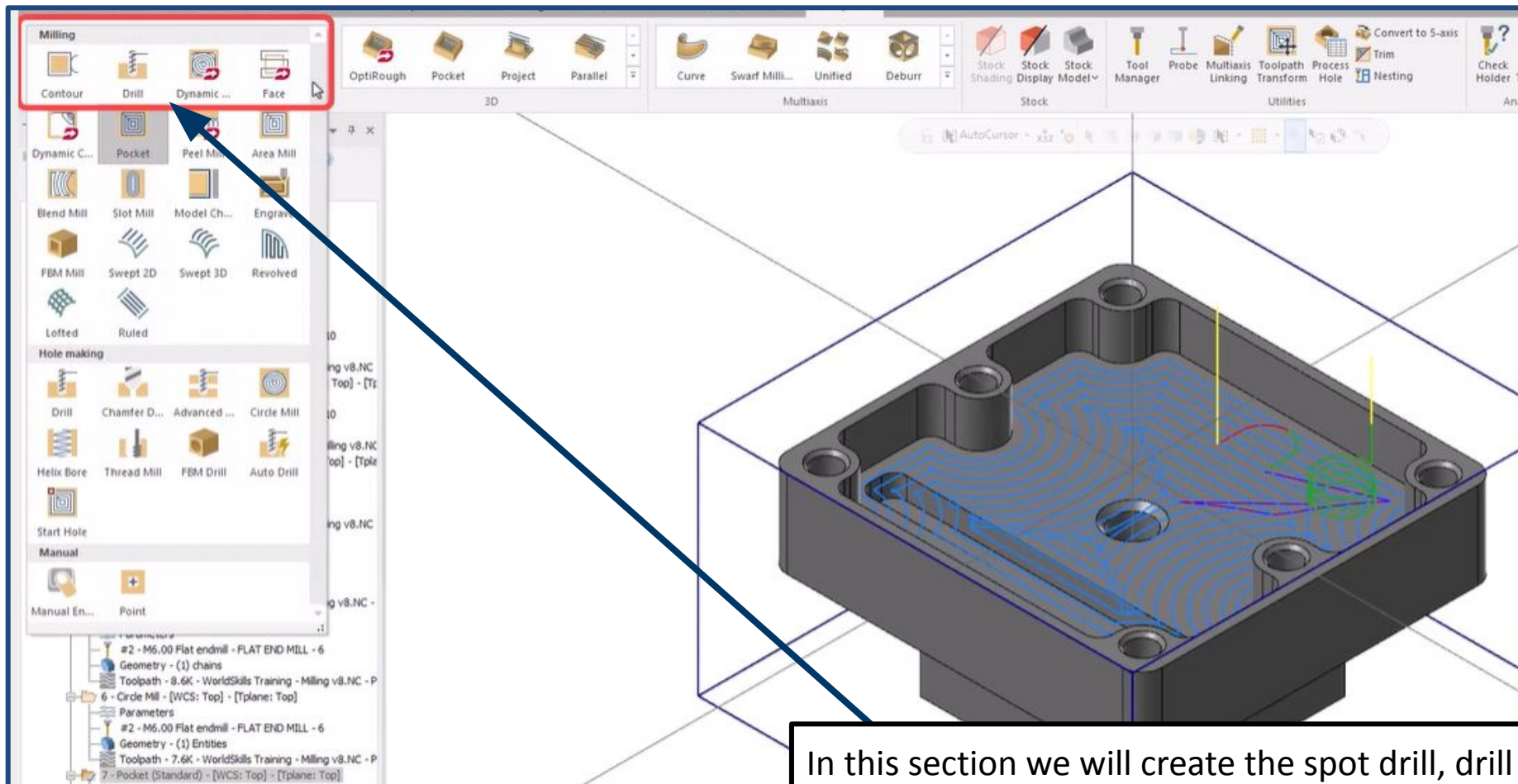


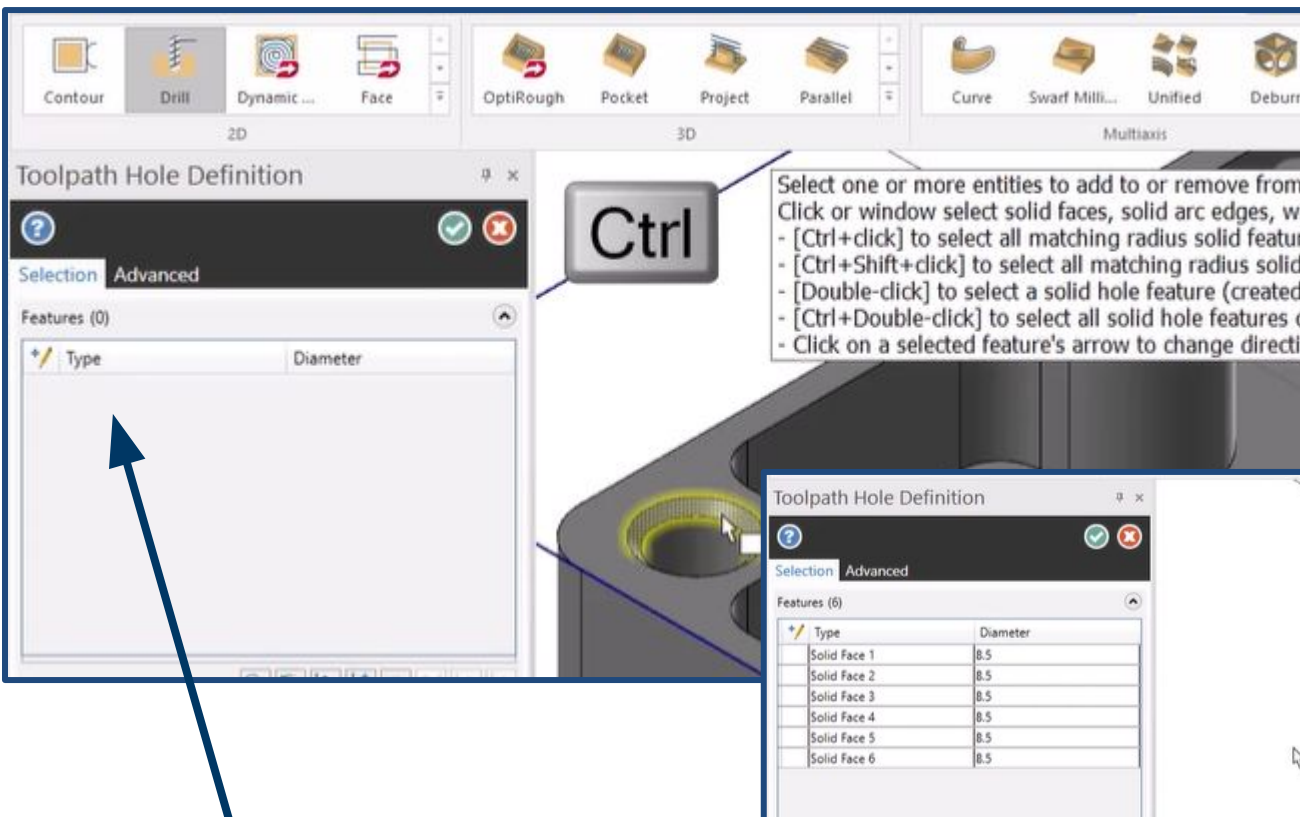
Mastercam Training Video Series

Video 4 - Hole Making Operations

[Video Link](#)

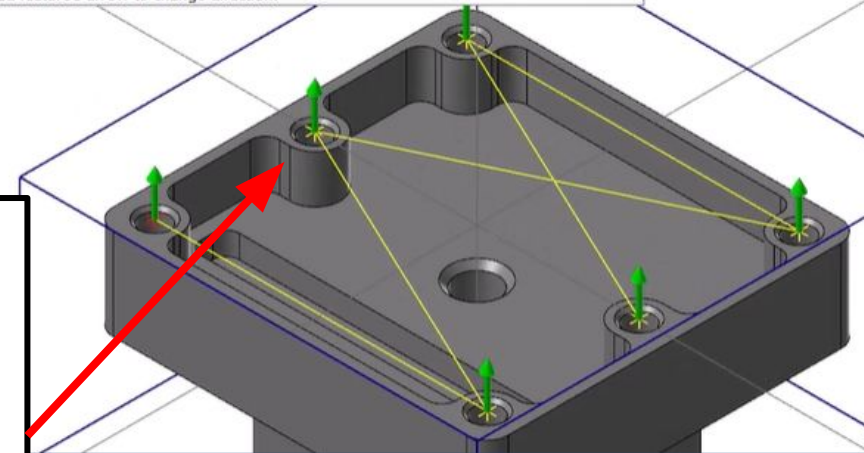


In this section we will create the spot drill, drill and tapping operations using the Drill toolpath. Click on the Drill toolpath icon to start



Select one or more entities to add to or remove from the Features list. Click or window select solid faces, solid arc edges, wireframe arcs, lines, points, or AutoCursor positions.

- [Ctrl+click] to select all matching radius solid features.
- [Ctrl+Shift+click] to select all matching radius solid features on the same vector as the initial selection.
- [Double-click] to select a solid hole feature (created using the Solids Hole function).
- [Ctrl+Double-click] to select all solid hole features of the same type.
- Click on a selected feature's arrow to change direction.



The Hole Definition box appears.

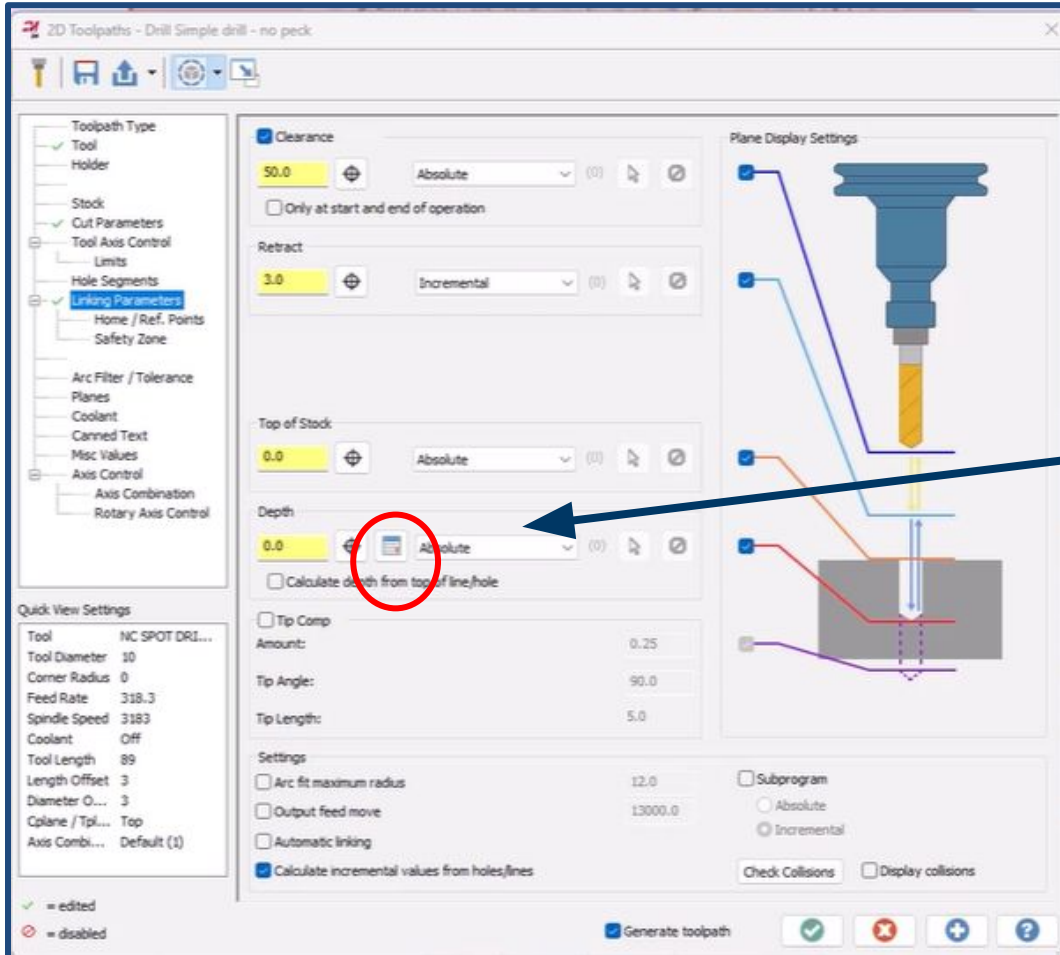
Press and hold the “Ctrl” key while clicking one of the required hole edges.

This is a shortcut to select all holes with the same radius.

You can use the sort option to change the order in which the holes are drilled.

Press OK to accept and move onto the toolpath parameter page

-
- 2D Toolpaths - Drill Simple drill - no peck
- Toolpath Type
- ✓ Tool
 - Holder
 - Stock
 - Cut Parameters**
 - Tool Axis Control
 - Limits
 - Hole Segments
 - Linking Parameters
 - Home / Ref. Points
 - Safety Zone
 - Arc Filter / Tolerance
 - Planes
 - Coolant
 - Canned Text
 - Misc Values
 - Axis Control
 - Axis Combination
- Cycle: **Drill/Counterbore**
- First peck: 0.0
- Subsequent peck: 0.0
- Peck clearance: 0.0
- Retract amount: 0.0
- Dwell: **3.0**
- Shift: 0.0
- ☐ Apply custom drill parameters
- 1-Drill parameter #1: 0.0
- 1-Drill parameter #2: 0.0
- 3D model of a drill bit.



On the 'Linking Parameters' page set:

- Clearance to 50.0 Absolute
- Retract to 3.0mm Incremental
- Top of Stock to 0.0mm Absolute

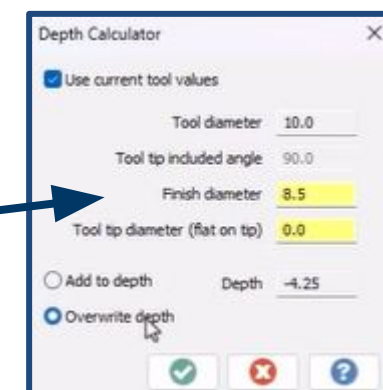
On the Depth selection set Absolute and then press the calculator button.

This opens its own dialogue page.

Set:

- Finish Diameter to: 8.5
- Select 'Overwrite depth'

This function uses the tool data to work out what programmed drill depth is required to achieve a particular finish diameter - In this case it will leave a spot drill diameter of 8.5mm on the part face Press Green Tick to accept



TOP TIP!

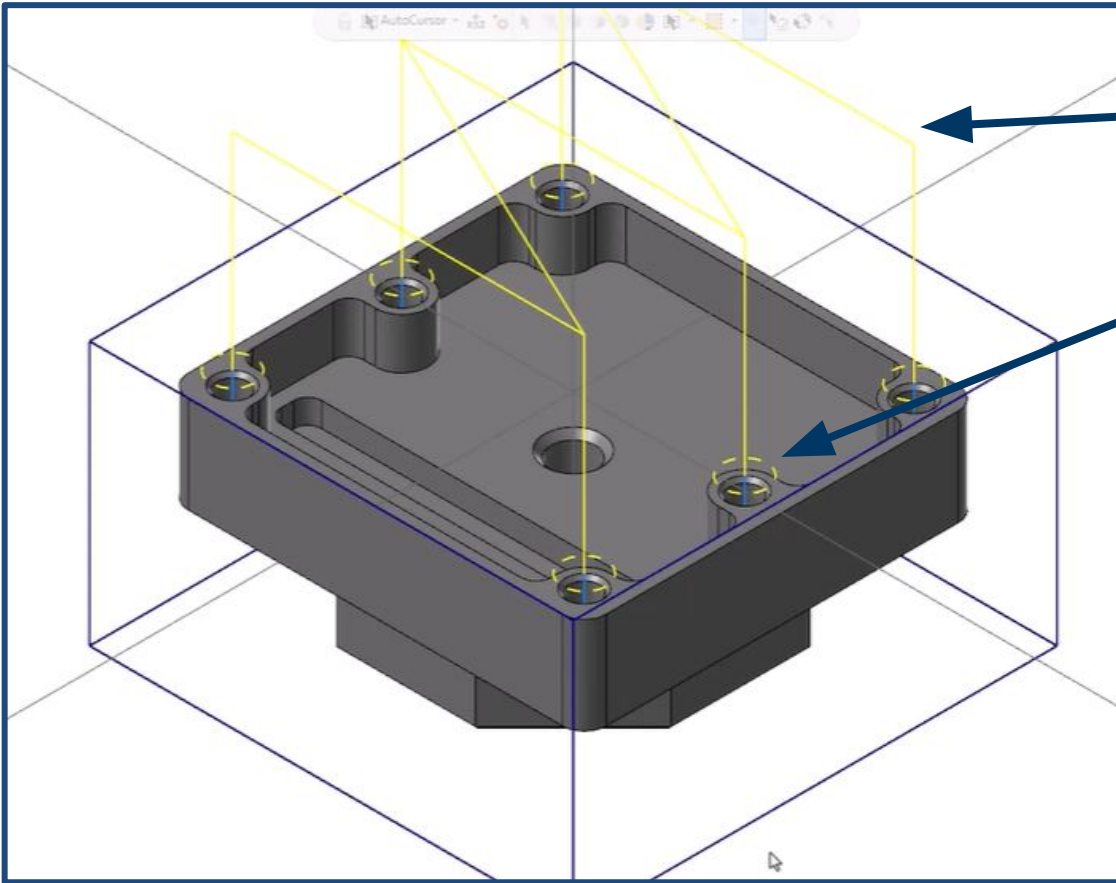
On Mastercam Linking parameters remember:

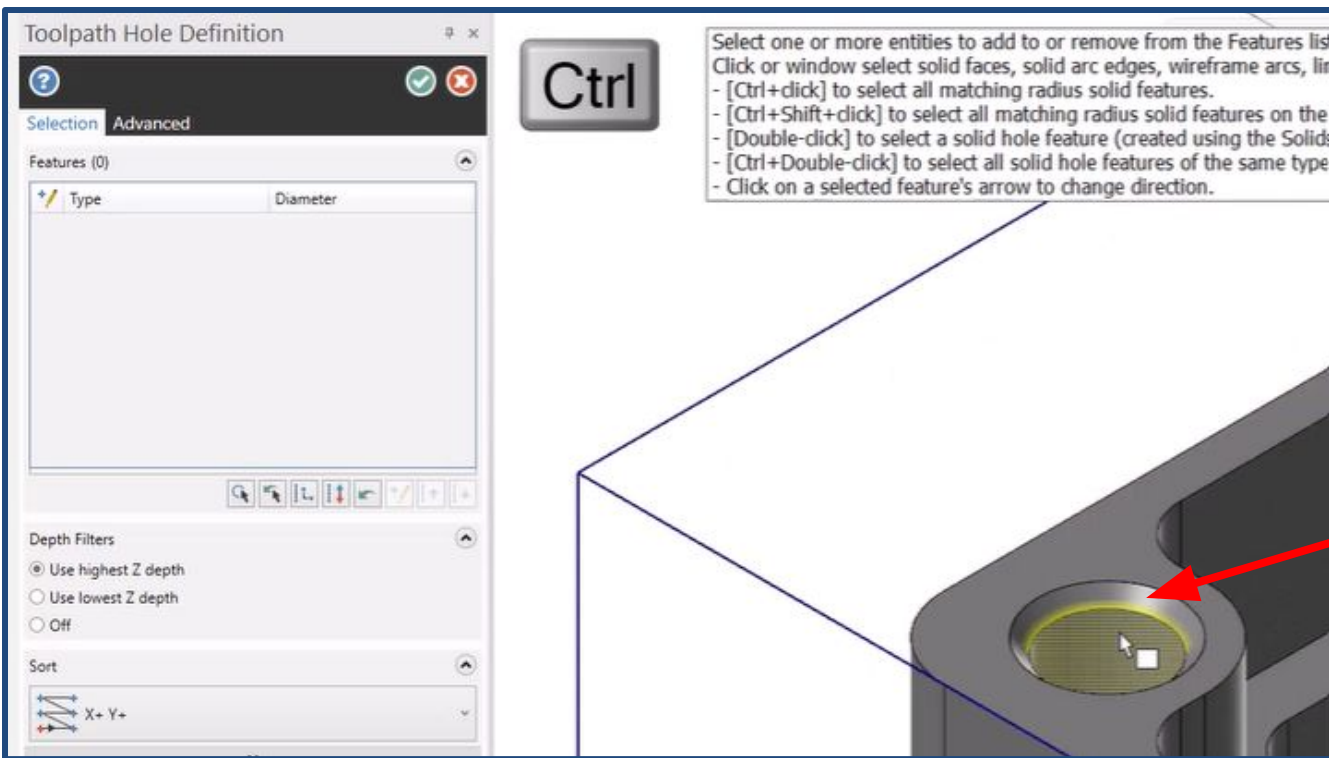
Absolute - This means that the height will be calculated from the active plane

Incremental - This means the height will be calculated from the geometry chosen for the toolpath - in this case the top edge of the holes

The tool path should look similar to the one shown:

- These top heights are 'Clearance' heights and will be 50mm above the active plane / WCS
- These circles represent the retract heights, these are set at 3mm above the Absolute Z zero position. The Drill will rapid to this point before feeding down to the specified depth.

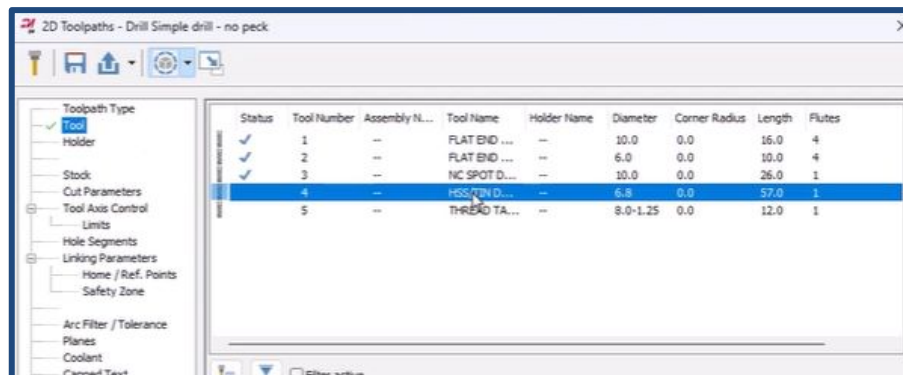


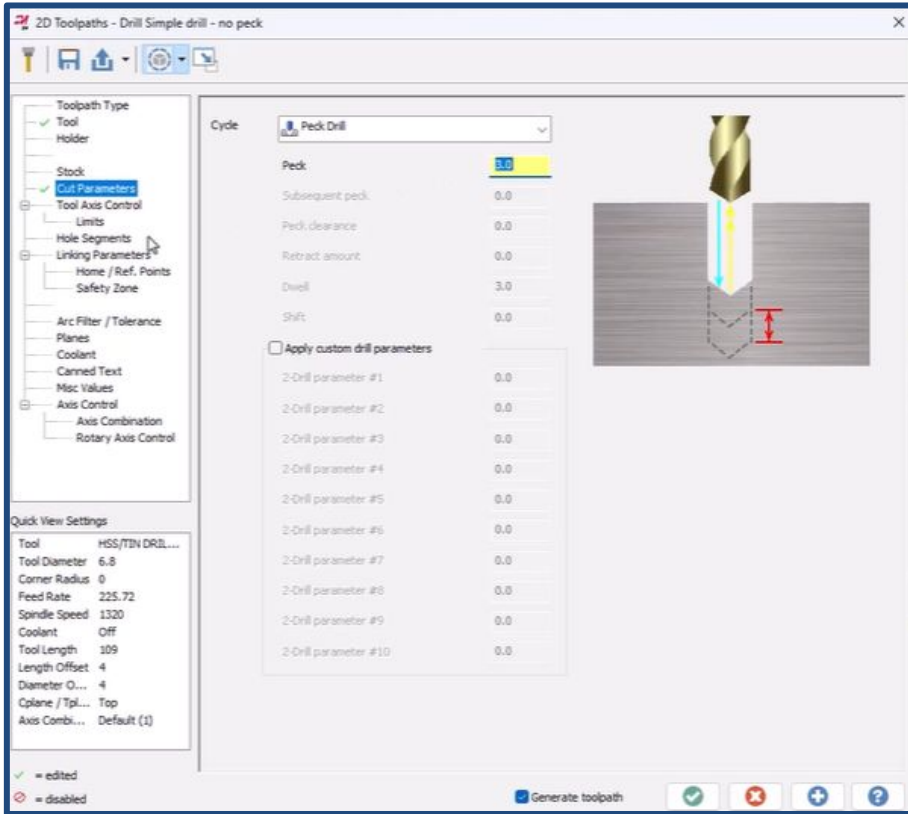


Now select the drill toolpath again

Select the drilled hole body face while holding down the 'Ctrl' key to select all similar holes.

Select the 6.8 Drill from the tool page



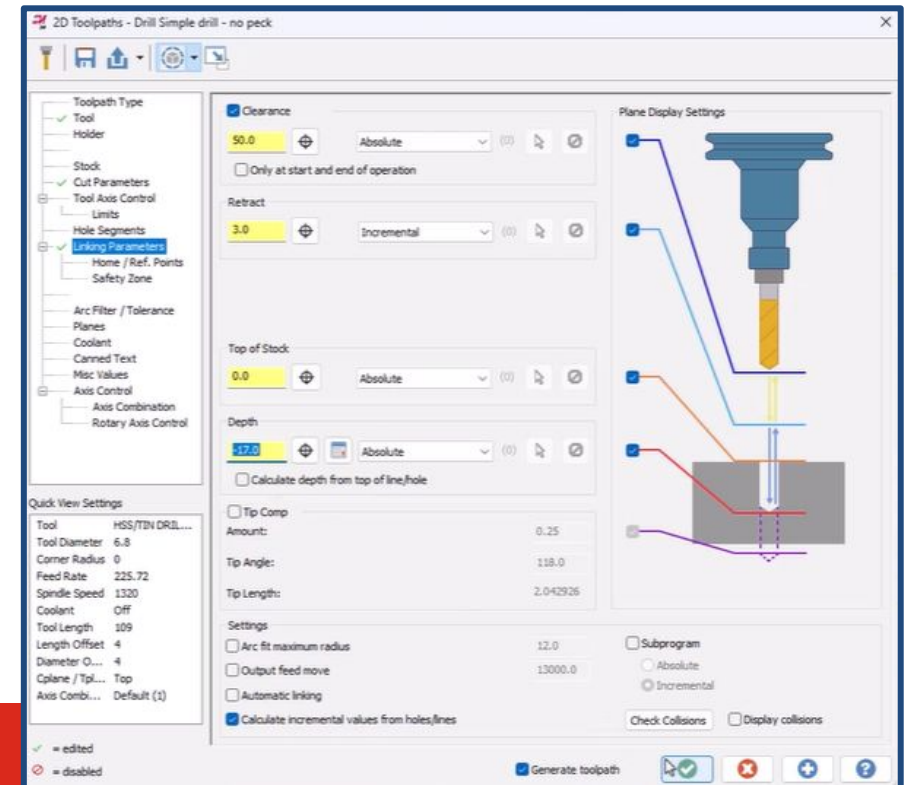


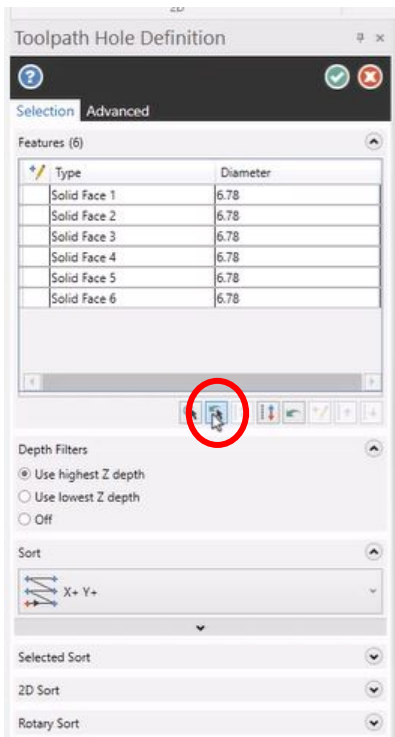
On the 'Cut Parameters'

- Change Cycle to 'Peck Drill'
- Set Peck to 3.0

On the 'Linking Parameters'

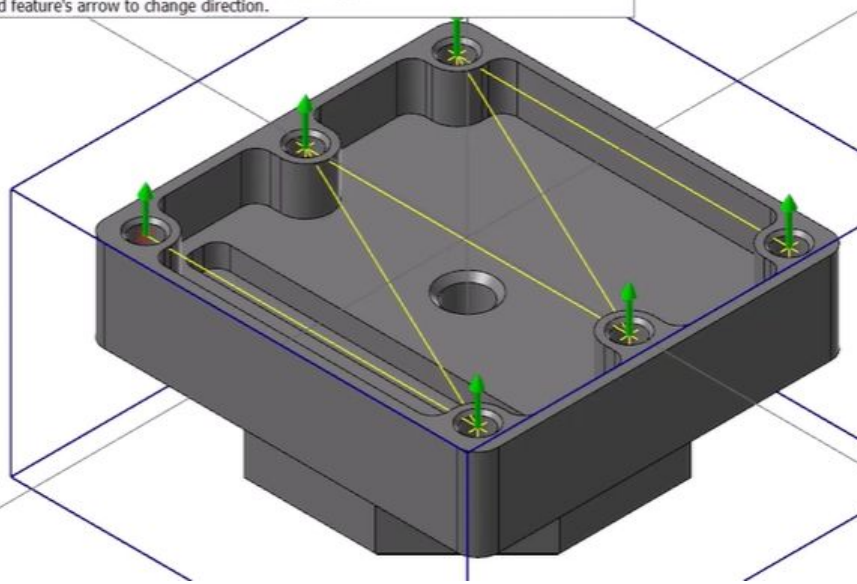
- Set depth to -17.0
- Green Tick to accept



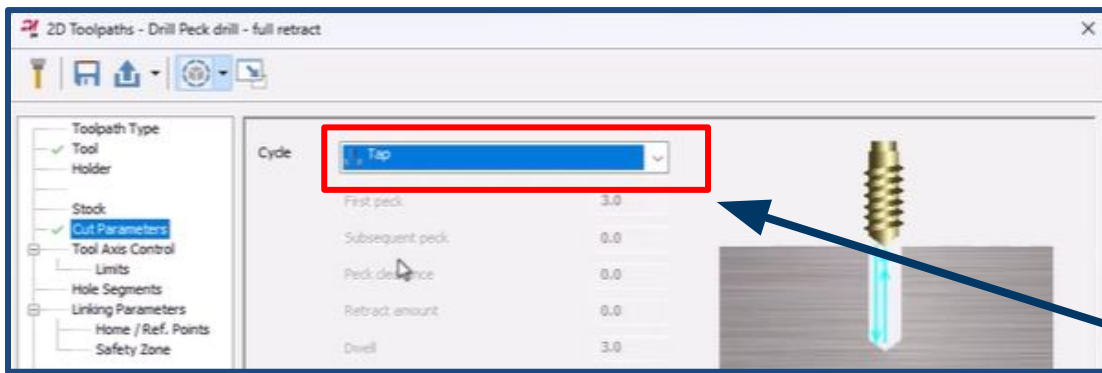


Select one or more entities to add to or remove from the Features list.
 Click or window select solid faces, solid arc edges, wireframe arcs, lines, points, or AutoCursor positions.

- [Ctrl+click] to select all matching radius solid features.
- [Ctrl+Shift+click] to select all matching radius solid features on the same vector as the initial selection.
- [Double-click] to select a solid hole feature (created using the Solids Hole function).
- [Ctrl+Double-click] to select all solid hole features of the same type.
- Click on a selected feature's arrow to change direction.

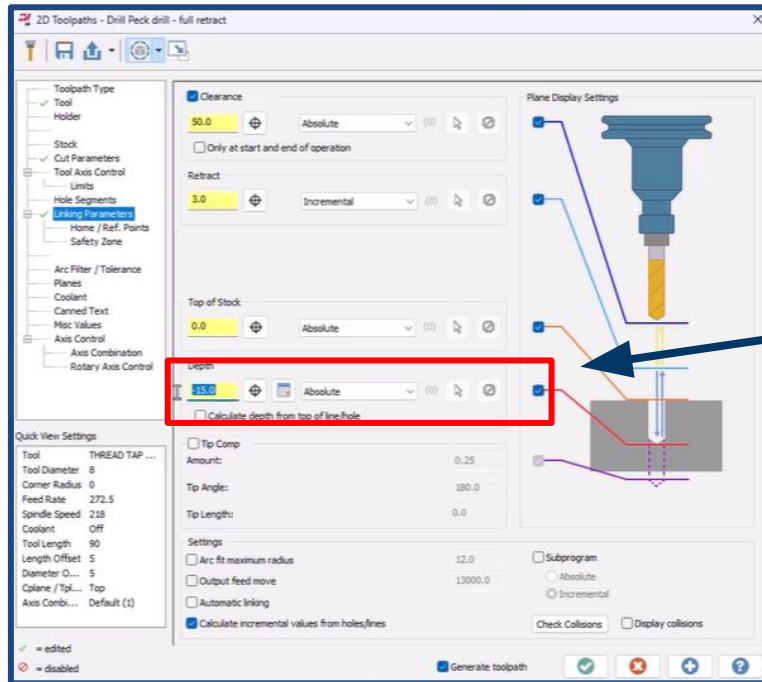


Now to the tapping - This is accessed from the drilling toolpaths - Select another Drilling Toolpath
 When the Hole Definition box appears use the 'Copy Previous Points' button to quickly select the same holes used in the previous drilling toolpath.
 Green Tick to accept the selection



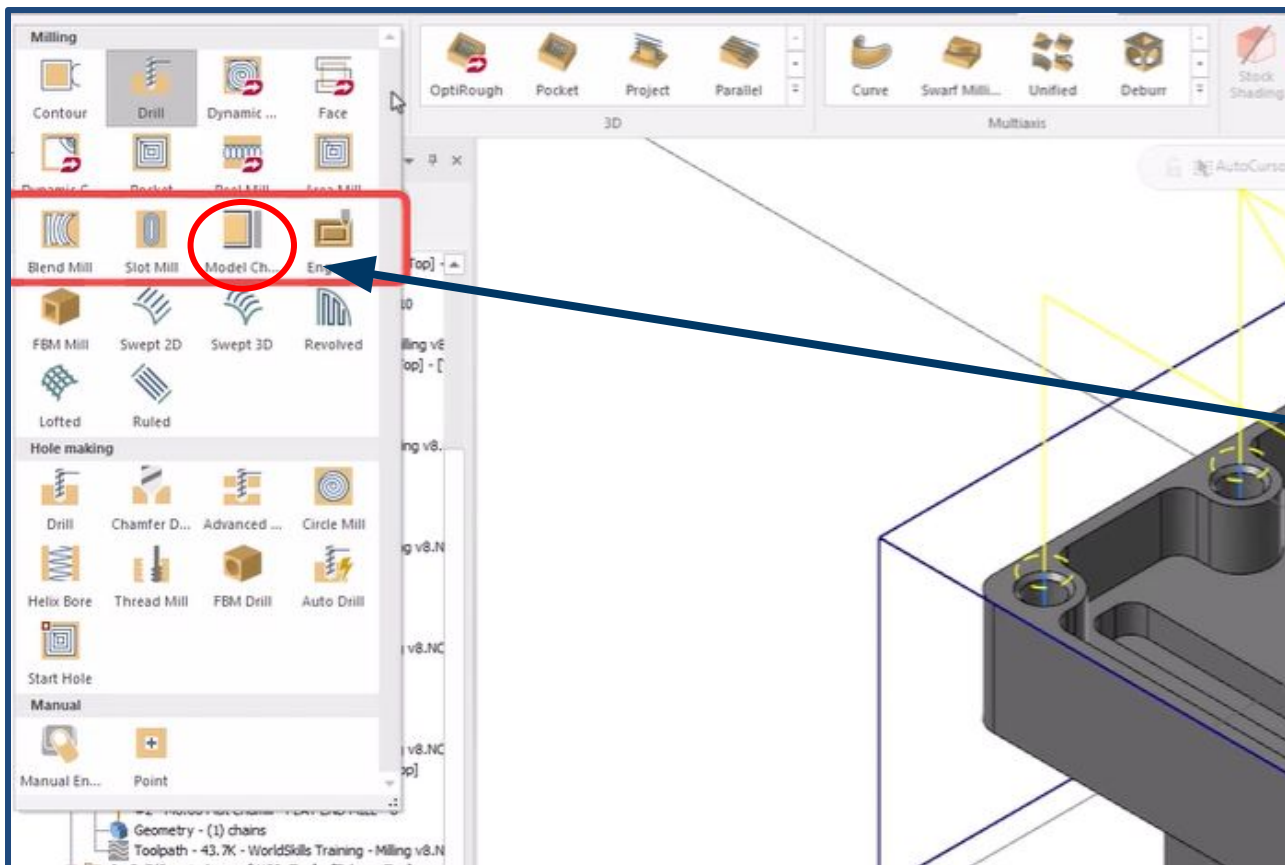
On the tool page select the M8 Tap and use the manufacturers recommended speeds

On the 'Cut Parameters' page use the dropdown to change the cycle to 'Tap'



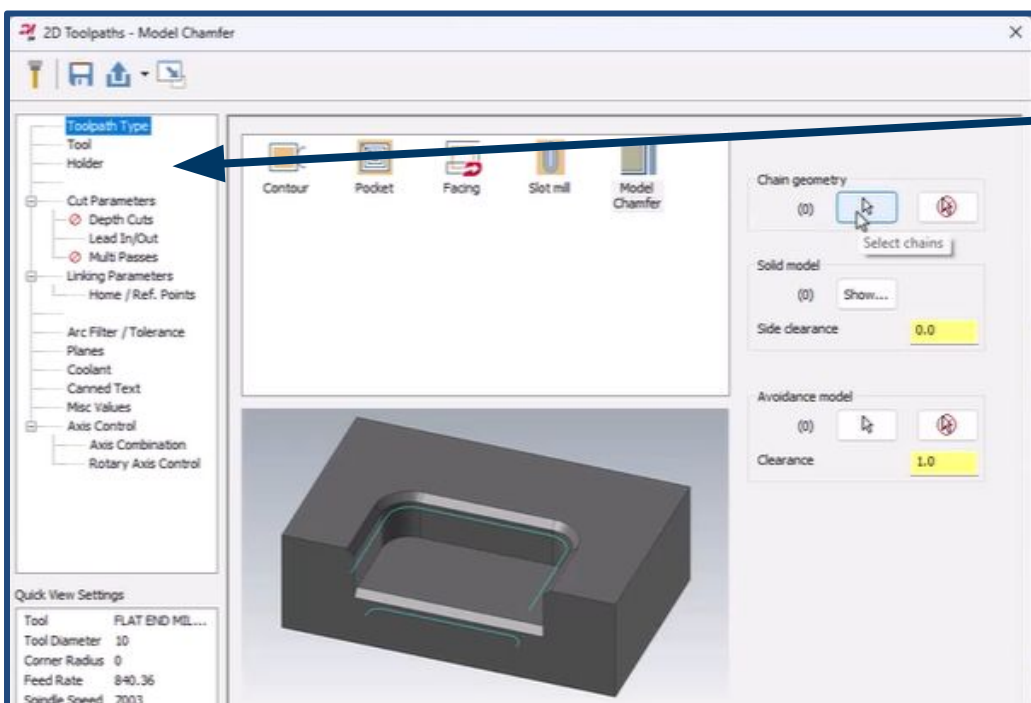
On the 'Linking Parameters' page Select 'Depth' as -15.0 This is taken from the component drawing.

Green Tick to accept and generate the toolpath

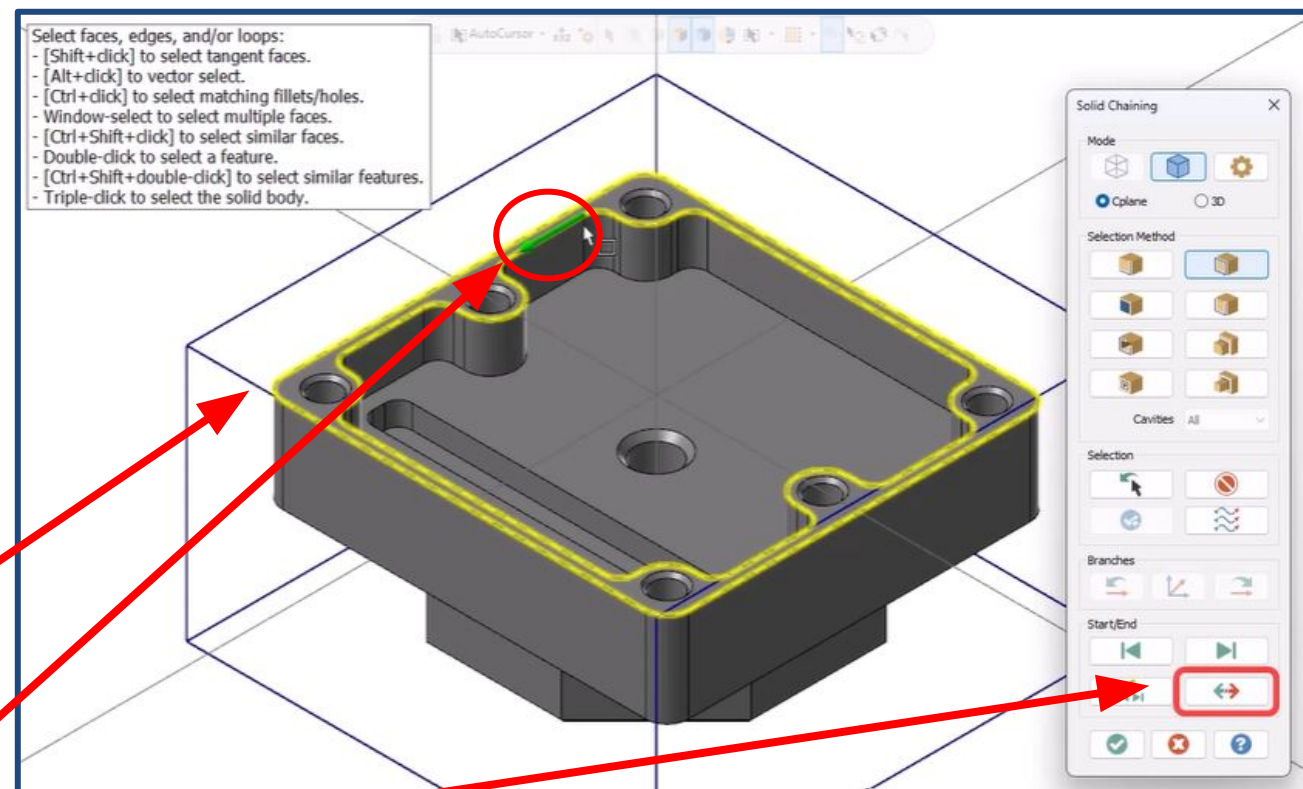


We will now deburr the part with a small break edge using the 'Model Chamfer' toolpath.

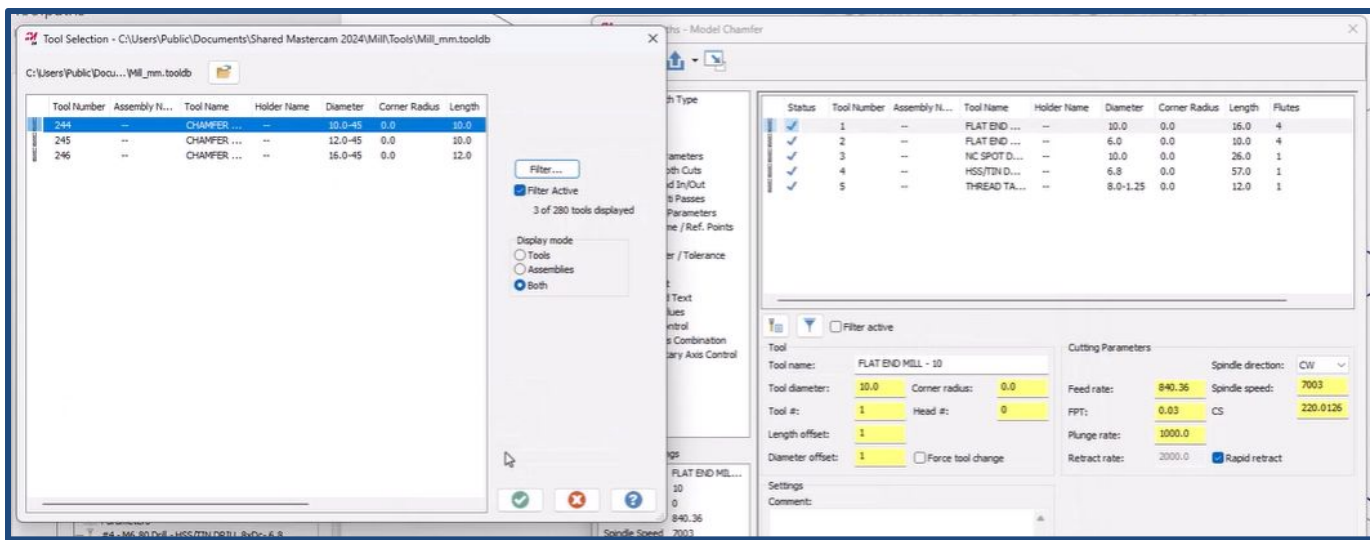
Expand the 2D toolpath selection frame and select the 'Model Chamfer' toolpath



On toolpath parameter page use the 'Chain Geometry' option to select the edges to chamfer

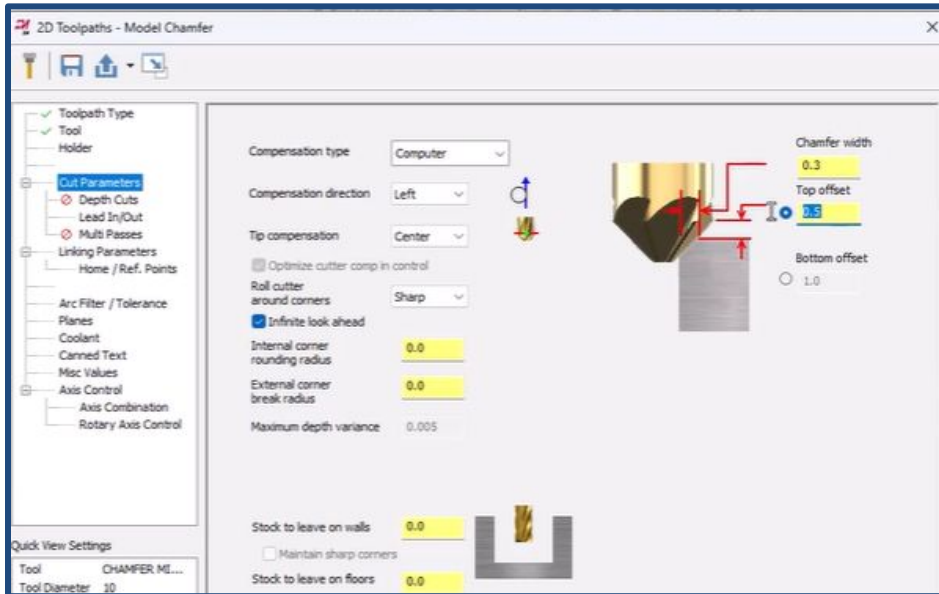


On the Chaining dialogue box
Select Solid Selection Method
Select edge selection
Click on the Outside edge and pocket inner edge
shown
Ensure the direction arrow is pointing in the correct
direction (Climb Milling)
Use the 'Reverse' function if needed
Green Tick to accept.



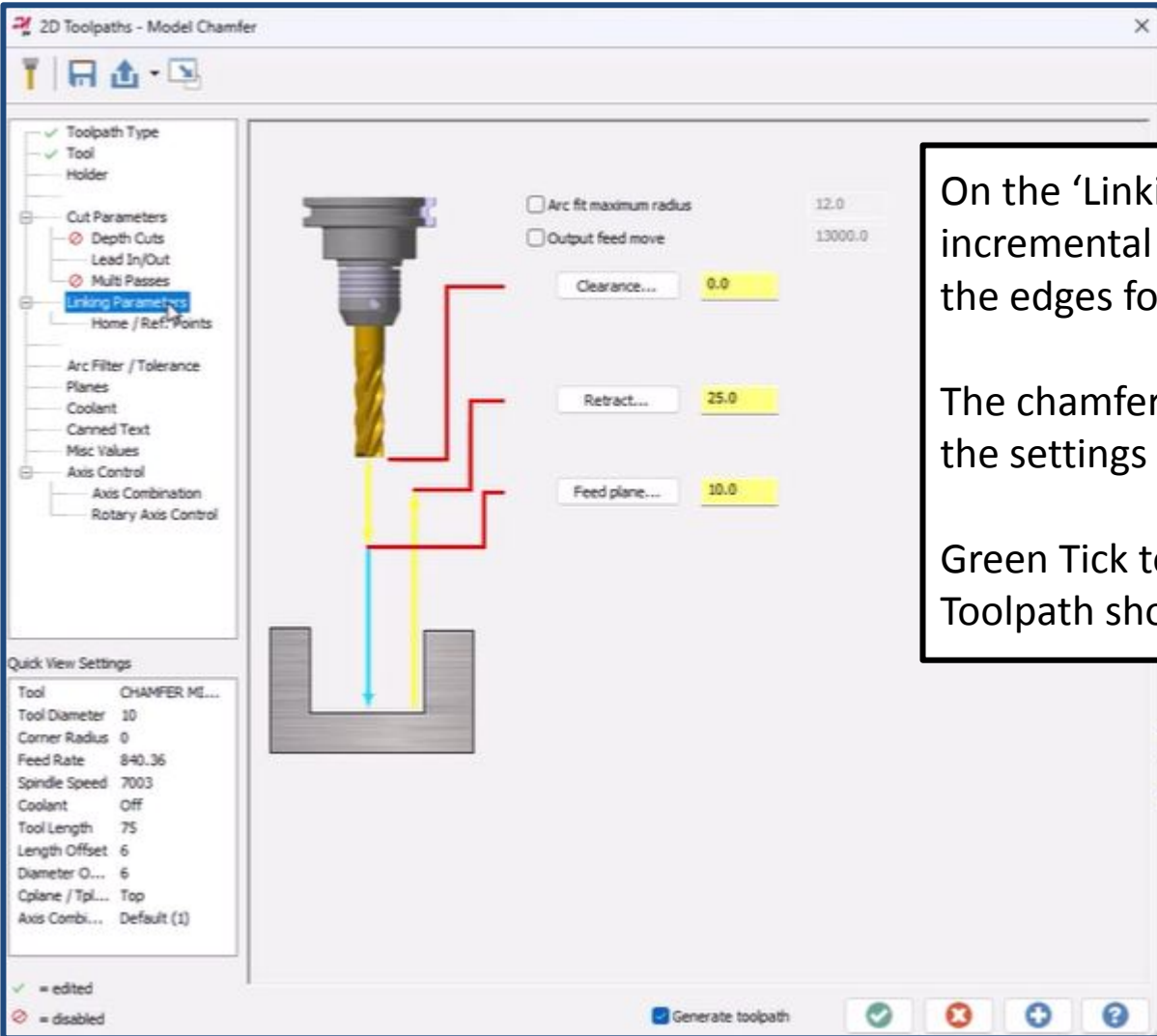
On the 'Tool' page use the tool library and filter options to select a 10mm 90 degree chamfer tool.

Update the speeds and feeds as necessary



Now select the 'Cut Parameters' page

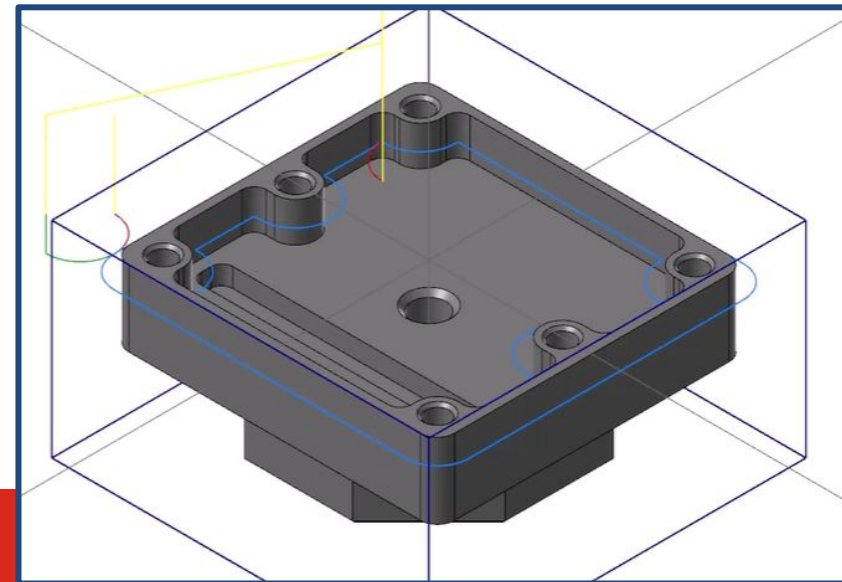
- Set 'Chamfer Width' to 0.3mm
- Set 'Top Offset' to 0.5mm

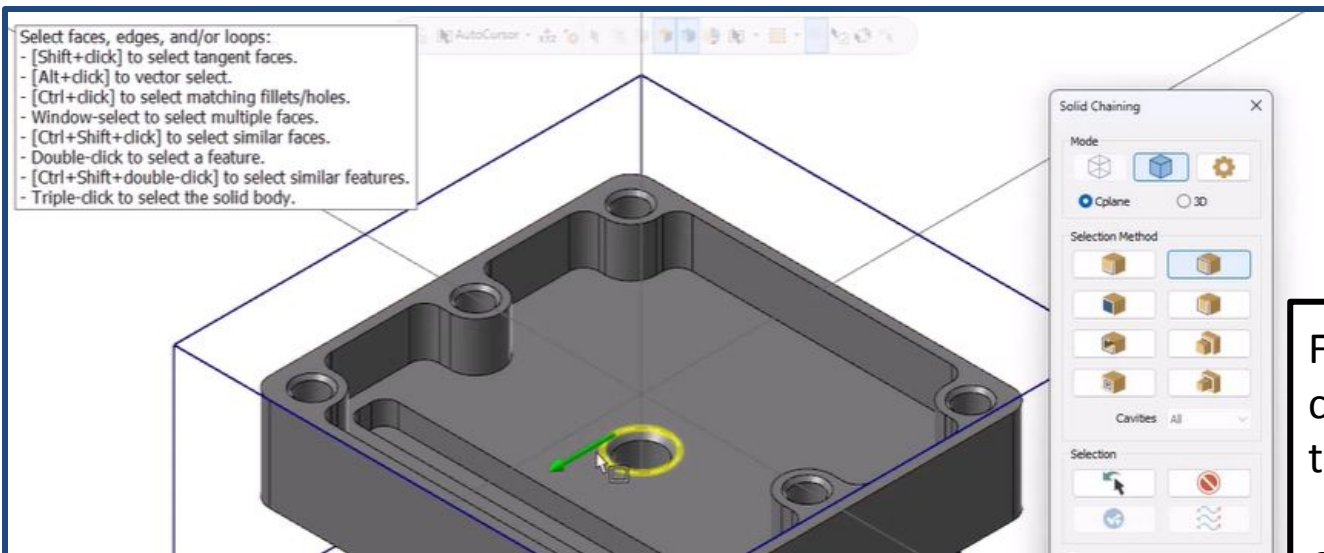


On the 'Linking Parameters' page all the figures are incremental from the chained geometry used to define the edges for chamfering

The chamfer width and tool depth are controlled with the settings on the previous 'Cut Parameters' page.

Green Tick to accept and generate the toolpath
Toolpath should look similar to the one shown below.

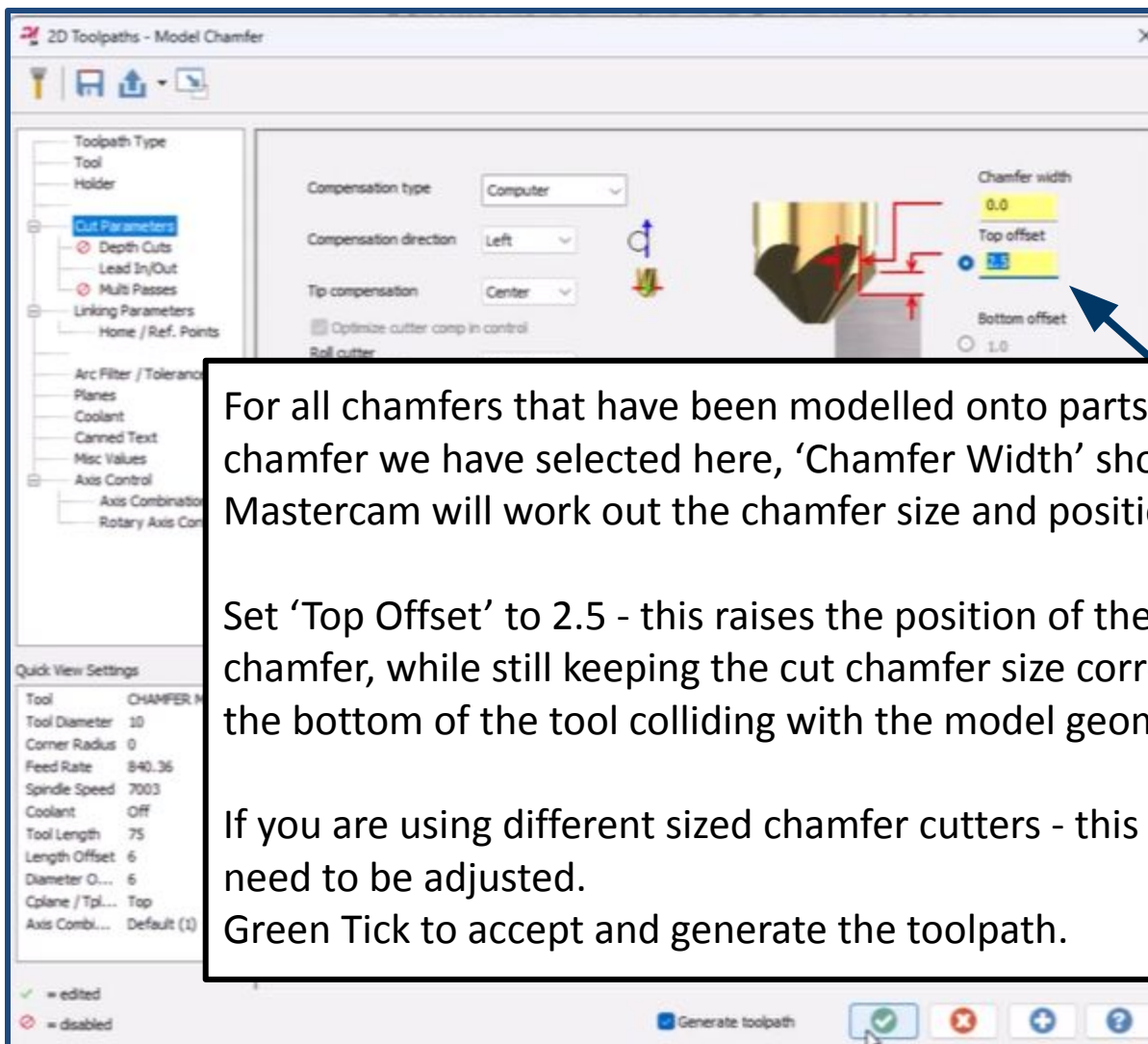




For the chamfer on the centre hole we will need to create a separate 'Model chamfer' toolpath to control the parameters separately

Choose another 'Model Chamfer' toolpath from the '2D Mill Toolpaths' selection.

Chain the chamfer edge shown, using the same process as before.

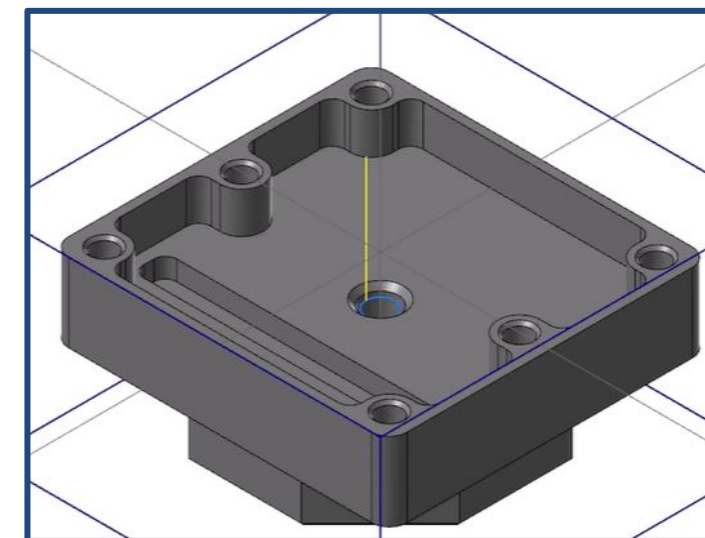


For all chamfers that have been modelled onto parts, such as the chamfer we have selected here, 'Chamfer Width' should be set to Zero - Mastercam will work out the chamfer size and position from the model.

Set 'Top Offset' to 2.5 - this raises the position of the tool on the chamfer, while still keeping the cut chamfer size correct. This will stop the bottom of the tool colliding with the model geometry.

If you are using different sized chamfer cutters - this parameter may need to be adjusted.

Green Tick to accept and generate the toolpath.



End of Session - Save Your Work