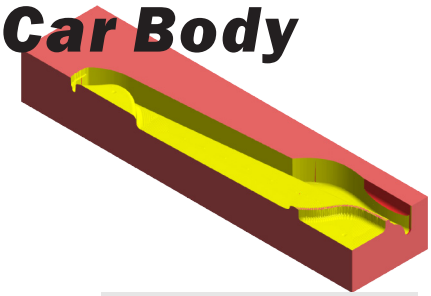
CO2 Rail Car
HST Toolpaths for Rail Car Body
Cut Body (HST Hybrid)



A. Machine Type and Stock Setup.

Step 1. If necessary, open your file from Chapter 20.

Step 2. If necessary, display Operations Manager. Use **Alt-O**.

Step 3. If Machine Group is **not** displayed in the Operations Manager, **Fig. 1**, click MACHINE TYPE Menu > Mill > Default.

Step 4. Expand **Properties** (click the +) in the Toolpaths Manager, **Fig. 1**.

Step 5. Click **Stock Setup** in the Toolpaths Manager.

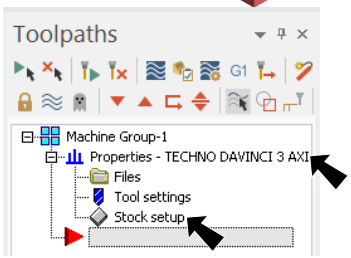


Fig. 1

Step 6. Confirm Stock View is **Top**, **Fig. 2**.

Step 7. Click the **left front top corner of the stock** to move the origin. After you click corner the arrow will point to corner.

Step 8. Key-in X, Y and Z stock dimension into the fields
- **X 305**
- **Y 70**
- **Z 42**

Step 9. Key-in Stock Origin coordinates:
- **X 0**
- **Y -36**
- **Z 21**

Step 10. Click OK in Machine Group Properties. The Stock is displayed as red wireframe, **Fig. 3**.

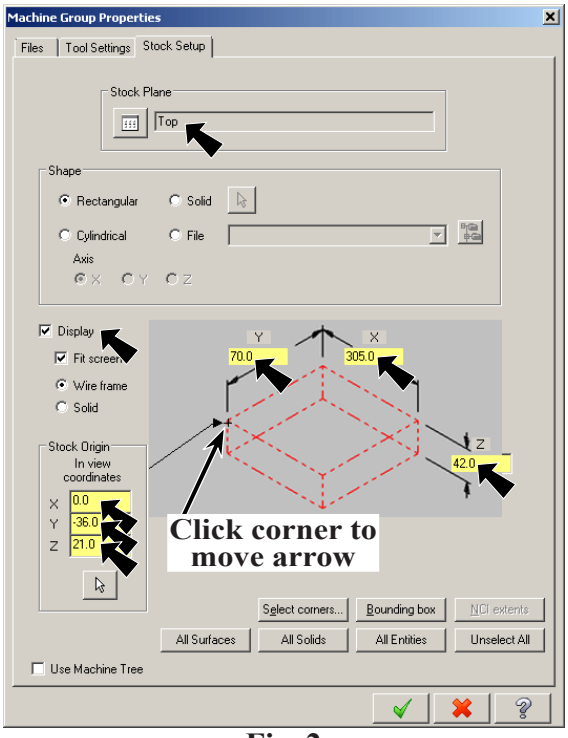


Fig. 2

B. Confirm WCS LEFT CUT.

Step 1. In bottom left corner of display, confirm Tool Plane (T/Cplane) is **LEFT CUT**, **Fig. 3**.

Step 2. Confirm **Left Cut Origin**. Use **F9** to toggle axes.

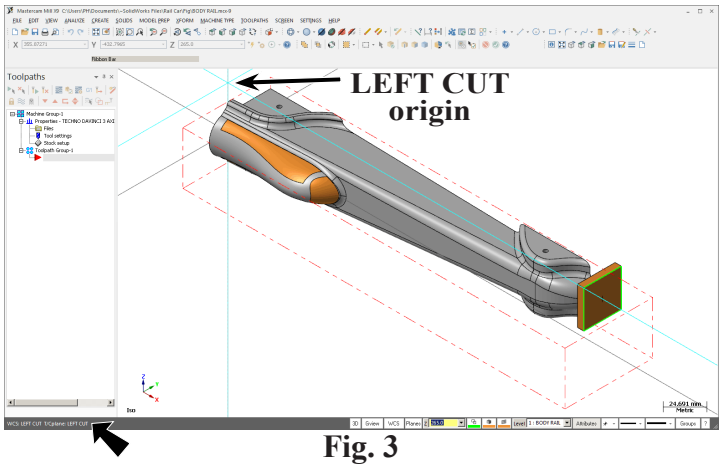


Fig. 3

C. Left Cut Finish Hybrid Toolpath.

Step 1. Click TOOLPATHS Menu > Surface High Speed > **Hybrid**.

Step 2. Click OK in NC name dialog, **Fig. 4**.

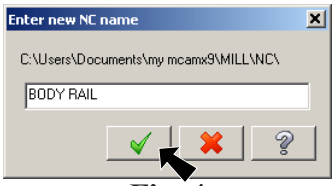


Fig. 4

Step 3. Click the **solid car body** to select as Drive Surfaces, **Fig. 5**. The solid will highlight when selected.

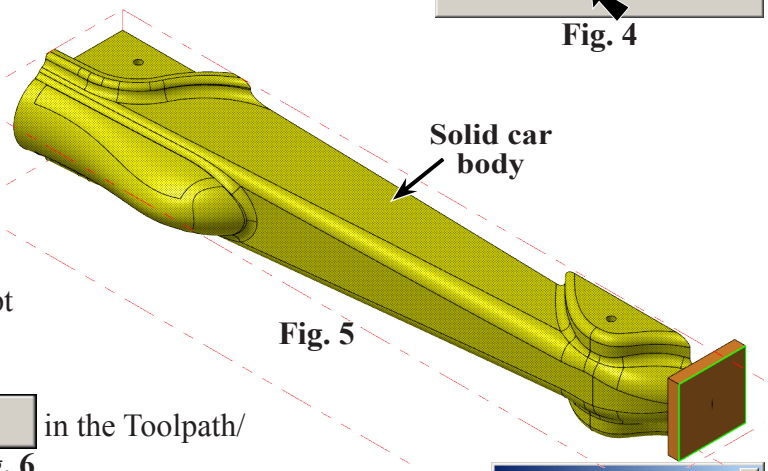


Fig. 5

Step 4. Click **End Selection** (**ENTER**) in ribbon bar to accept solid as drive surfaces.

Step 5. Click **Check Select** button in the Toolpath/surface selection dialog box, **Fig. 6**.

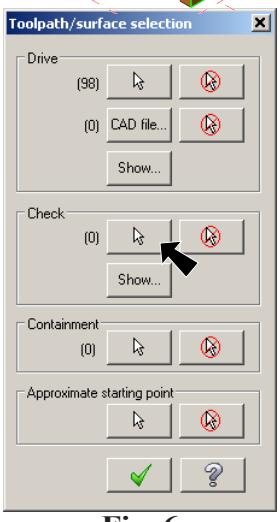


Fig. 6

Step 6. Click the **solid check body**, **Fig. 7**.

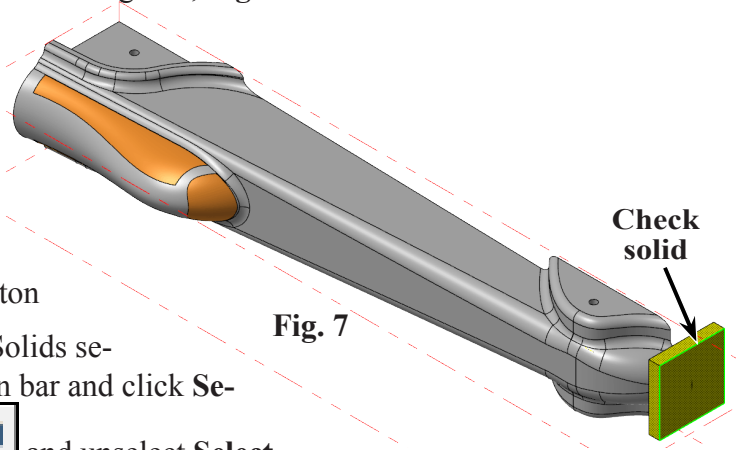


Fig. 7

Step 7. Click **Activate solid selection** button in the Solids selection ribbon bar and click **Select face** and unselect **Select body**, **Fig. 8**.


Fig. 8

Step 8. Click VIEW Menu > Orient > **Flip X for Y**.

Step 9. Click the **flat face on the underside of Body and flat face at rear**, **Fig. 9**.

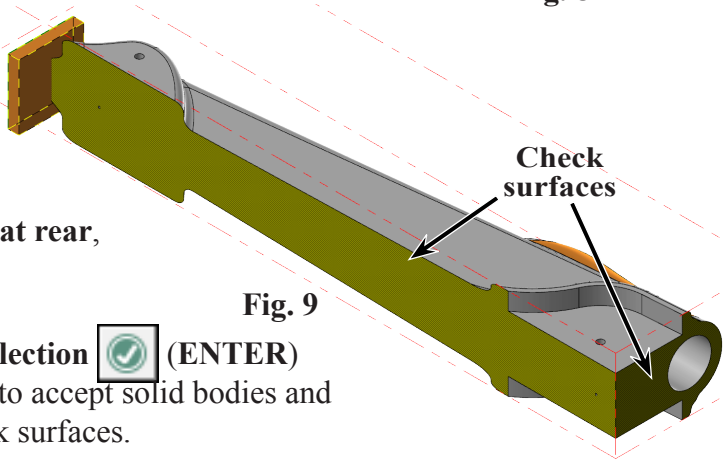


Fig. 9

Step 10. Click **End Selection** (**ENTER**) in ribbon bar to accept solid bodies and faces as check surfaces.

Step 11. Click OK in the Toolpath/surface selection dialog box, **Fig. 10**.

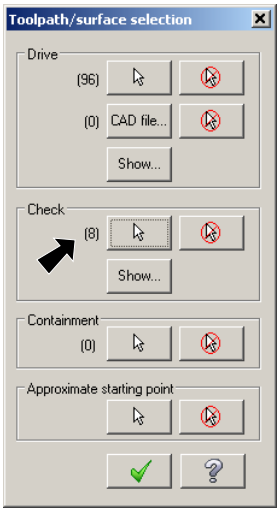


Fig. 10

Step 12. Select **Toolpath Type** from the tree control and confirm: **Hybrid toolpath** **Drive and Check** surfaces **Fig. 11**.

Step 13. Select **Tool** from tree control and:

Click **Select library tool** **Fig. 12**.

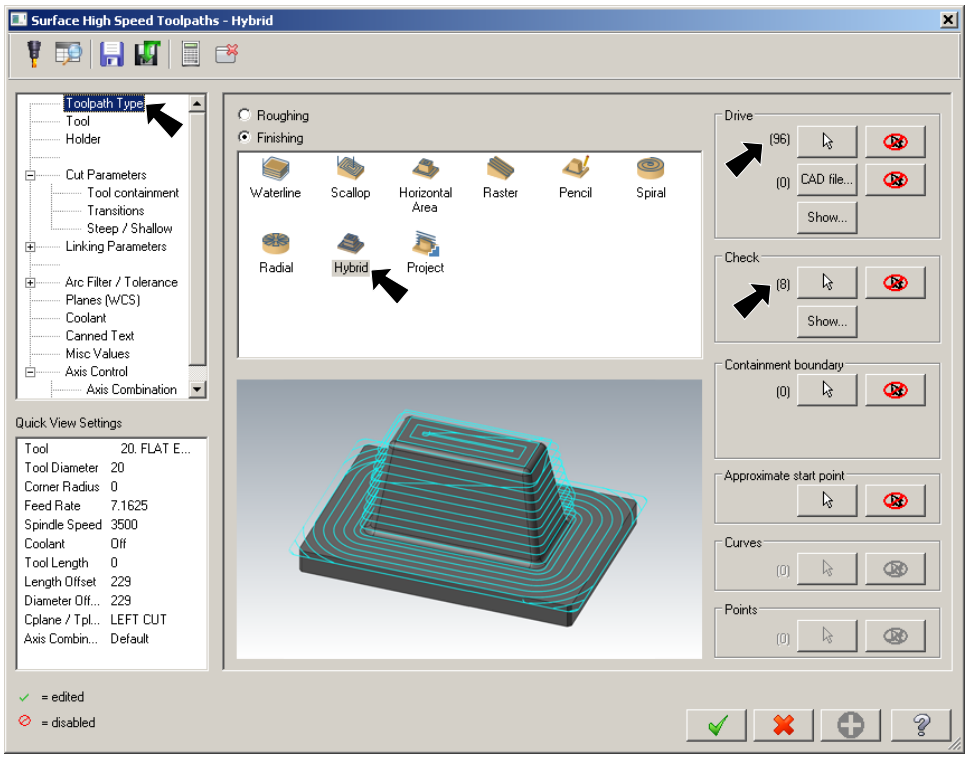


Fig. 11

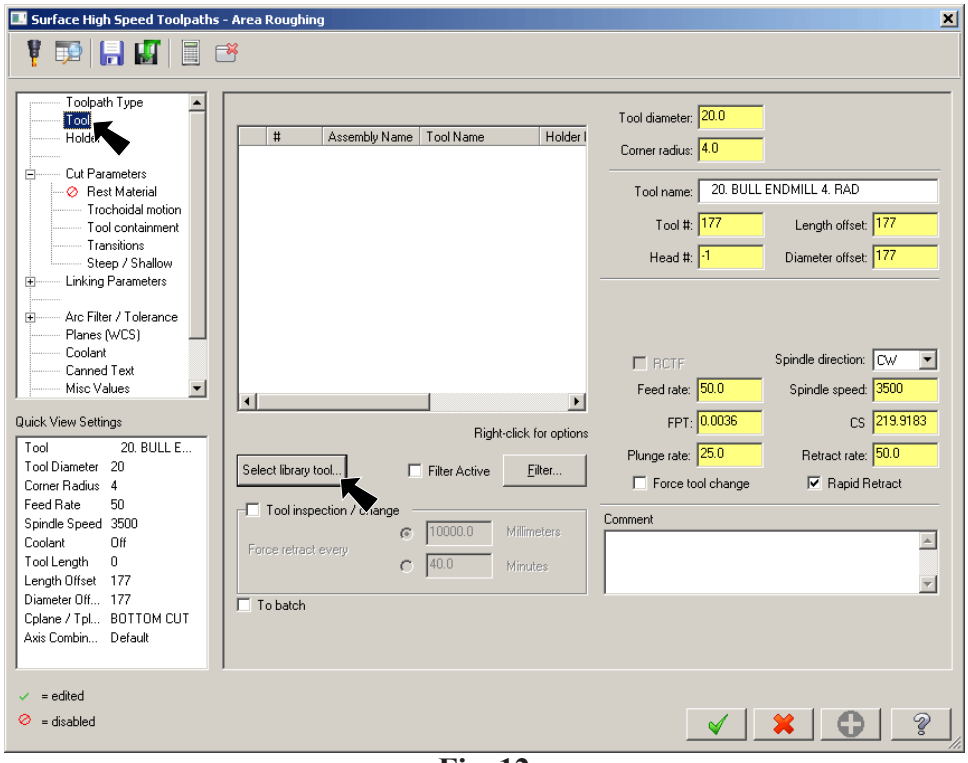


Fig. 12

Step 14. Click **Filter** button, **Fig. 13**.

Step 15. Click **None** button **under Tool Types**, **Fig. 14**.

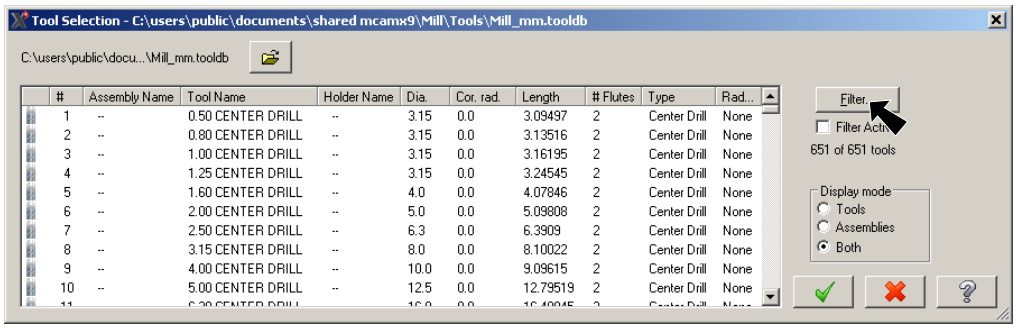


Fig. 13

Step 16. Click **Endmill2 Sphere** button (second button top row) and click OK, **Fig. 14**.

Step 17. Click **491 BALL END-MILL 6mm** and click OK, **Fig. 15**.

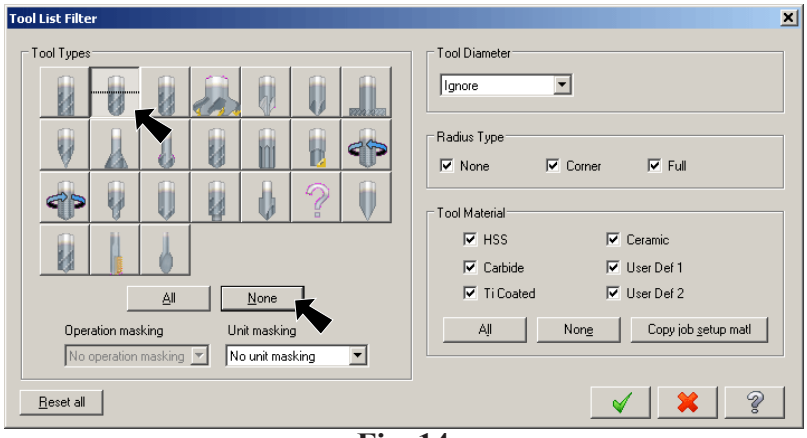


Fig. 14

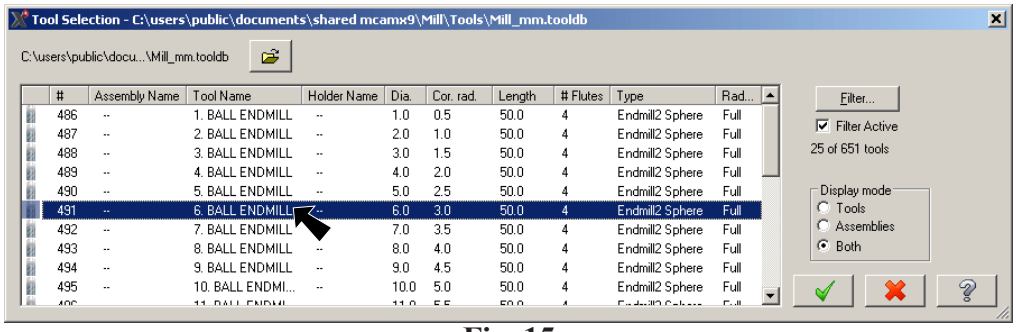


Fig. 15

Step 18. Back in **Tool** page set:

Feed rate 300

Plunge rate 200
Fig. 16.

Step 19. Select **Cut Parameters** from tree control and set:
Open contour direction Zigzag

Z stepdown 3

Limiting angle 34

3D stepover 1.2

Check **Preserve Z passes**

Offset method Upper to lower

Keep tool down within 80%

Stock to leave on walls and floors 0
Fig. 17.

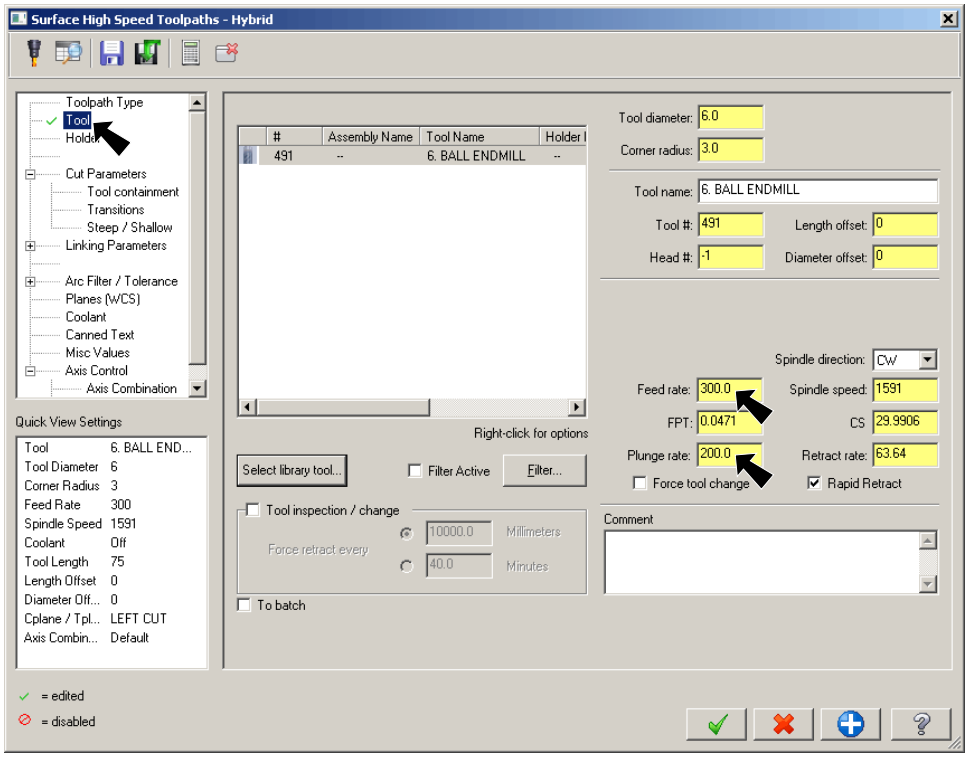


Fig. 16

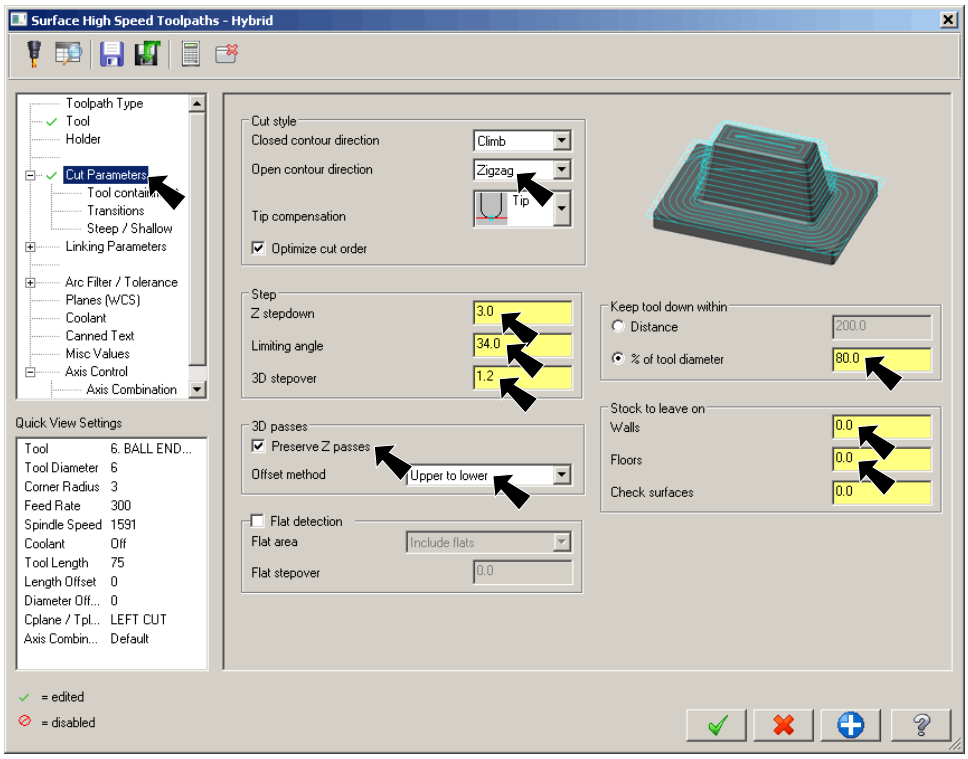


Fig. 17

Step 20. Select **Steep/ Shallow** from tree control and set:

Check **Use Z depths**
Minimum 0
Maximum -37

Click **Apply**

Fig. 18.

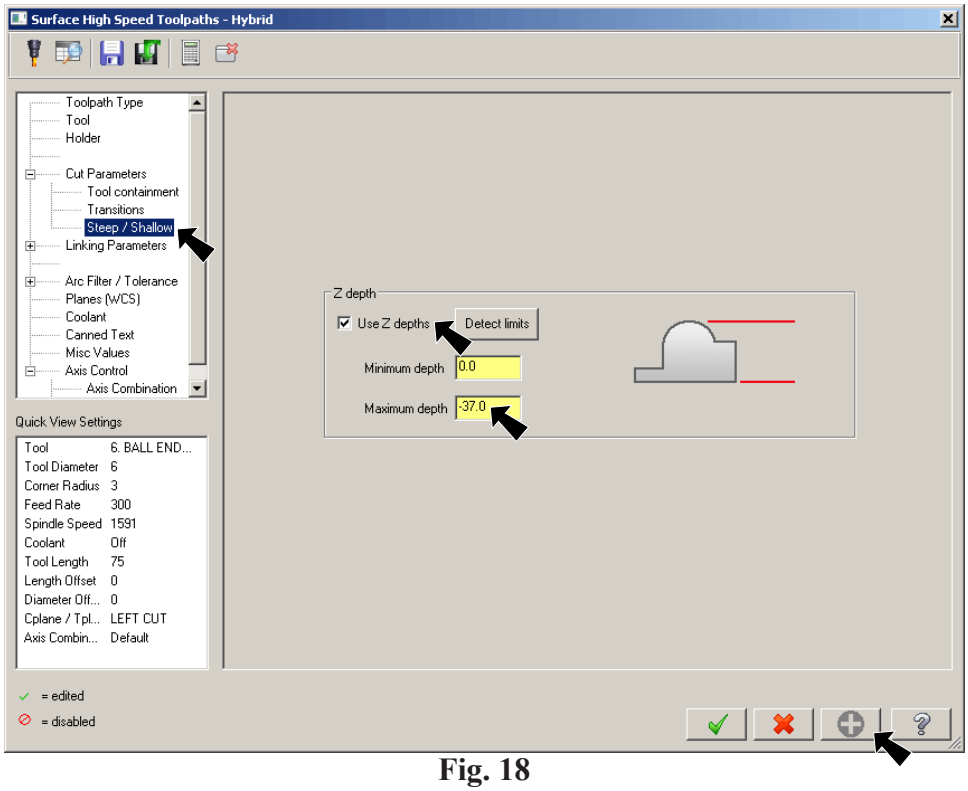


Fig. 18

Step 21. Select **Linking Parameters** from tree control and set:

Clearance plane 2

select **Minimum Vertical Retract**

Part clearance 4

All Leads 0
Fig. 19.

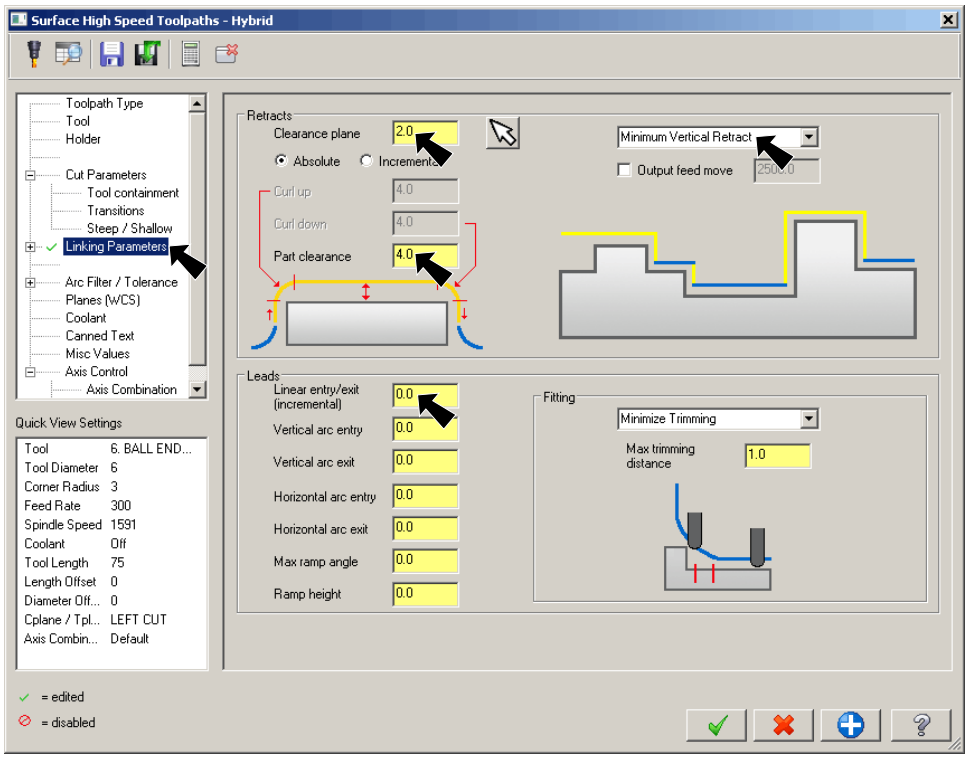


Fig. 19

Step 22. Select **Arc Filter/Tolerance** from tree control and set:

Total tolerance .0625

check **Line/Arc Filtering Settings**

uncheck **Create arcs in XY**

check **One way filtering**

set **Minimum arc radius .0625**

Cut tolerance 10%
Fig. 20.

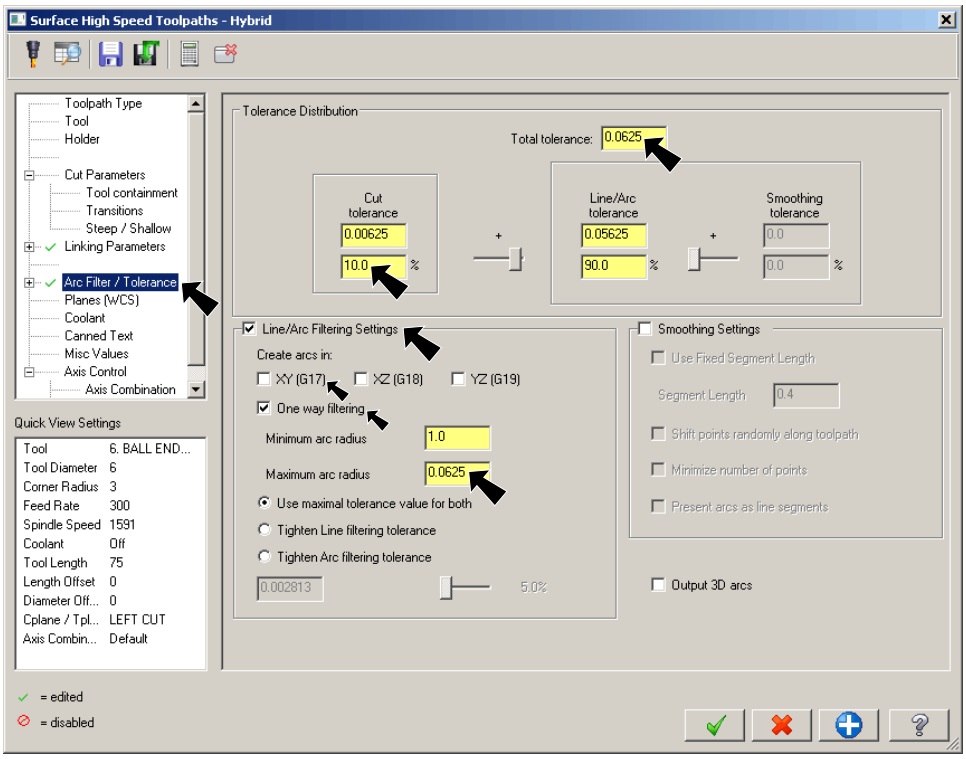
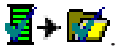


Fig. 20

Step 23. Click OK  in Hybrid dialog box and allow Mastercam to calculate toolpath  .

Step 24. Save  **(Alt-F S)**.

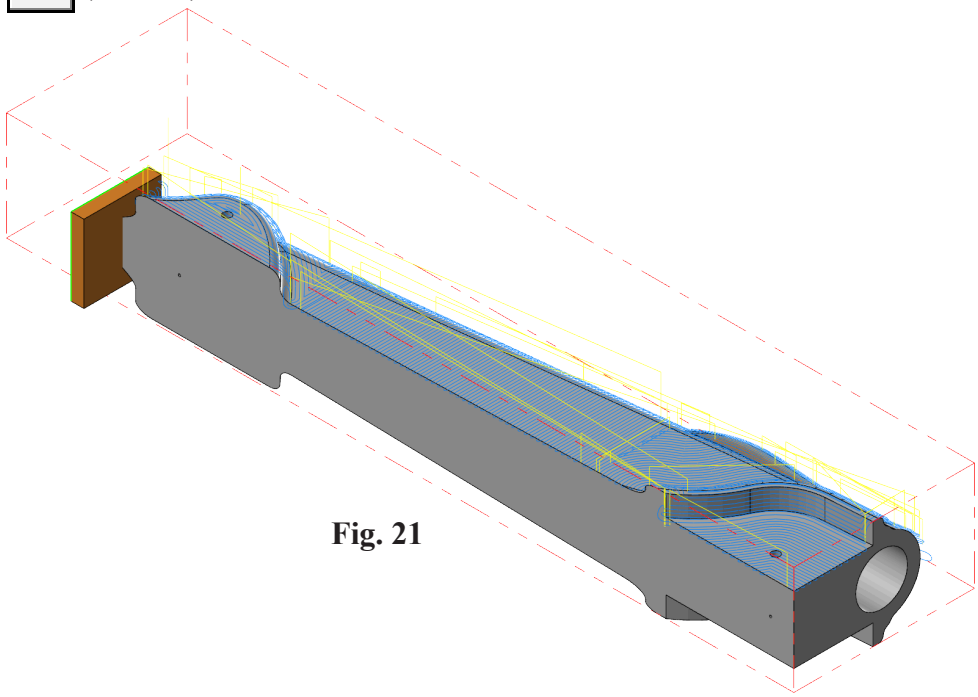


Fig. 21

D. Verify Left Cut.

Step 1. In the Toolpaths Manager, click the toolpath to select and click **Verify**, **Fig. 22**.

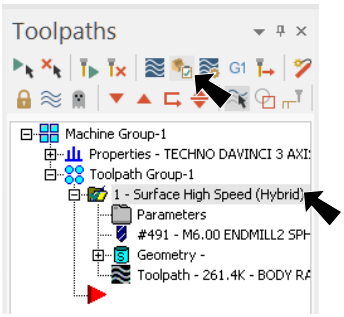


Fig. 22

Step 2. Click **Play** **(R)** in VCR bar, **Fig. 23**.

Fig. 23

Step 3. Note **Total Time** to run program **(38min 57.19s)**, **Fig. 24**.

Step 4. Switch back to Mastercam (**Alt-Tab**).

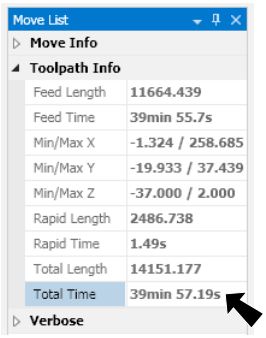


Fig. 24

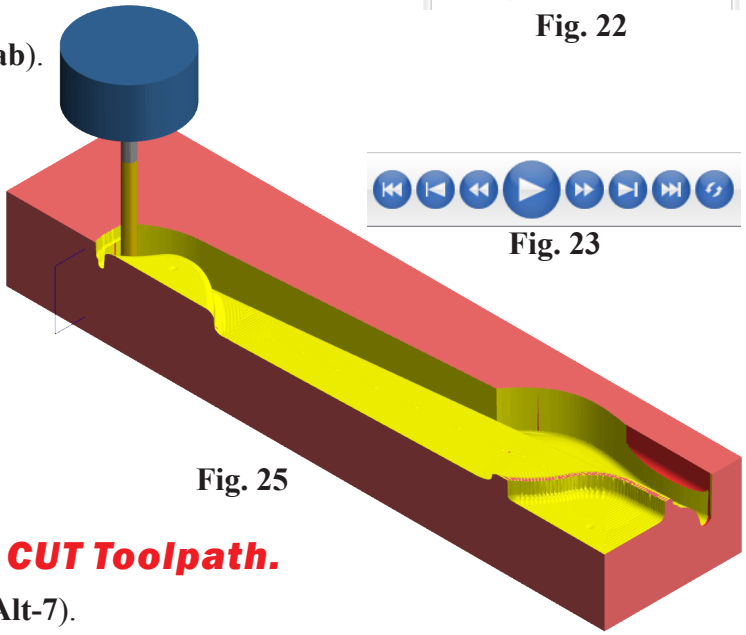


Fig. 25

E. Mirror LEFT CUT to RIGHT CUT Toolpath.

Step 1. Change to Isometric View (**Alt-7**).

Step 2. Click TOOLPATHS Menu > Transform.

Step 3. Under Type, select **Mirror**, **Fig. 26**.

Step 4. Under Source, select **NCI**, **Fig. 26**.

Step 5. Under Source operations, select **Surface High Speed (Hybrid)**, **Fig. 26**.

Step 6. Click the **Mirror tab** at the top of the dialog box, **Fig. 26**.

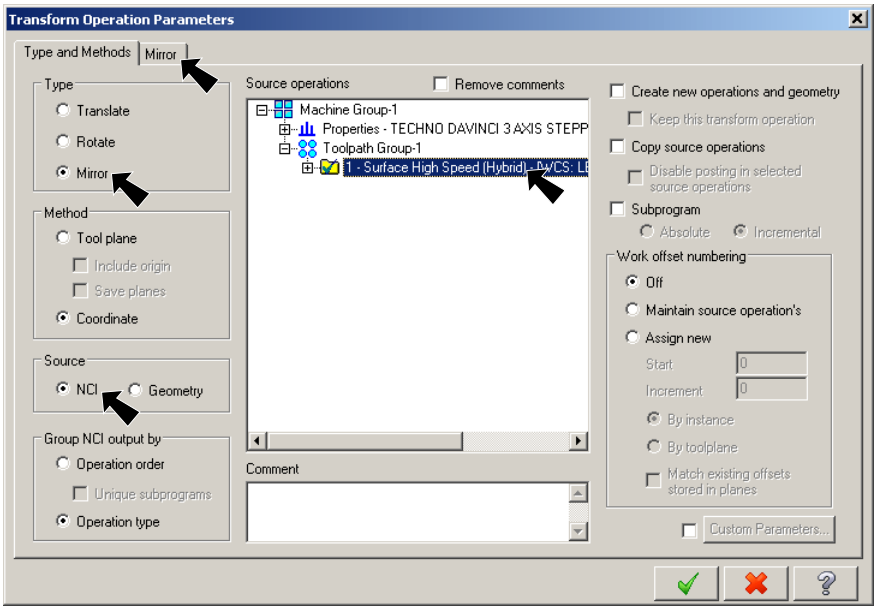


Fig. 26

Step 7. Under Method, select **Mirror about X axis** , **Fig. 27**.

Step 8. Click OK in Transform Operation Params dialog box.

Step 9. Allow Mastercam to calculate toolpath.

Step 10. Save (**Alt-F S**).

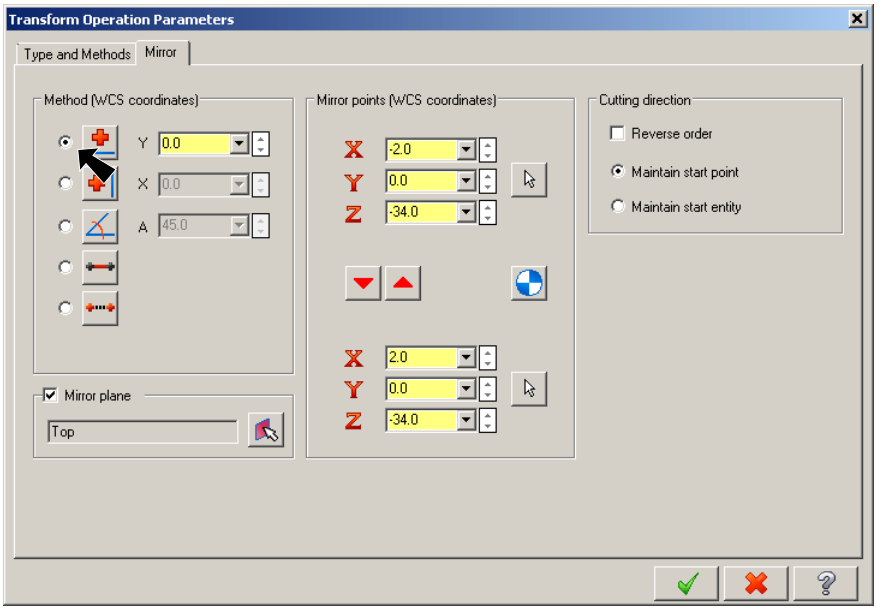


Fig. 27

F. Verify Right Cut.

Step 1. In Toolpaths Manager, click the **Transform/ Mirror** toolpath, **Fig. 28**.

Step 2. Click **Verify** in Toolpaths Manager, **Fig. 28**.

Step 3. Click **Play** (**R**) in VCR bar.

Step 4. Click **Close** to close Mastercam Simulation.

Step 5. Save (**Alt-F S**).

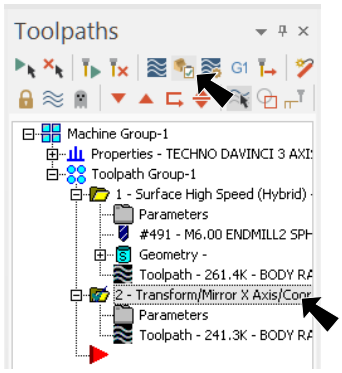


Fig. 28

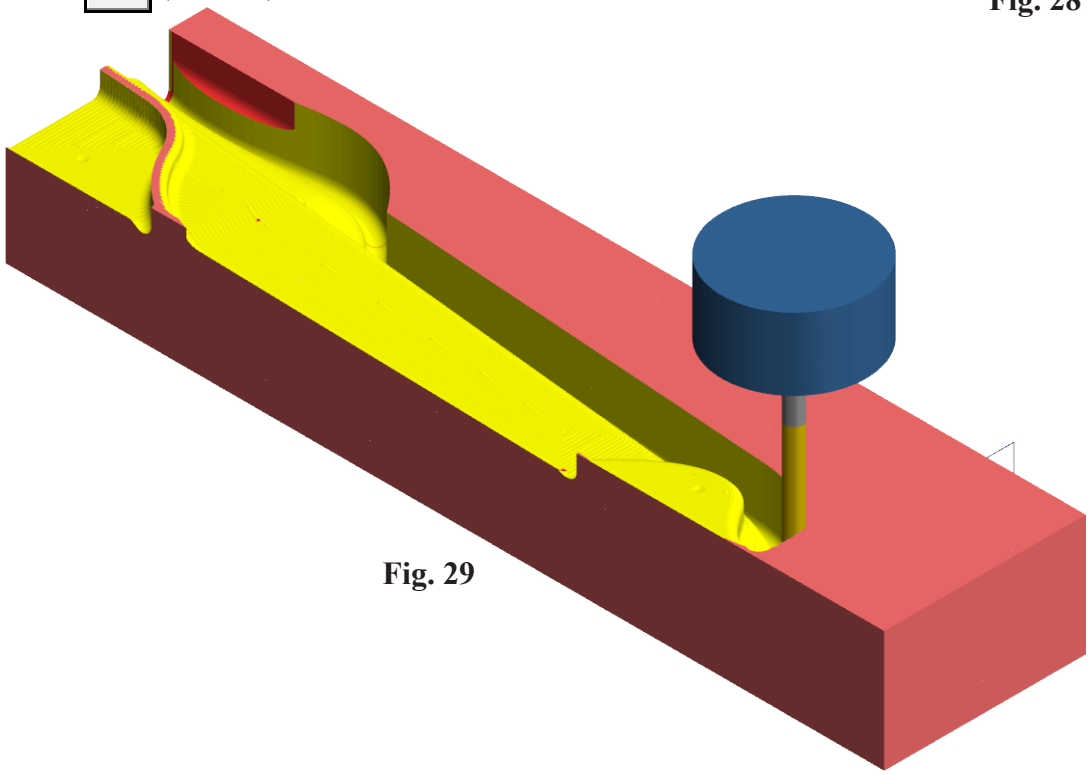


Fig. 29