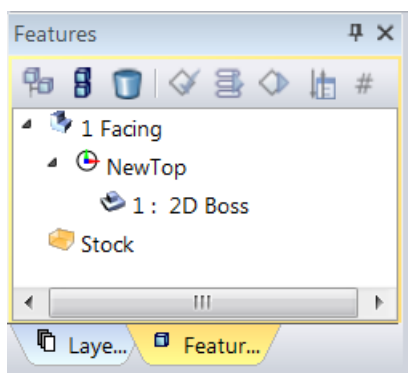|	
	Use the feature finder to allow EdgeCAM to evaluate the 3D CAD model in order to break the model into manufacturable features **Hole Options** Find Vertical CAD Link Information Match to Template Group Similar Holes Maximum Diameter = The largest tool you have to be able to drill with (Example = 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 has your new CPL selected [NewTop]
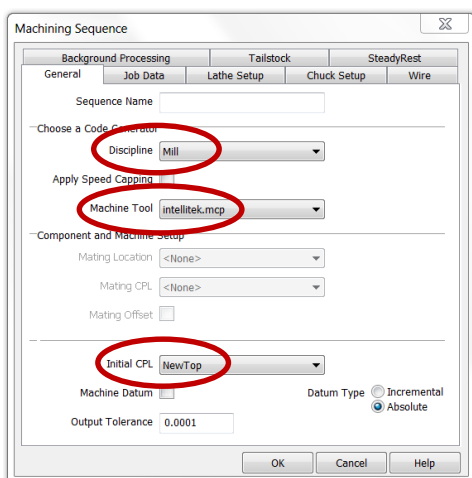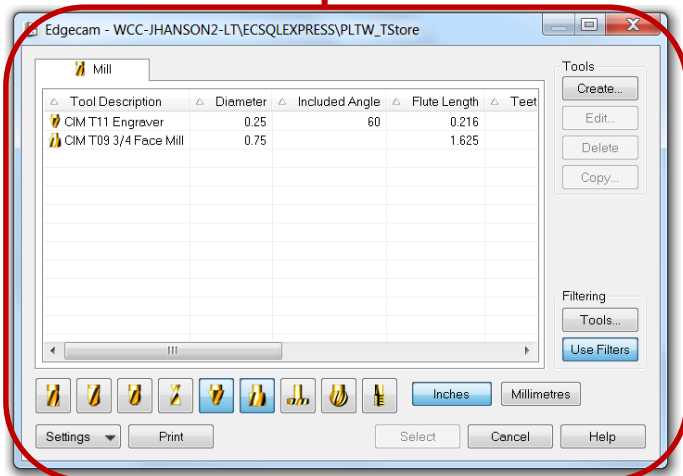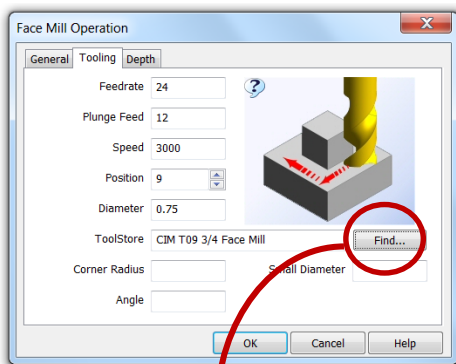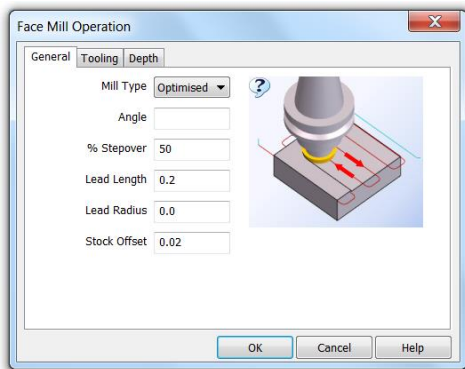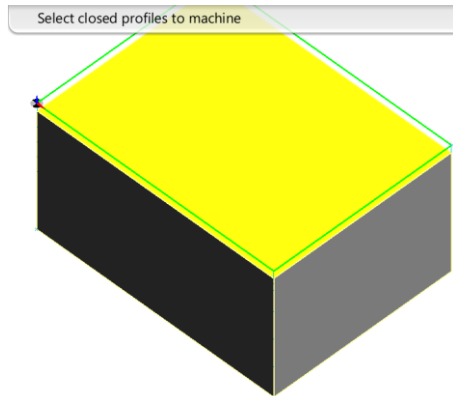
Select the FACING OPERATION from the top tool bar

-Select the top four edges of the stock boundary and either select the check mark in the top right corner, select enter on the keyboard, or right click to select DONE
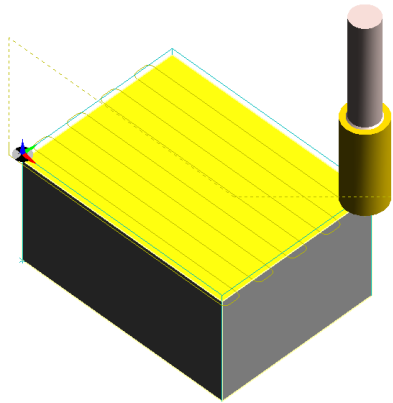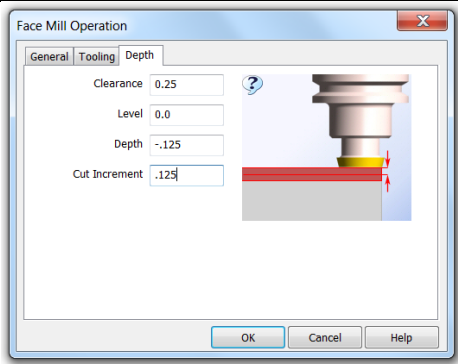
GENERAL
-MILL TYPE – Either select [CLIMB] or [OPTIMISED]
-% STEPOVER – [50% - 70%]
-LEAD LENGTH – [0.2]
-LEAD RADIUS – [0]
-STOCK OFFSET – [0.02]

TOOLING
-1ST Choose the FIND box (choose the largest face mill or 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 PRZ
-DEPTH – [-0.125] - the amount of material that is to be removed from the level. Negative numbers will be cut below the origin
-CUT INCREMENT – [0.125] - 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.
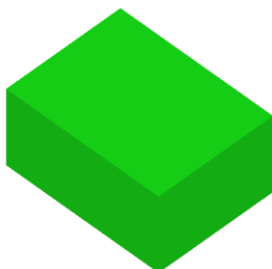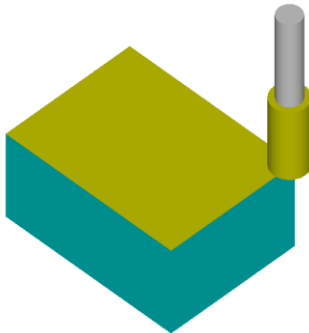
Simulate the machining process

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