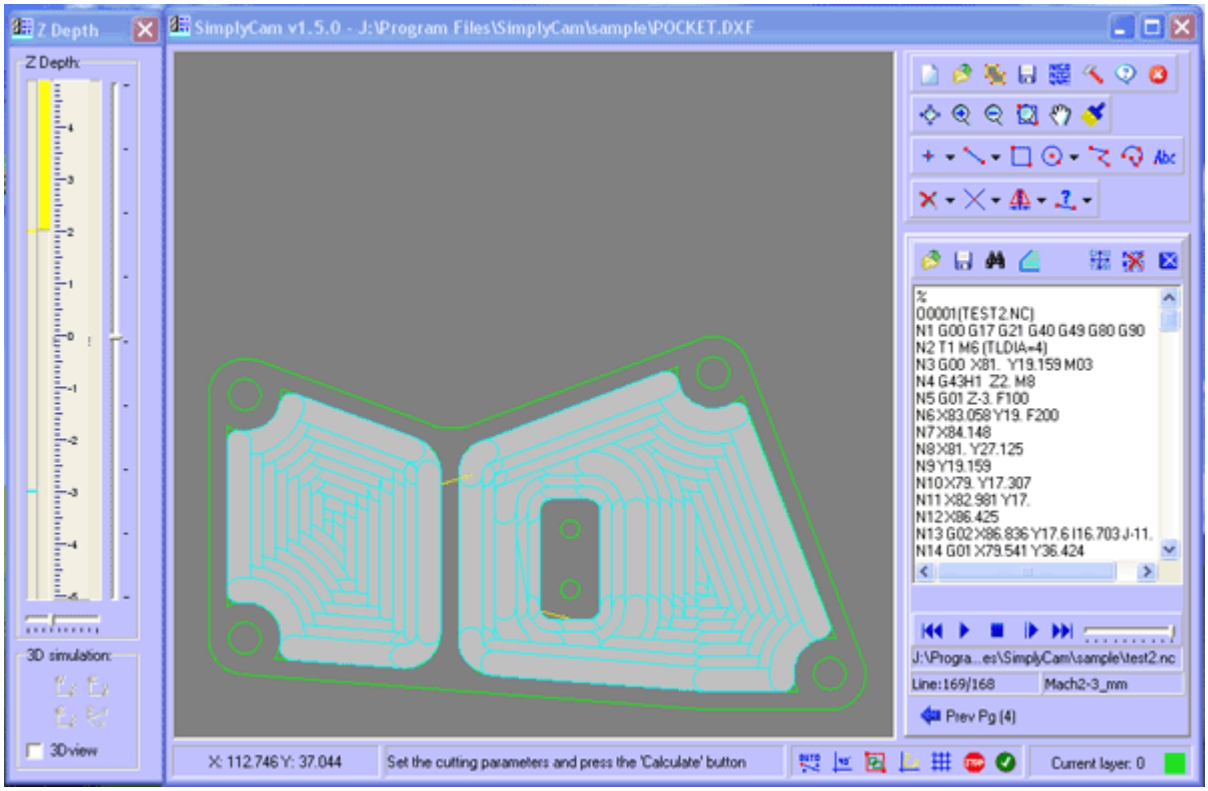
SimplyCam

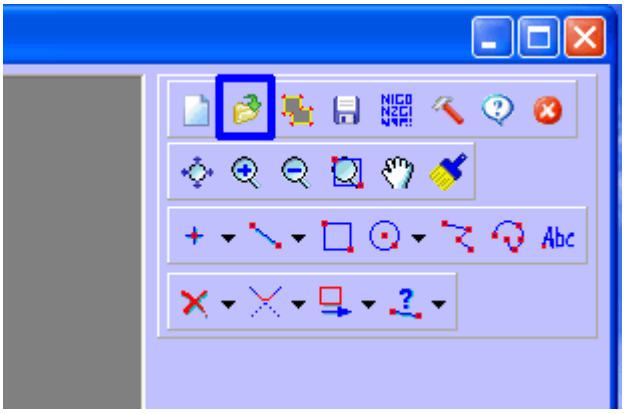
Tutorial 3 - Open Dxf file and create the Pocket toolpath.

In this tutorial you will open a Dxf file and create the toolpath to remove the material contained in a **closed** profile.



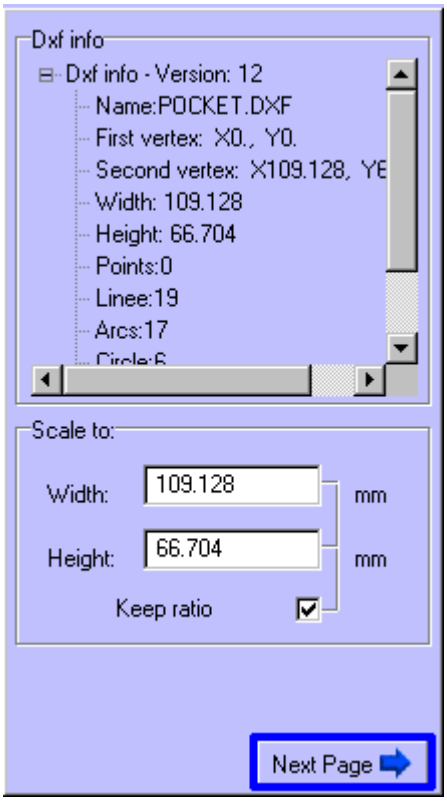
Caution:
CNC machines are potentially dangerous. The post-processor can output code unsuitable for your machine's control. Check the Nc file before sending it to a CNC machine.

1. Open the Dxf file with the "Open" button.

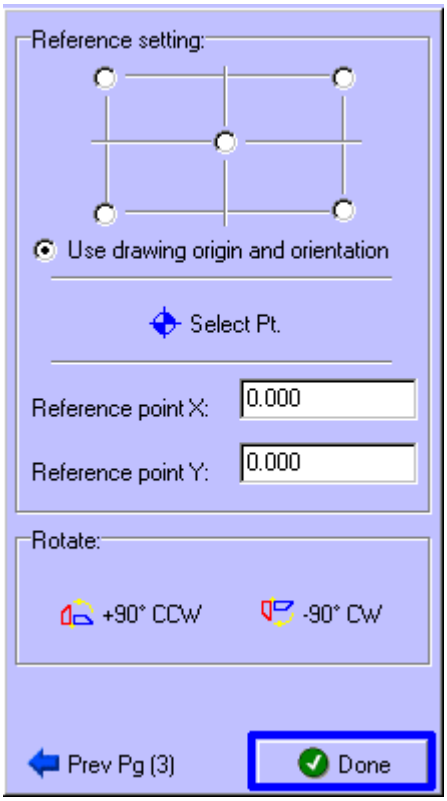


2. Select in "..\SimplyCam\Sample\" folder the **SAMPLE_POCKET.DXF** file.

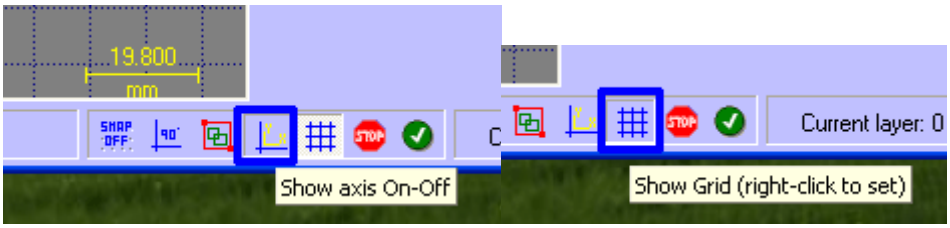
3. The Info panel will appear with the info and dimension of image. Press "Next Page" button.



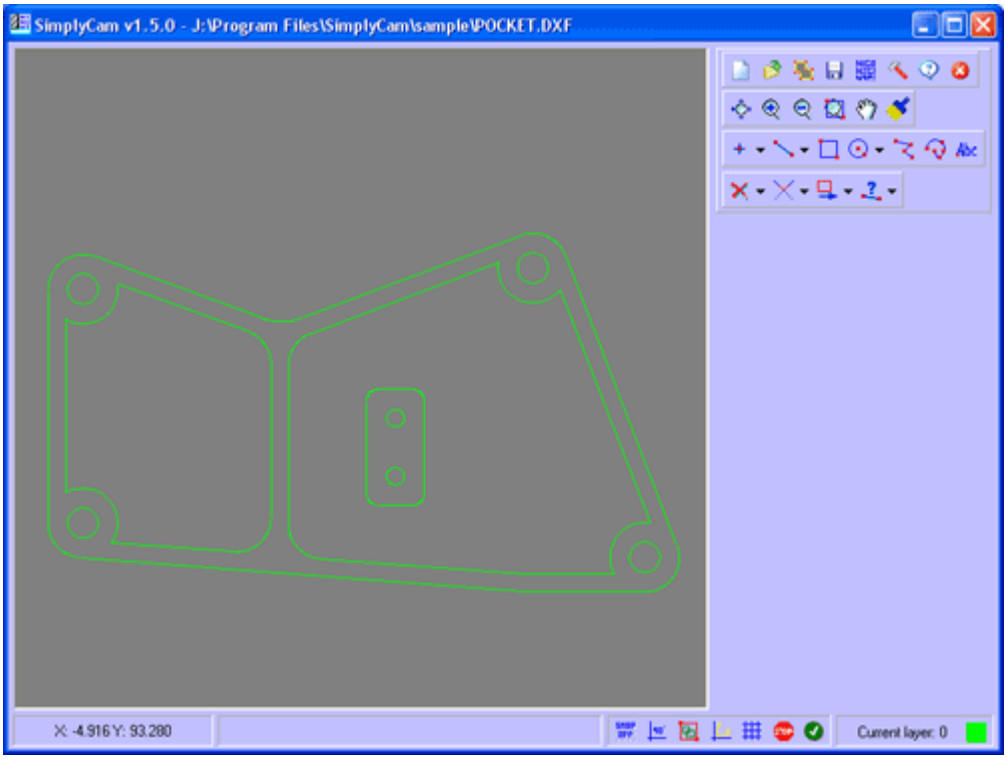
4. Define the reference point of the drawing and press "Done" button.



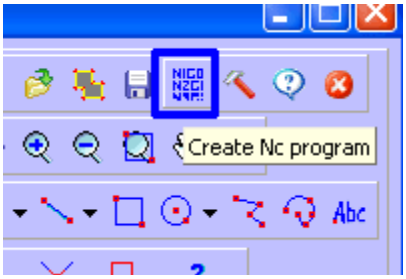
5. Turn in the "Off" state the button "Show grid" and "Show Axis".



6. The drawing is displayed in graphic area without the grid, the axis direction, the origin and the scale info.

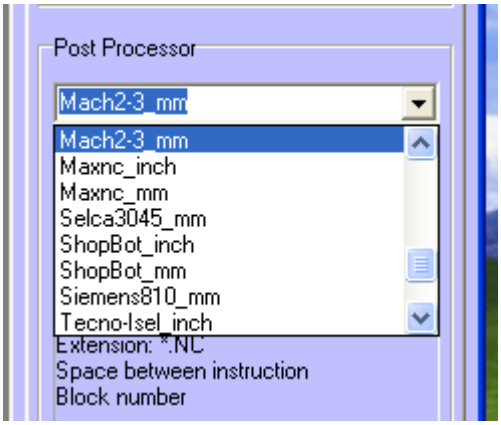


7. Press "Nc Program" button to go in toolpath section.

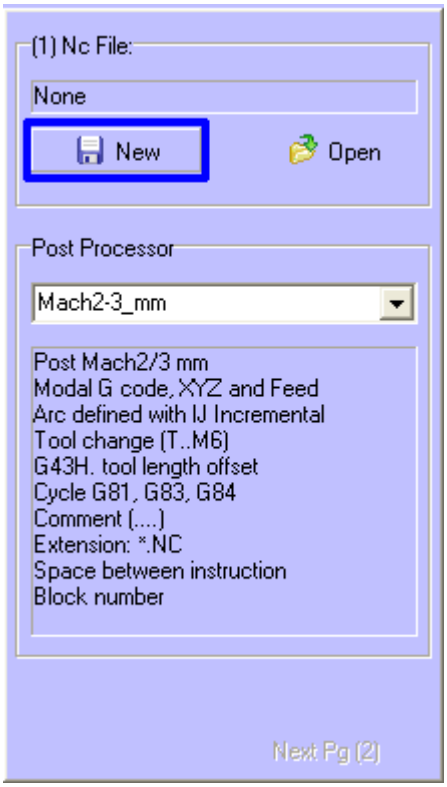


This section contain 5 pages:
Page 1: File/Postprocessor definition
Page 2: Tool/Operation definition
Page 3: Profile/Cutter compensation selection
Page 4: Cutting parameter setting
Page 5: Nc file simulating or editing

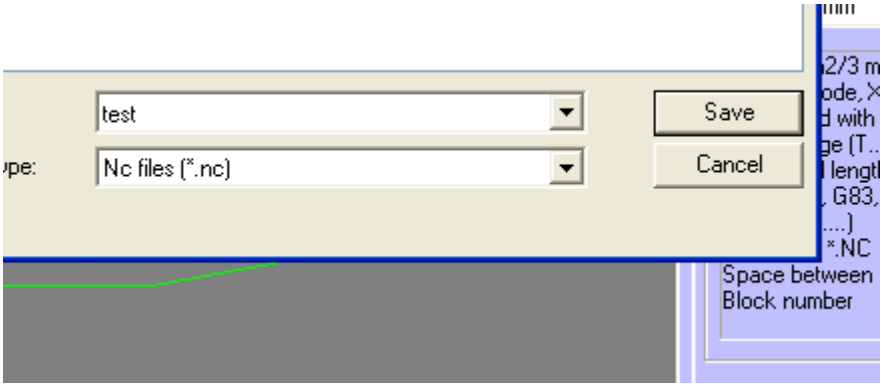
8. Select the postprocessor of your Cnc machine.



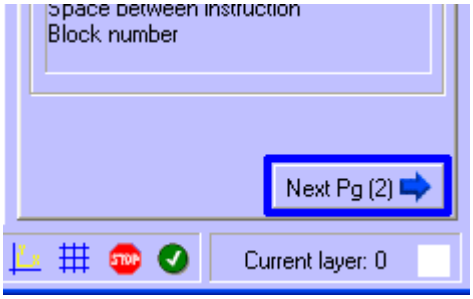
9. Press "New" button to create new toolpath.



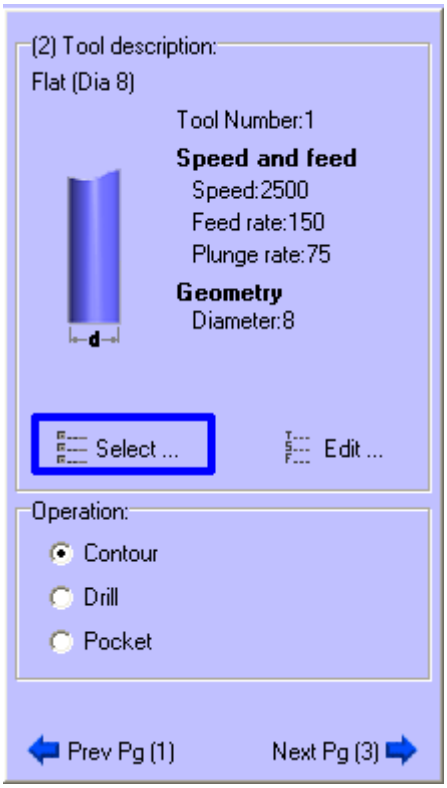
10. Type "test" in the Windows file dialog.



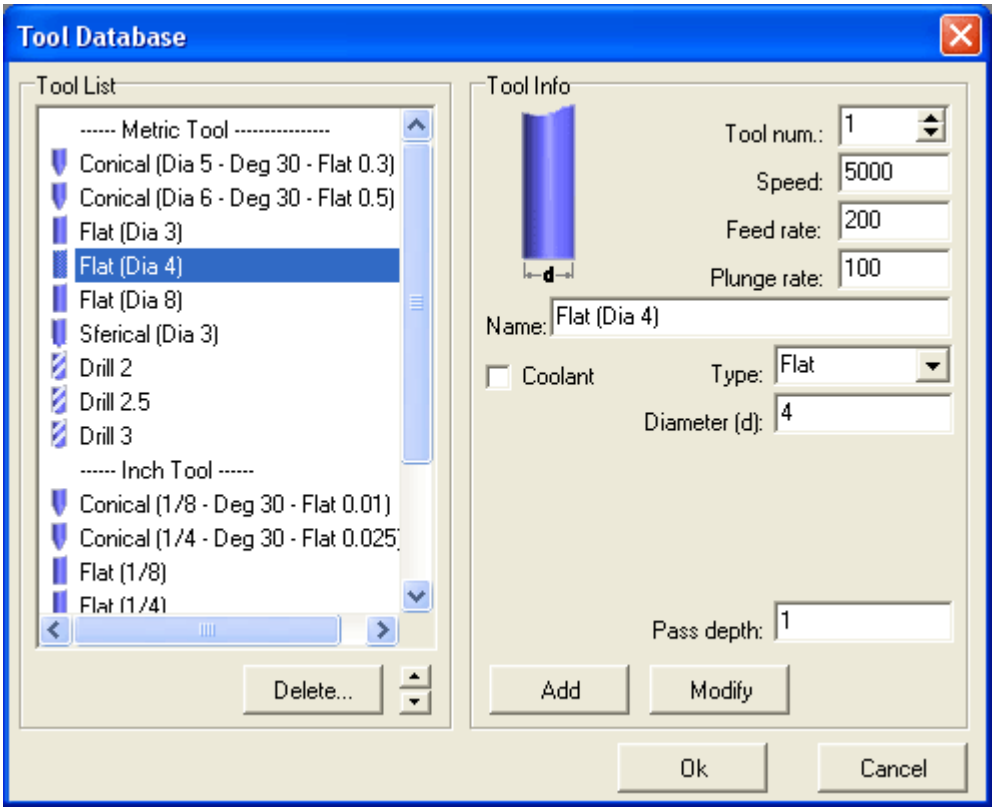
11. Press "Next Pg(2)" button.



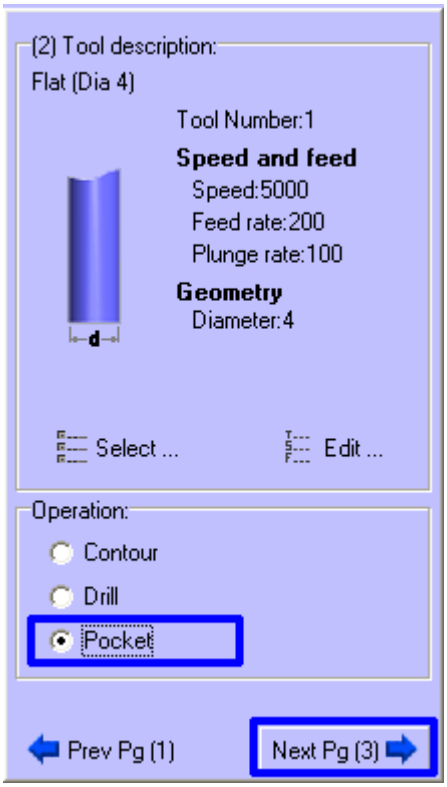
12. In Tool section press "Select" button to retrieve a tool from a tool library or create a new tool.



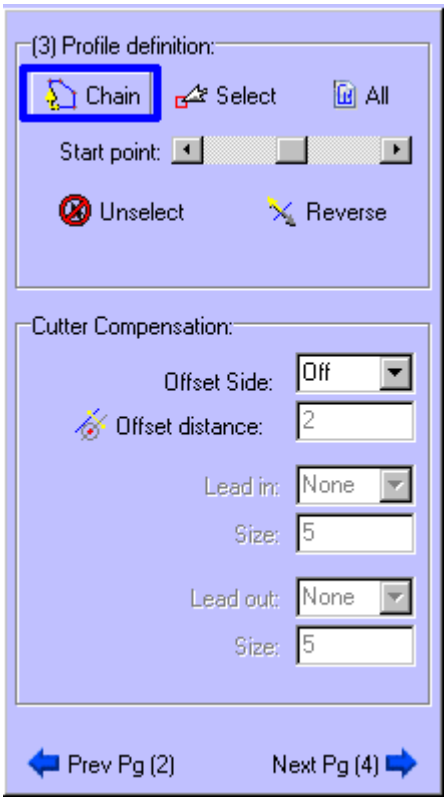
13. Click in the Tool List to select the Flat tool with **diameter 4**. Set feeds and speed of the tool. Press the "OK" button.



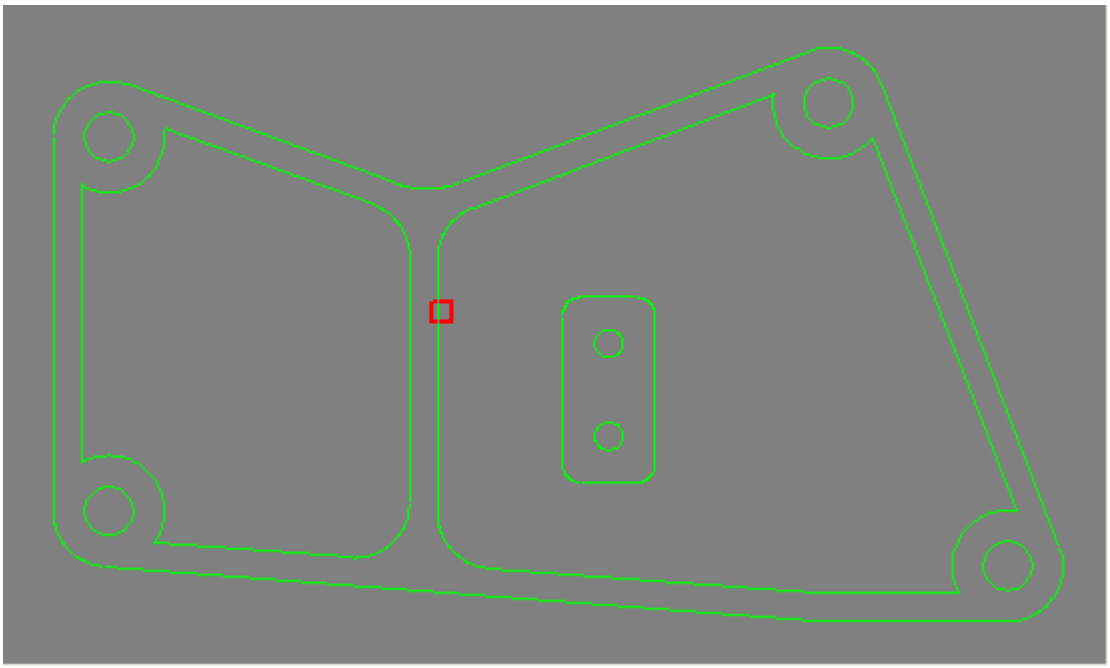
14. In the Operation section, select "Pocket" and press "Next Pg(3)".



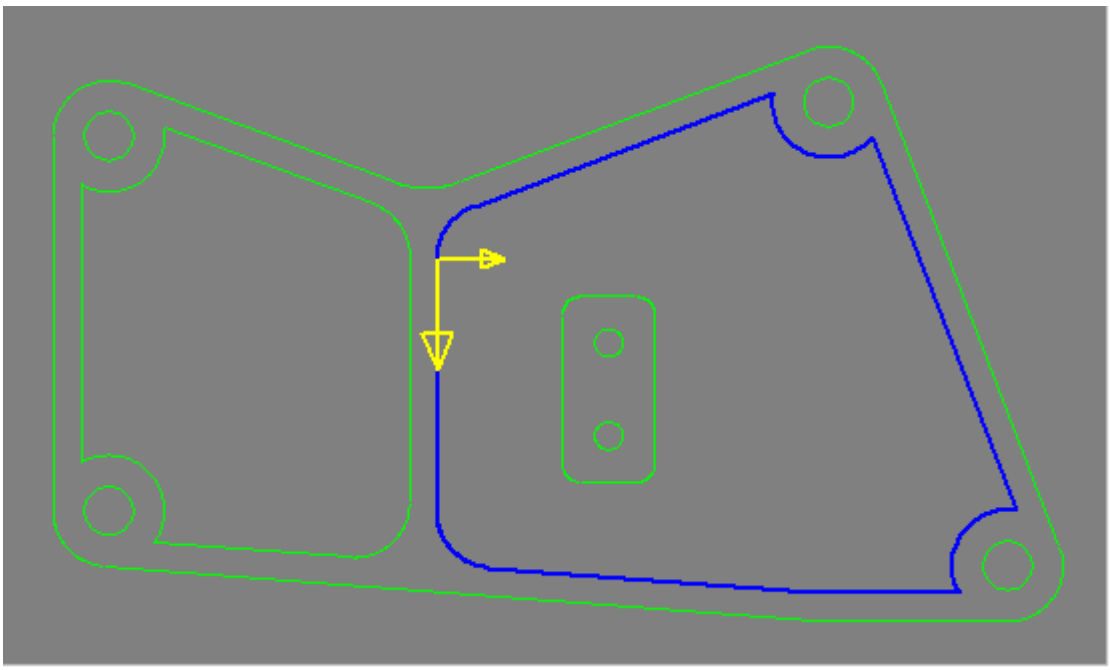
15. The "Profile definition" page appears. Press the "Chain" button.



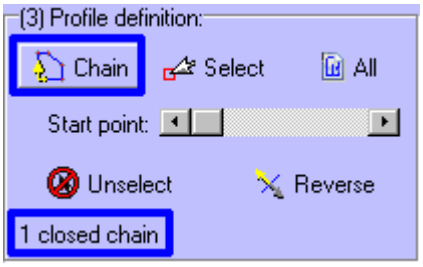
16. Pick the geometry near to start point as indicate by red square.



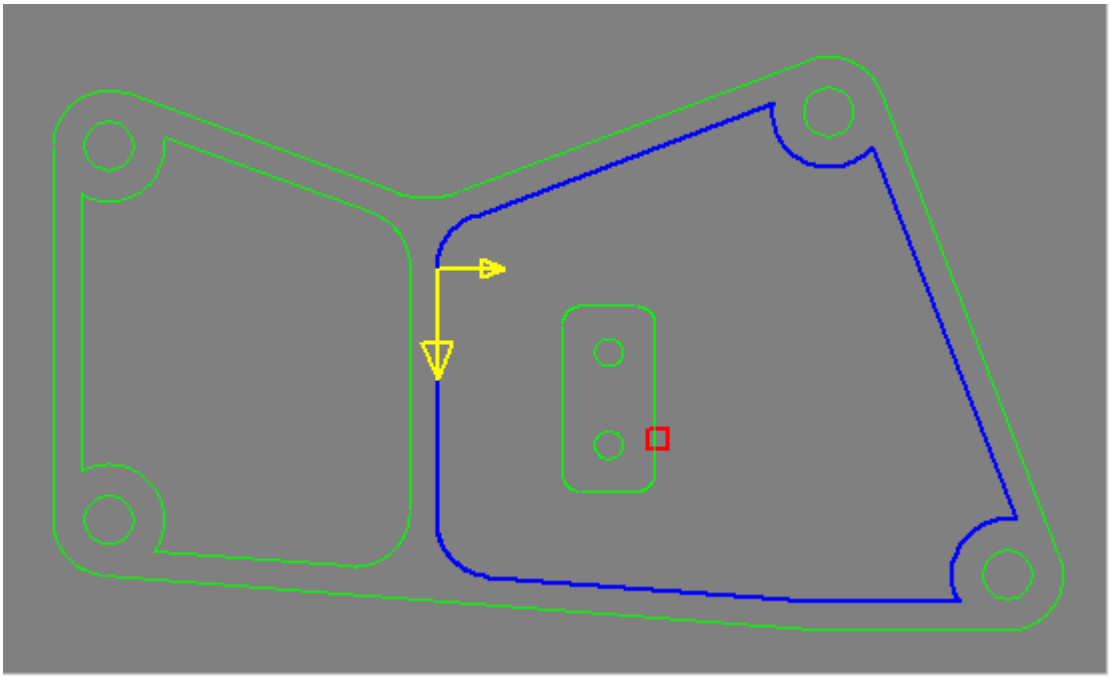
17. Two yellow arrows appear.
    The long arrow indicate the start point of boundary and the direction of toolpath.
    The small arrow indicate the side of toolpath.
    The blue boundary is the chained profile.



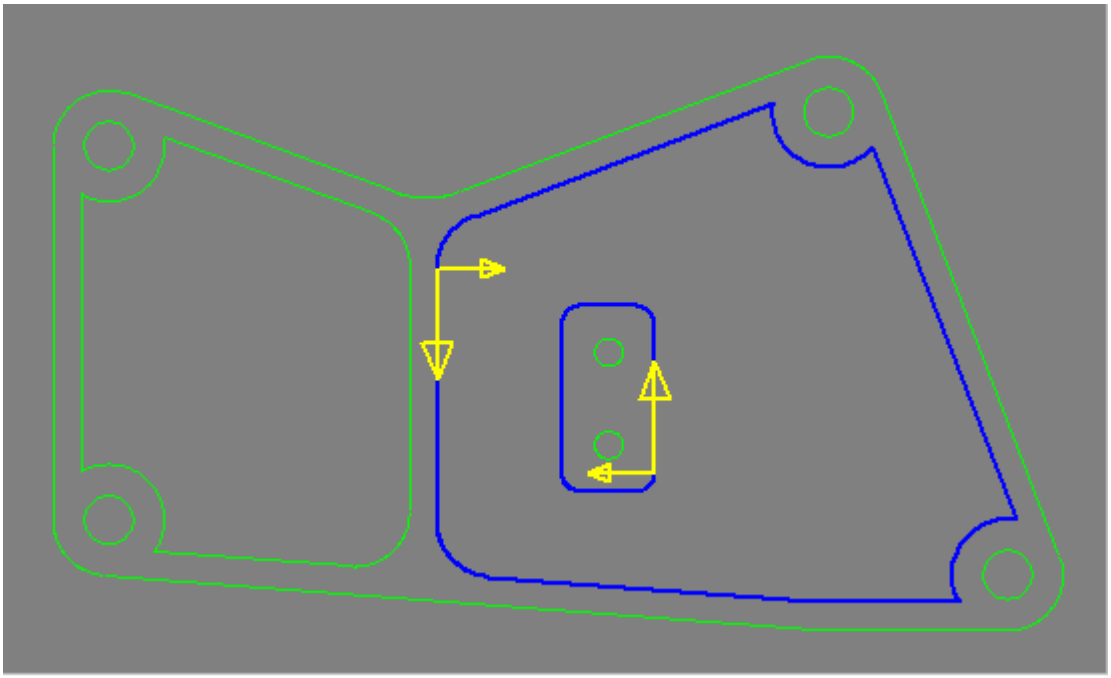
18. Press the "Chain" button again to define another boundary.



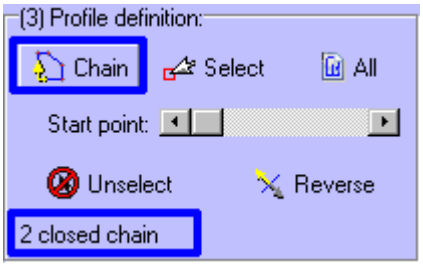
19. Pick the new geometry such as indicate by red square. This is an island.



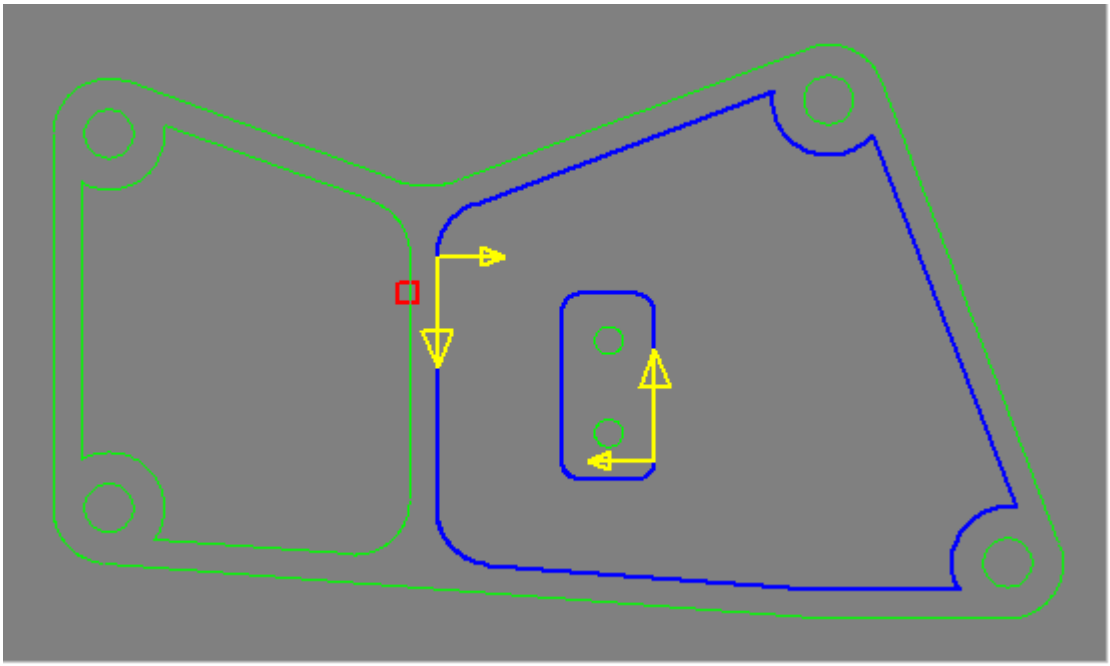
20. Now the island profile is chained.



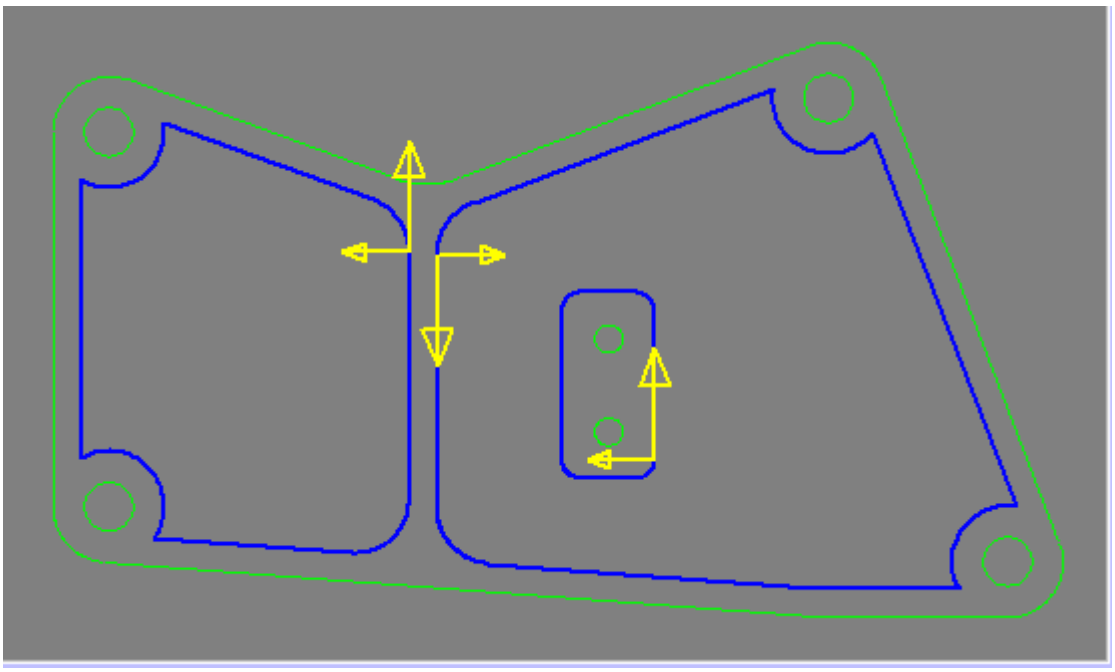
21. Press the "Chain" button again to define another boundary.



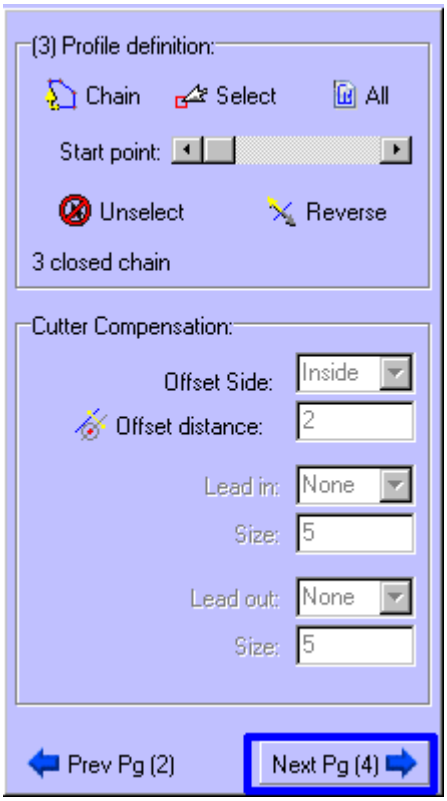
22. Pick the geometry such as indicate by red square. This is another pocket.



23. The boundary of another pocket is chained.

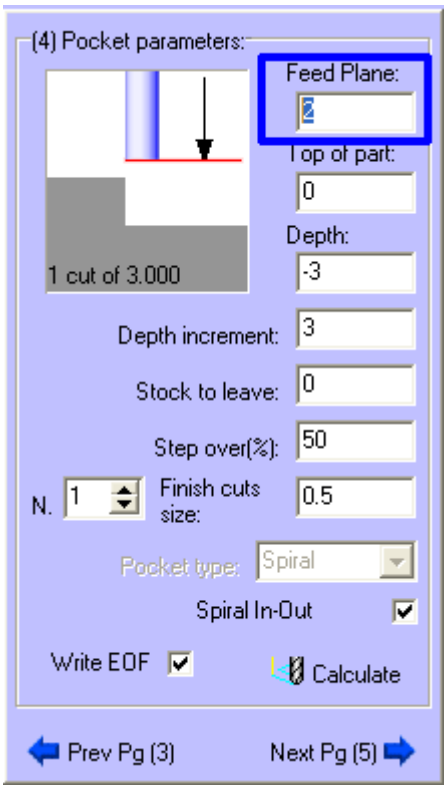


24. Press the "Next Pg(4)" button.

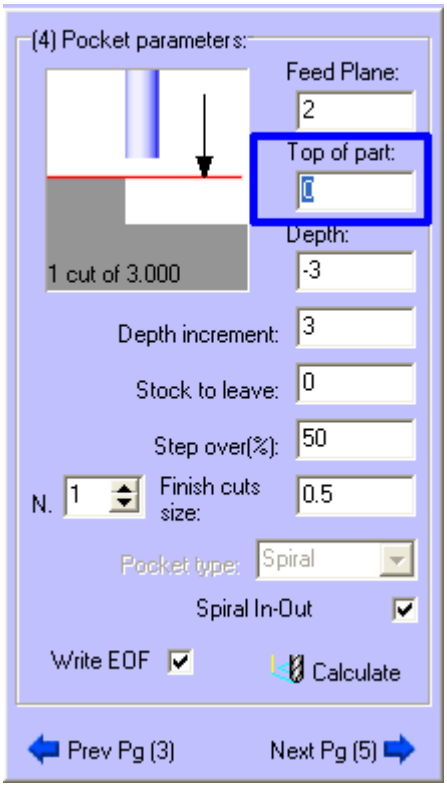


25. The "Pocket parameters" page appears. Set up the following parameters:

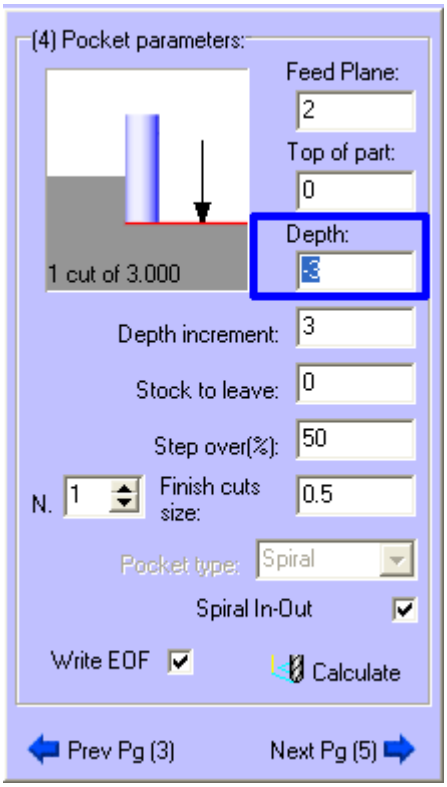
26. **Feed plane:** set the height that the tool rapids to (G0) before changing to the feed rate (G1) to enter in the part (absolute).



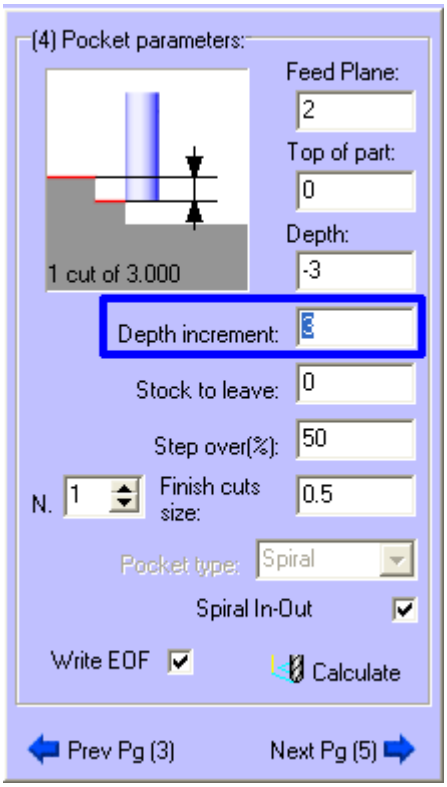
27. **Top of part:** set the height of the piece in the Z axis (absolute).



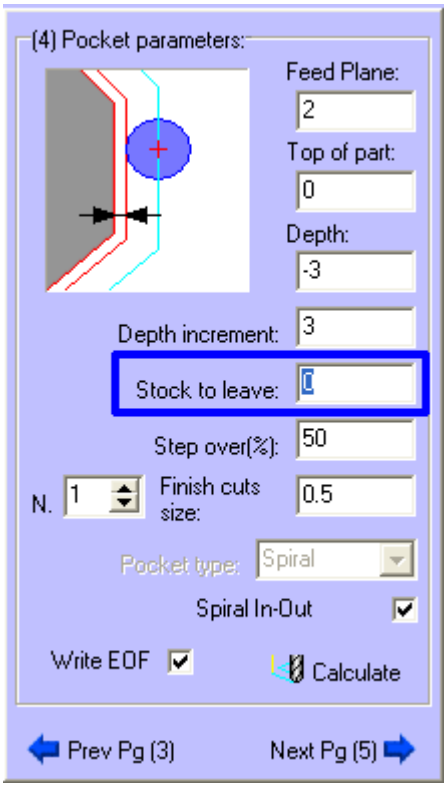
28. **Depth:** set the final machining depth (absolute).



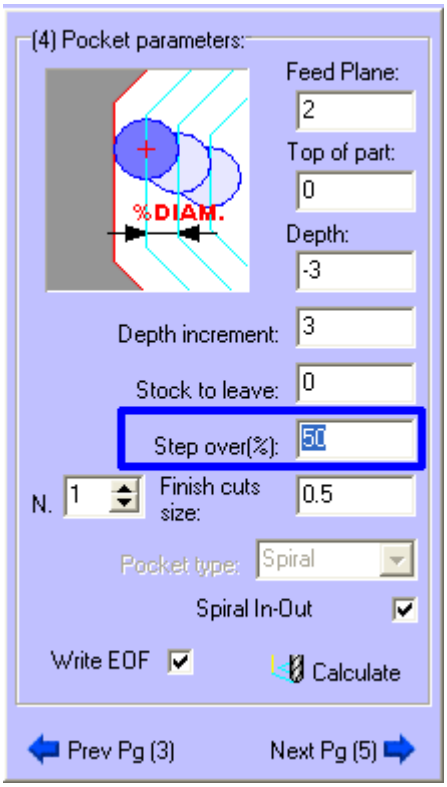
29. **Depth increment:** set the maximum amount of material to remove for each Z cut.



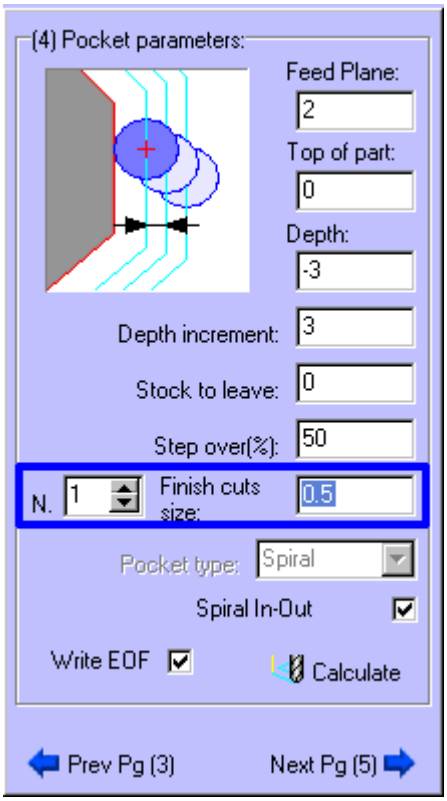
30. **Stock to leave:** set the amount of material to leave on profile; example if you need a finish pass with other tool.



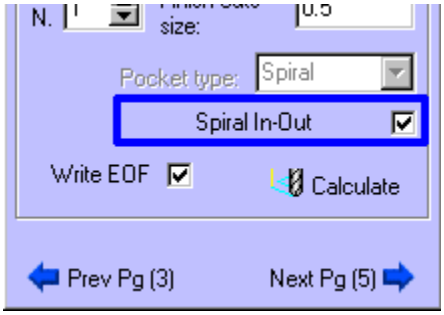
31. **Step over (%):** set the spacing between each pass. This is a percentage of the cutter diameter.



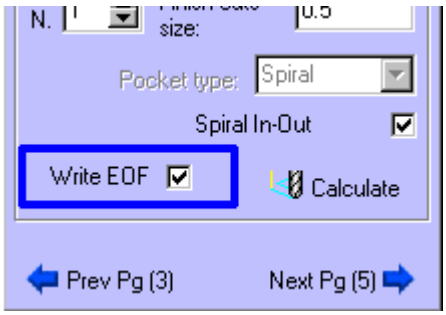
32. **Number of finish cuts / Finish cut size:** set the number of finish pass and the amount of material to remove for each finish pass.



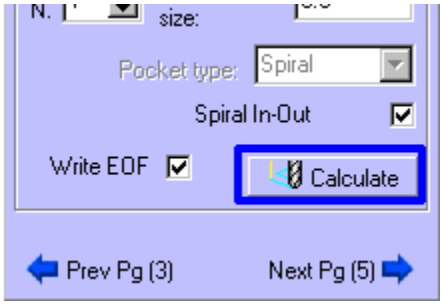
33. **Spiral In-Out:** "On" the toolpath spirals from the center to the external boundary of the pocket. "Off" the toolpath spirals from the external to the center of the pocket.



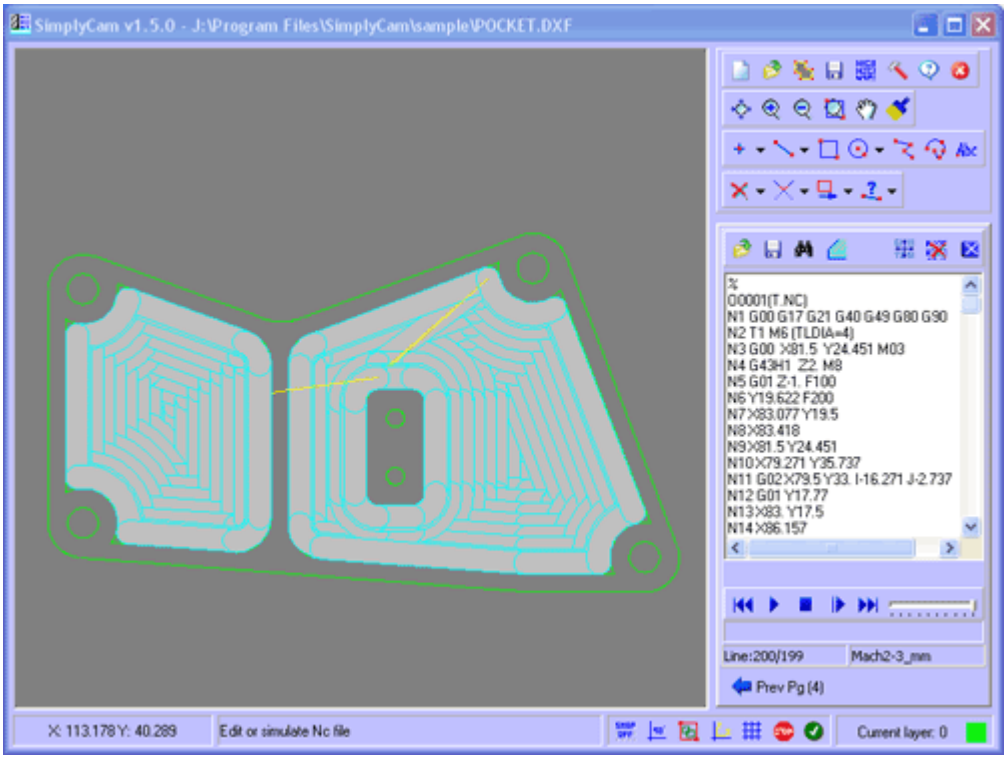
34. **Write EOF:** set this parameter for write in the Nc file the "End Of File" section (typically M2 or M30) .



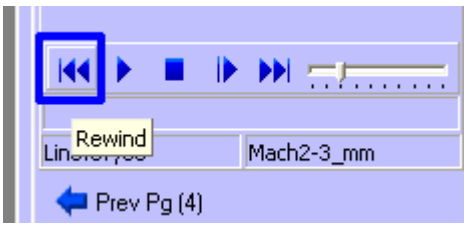
35. Press the "Calculate" button to machining the chained geometry with the cutting parameters.



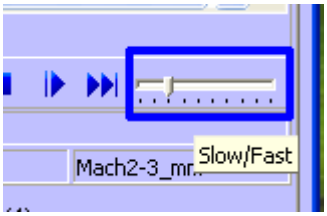
36. The "Nc File" page is displayed and in the graphic area the toolpath is simulated.



37. Press the "Rewind" button.



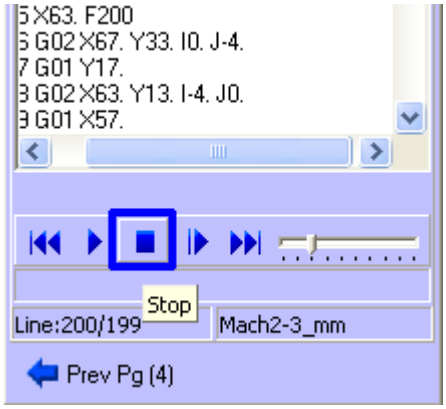
38. Move the slider near to slow position.



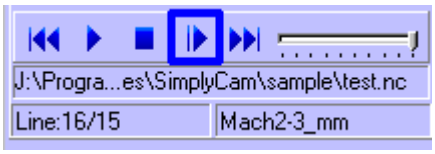
39. Press the "Play" button to simulate the toolpath (Yellow=Rapid, Cyan=Feed) in the graphic area.

40. The "Z Depth" panel indicator, reflect the actual Z tool position (Yellow=Rapid, Cyan=Feed).

41. Press the "Stop" button to break the simulation.

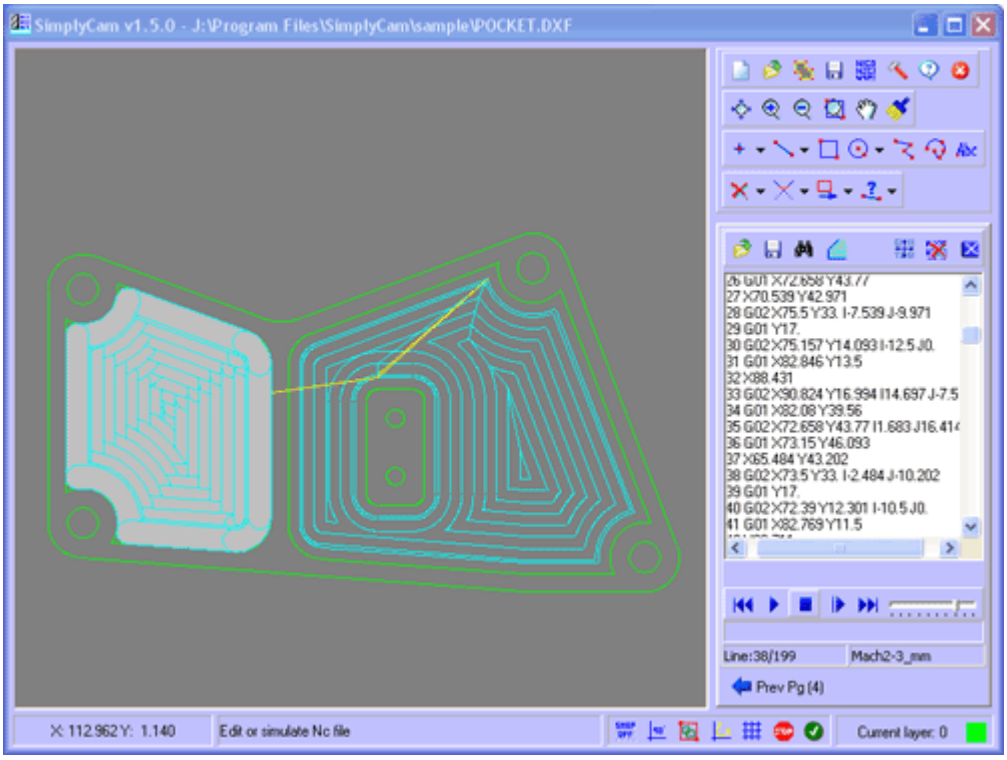


42. Press the "Step" button many time....



43. Set the "Wireframe / Solid simulation" button on Wireframe state.

44. You have simulated the pocket toolpath in various mode. See below the example.



45. You have successfully created the pocket toolpath with SimplyCam.

Caution:
CNC machines are potentially dangerous. The post-processor can output code unsuitable for your machine's control. Check the Nc file before sending it to a CNC machine.