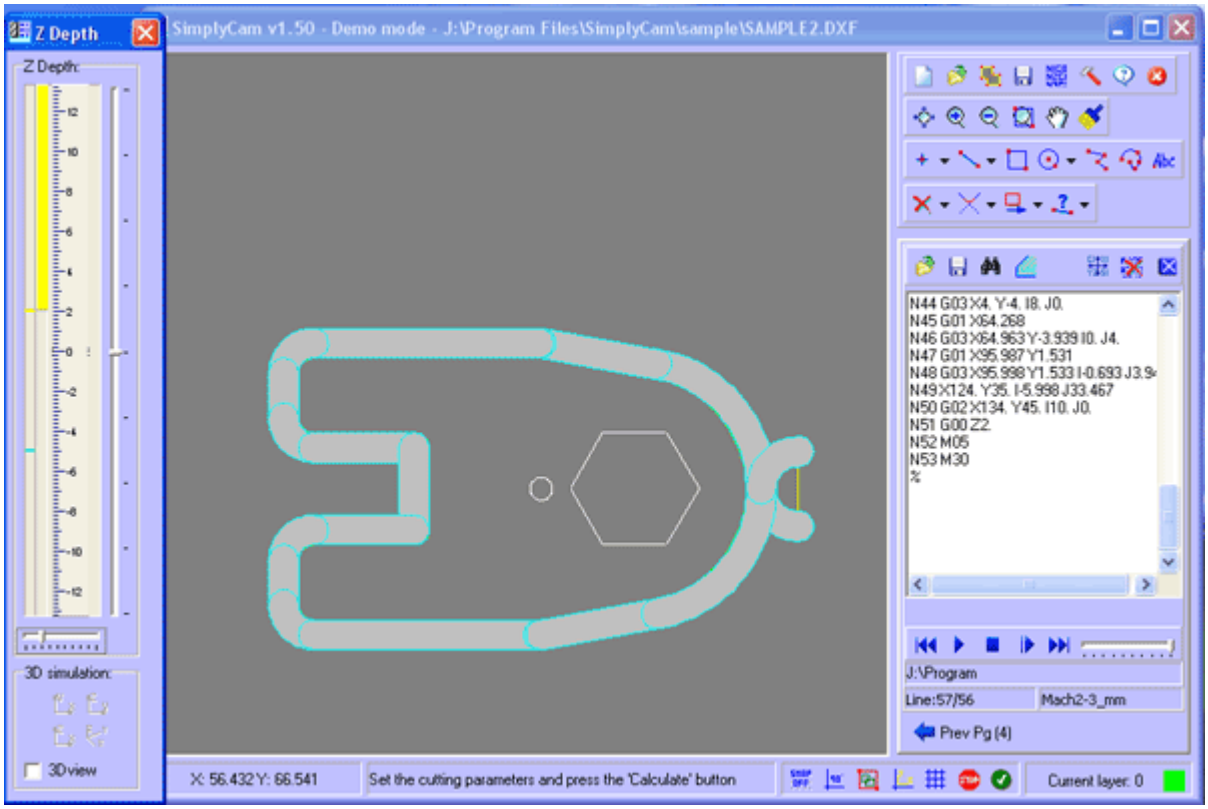
SimplyCam

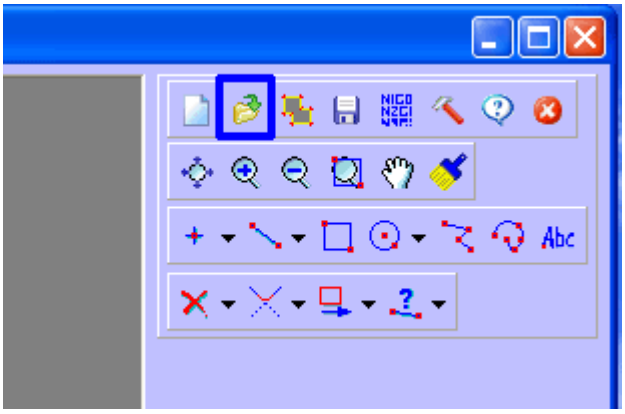
Tutorial 2 - Open Dxf file and create the outside Contour toolpath.

In this tutorial you will open a Dxf file and create the toolpath that cut the external of the part.



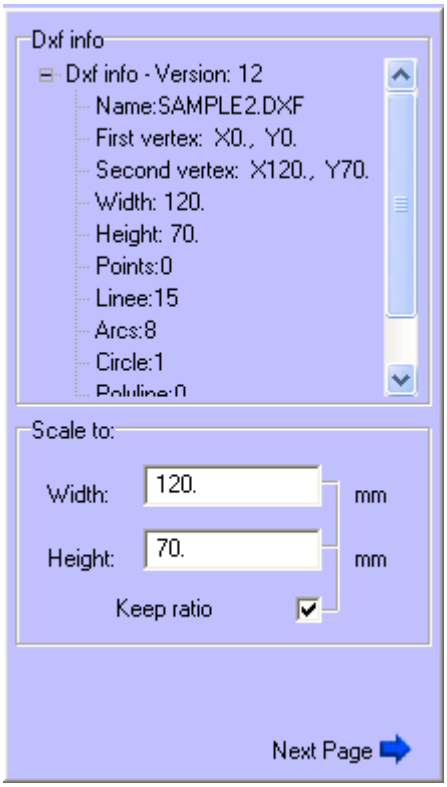
Caution:
CNC machines are potentially dangerous. The post-processor can output code unsuitable for your machine's control. Check the Nc file before sending it to a CNC machine.

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

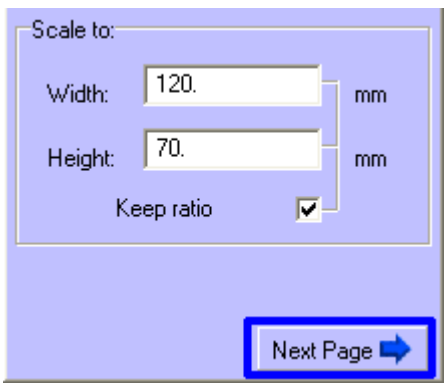


2. Select in "..\SimplyCam\Sample\" folder the **SAMPLE_CONTOUR.DXF** file.

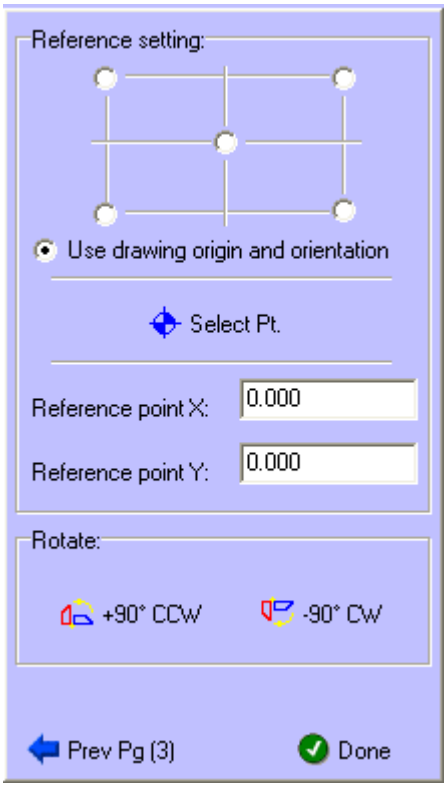
3. The Info panel will appear with the info and dimension of image.



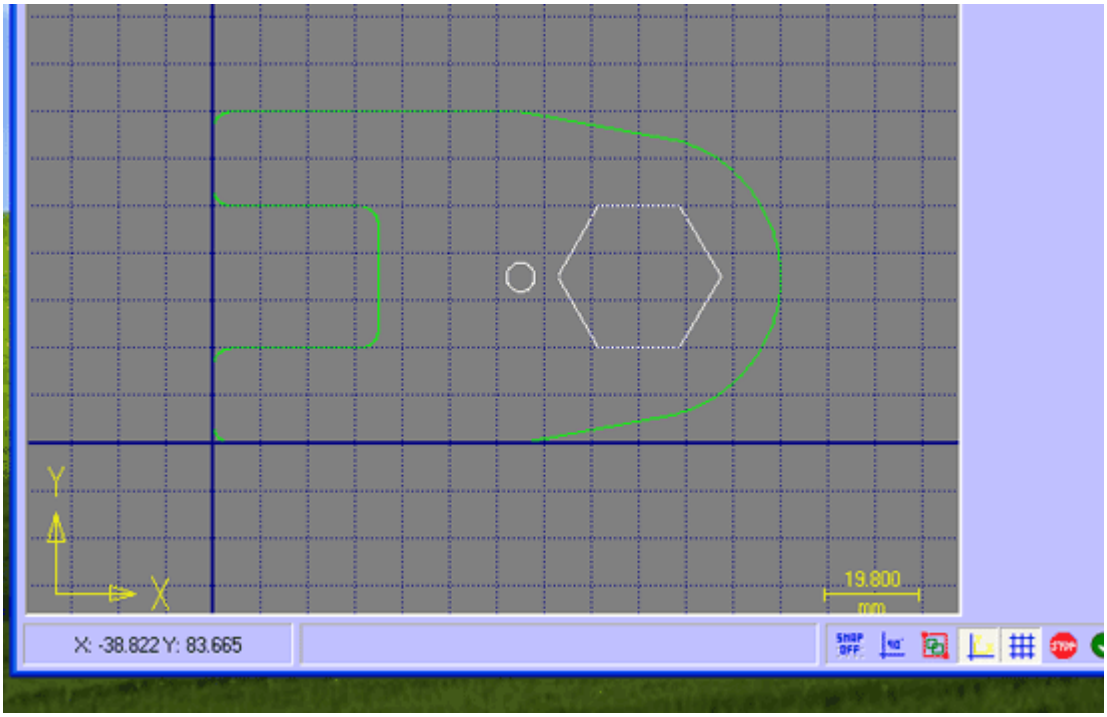
4. Scale the drawing dimension if desidered and press "Next Page" button.



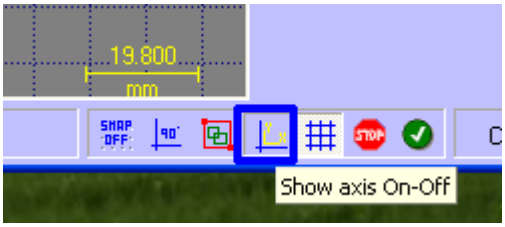
5. Define the reference point of the drawing and press "Done" button.



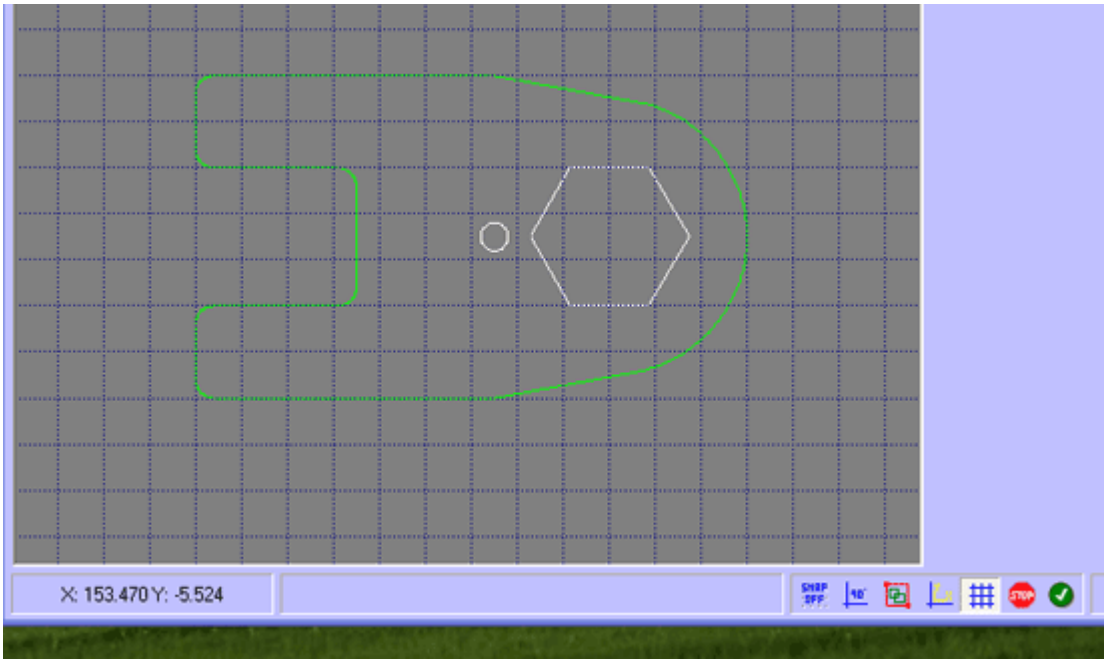
6. The drawing is displayed in graphic area with grid, axis direction, origin and scale info.



7. Press the "Show axis" button in status bar on Off state.

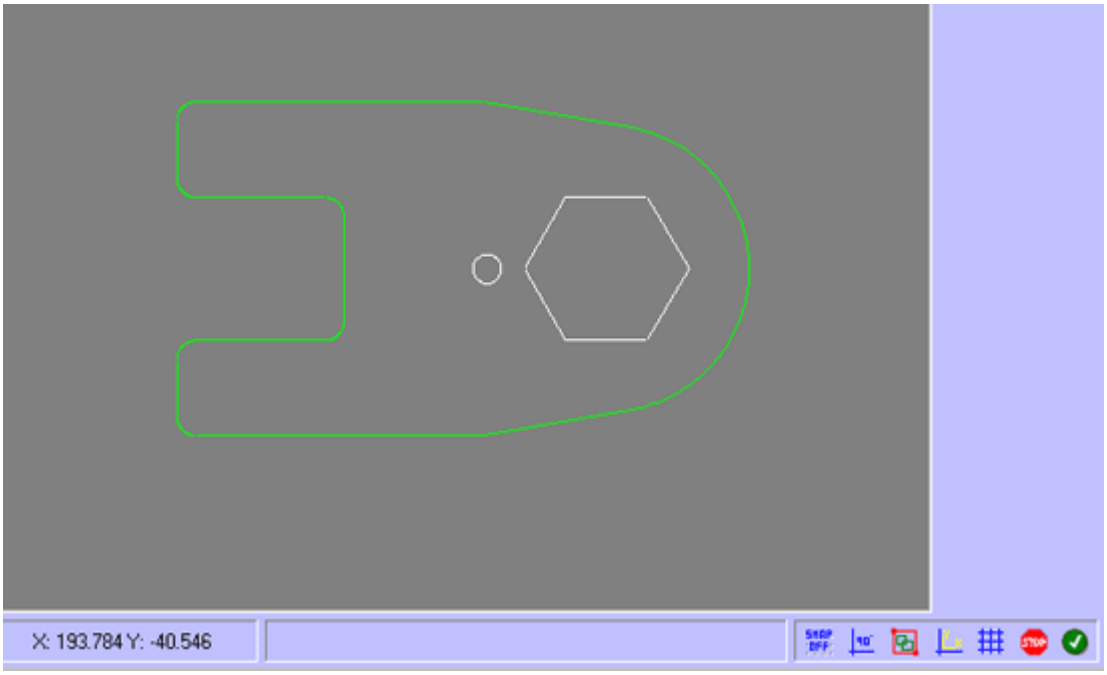


8. The drawing is displayed in graphic area with grid, but now without the axis direction, origin and scale info.

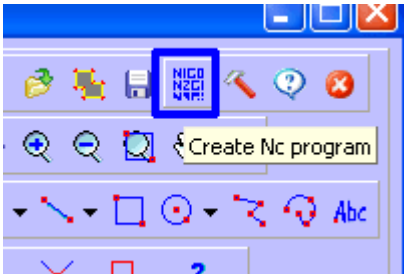


9. Press the "Show grid" button in status bar on Off state.

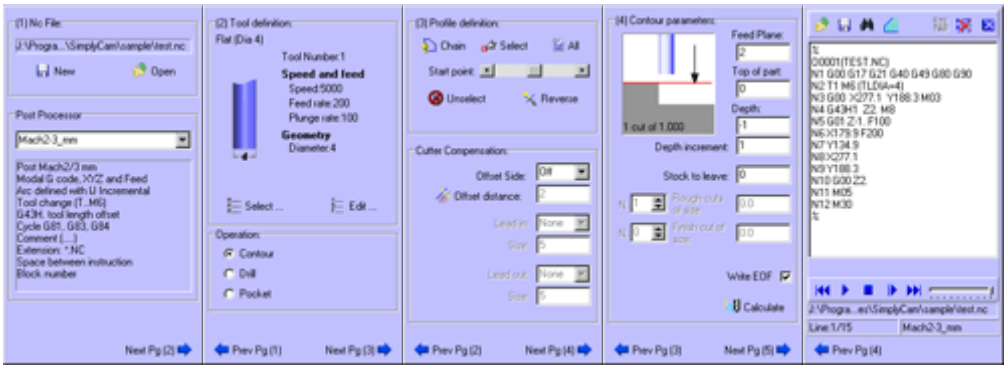
10. The drawing is displayed in graphic area now without grid.



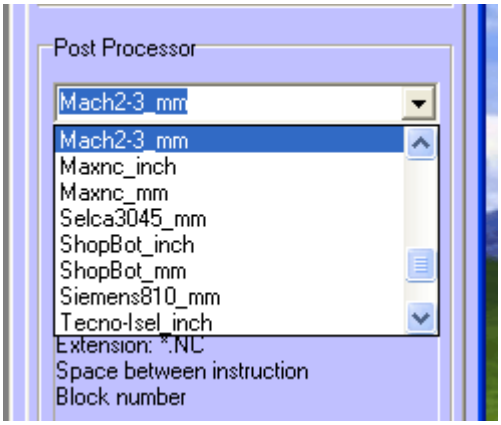
11. Press "Nc Program" button to go in toolpath section.



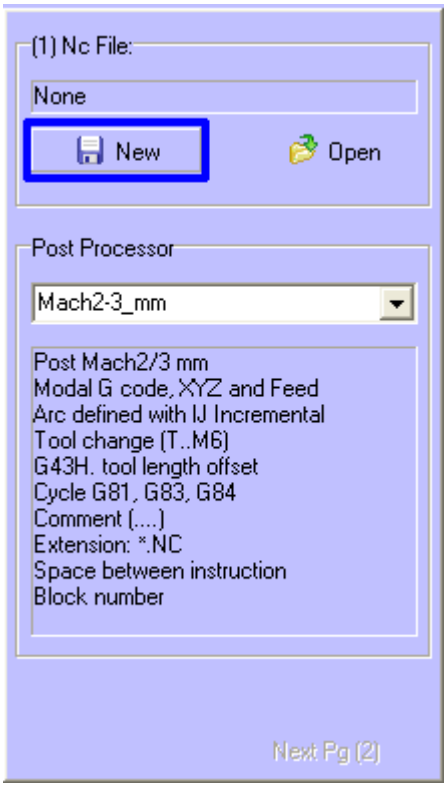
This section contain 5 pages:
Page 1: File/Postprocessor definition
Page 2: Tool/Operation definition
Page 3: Profile/Cutter compensation selection
Page 4: Cutting parameter setting
Page 5: Nc file simulating or editing



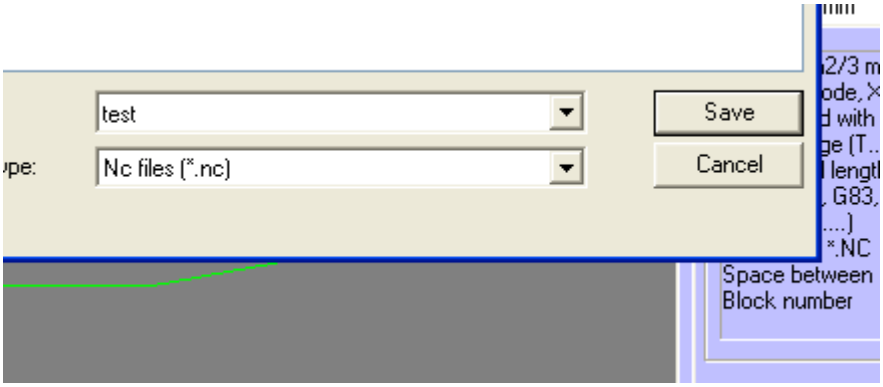
12. Select postprocessor of your Cnc machine.



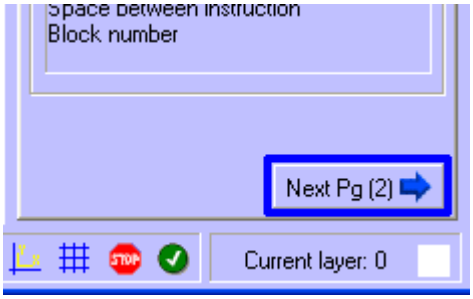
13. Press "New" button to create new toolpath.



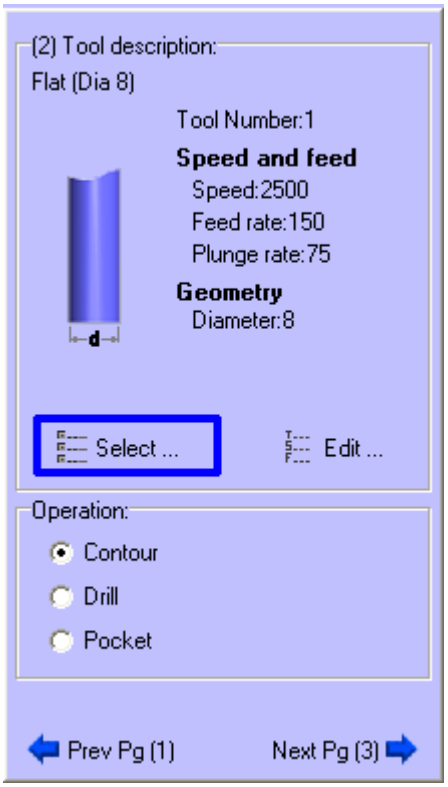
14. Type "test" in the Windows file dialog.



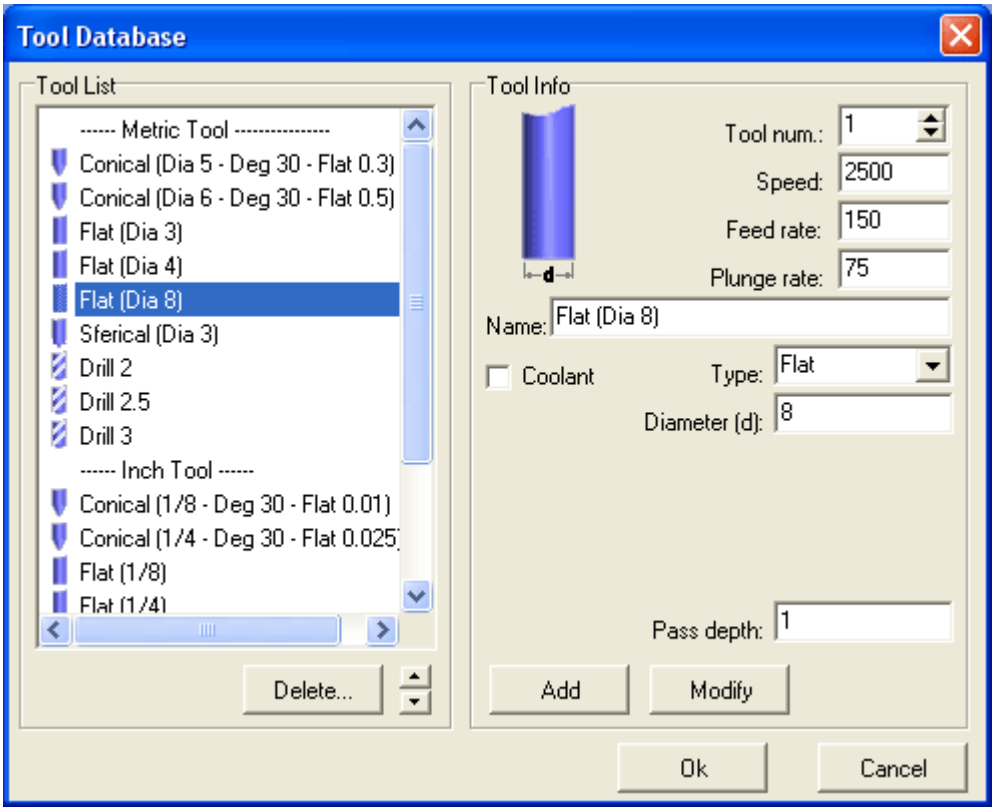
15. Press Next Pg(2) button.



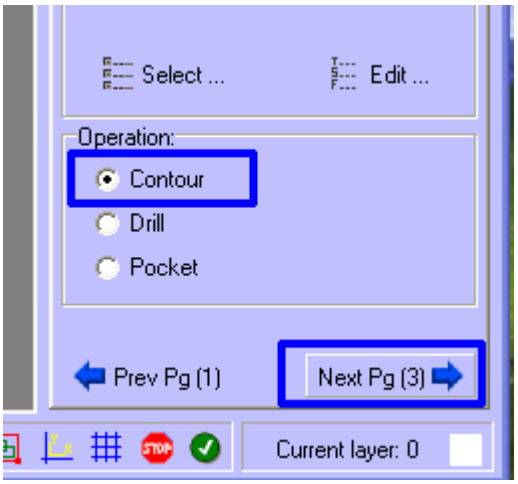
16. In Tool section press "Select" button to retrieve a tool from a tool library or create a new tool.



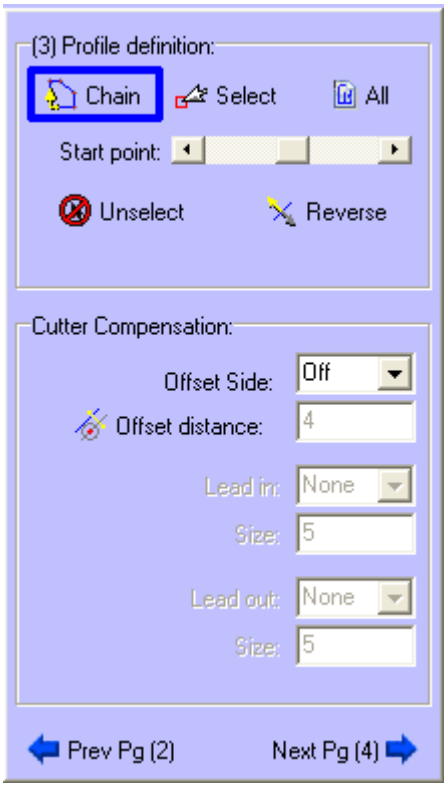
17. Click in the Tool List to select the Flat tool with **diameter 8**. Set feeds and speed of the tool. Press the "OK" button.



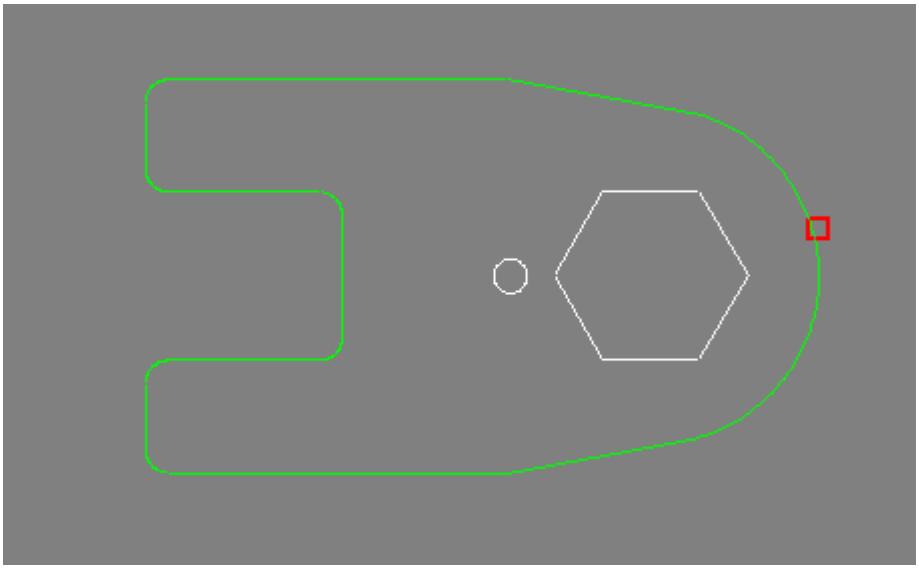
18. In the Operation section, select "Contour" and press "Next Pg(3)".



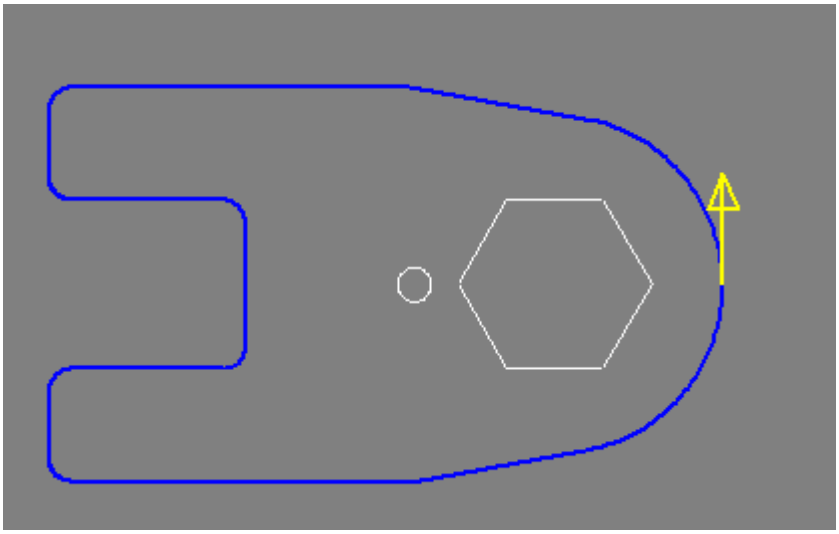
19. The "Profile definition" page appear. Press the "Chain" button.



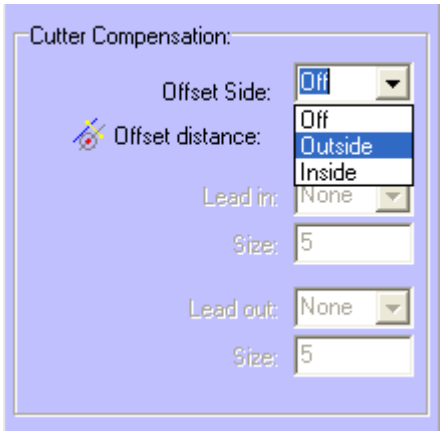
20. Pick the geometry near to stat point.



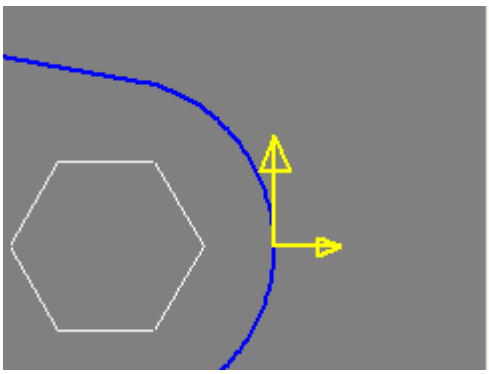
21. The yellow arrow appears to indicate the start point of contour and the direction of contour.
    The blue boundary is the chained profile.



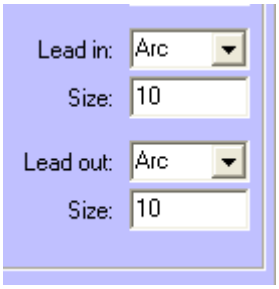
22. In the "Cutter compensation" section, set the "Offset Side" on OutSide.



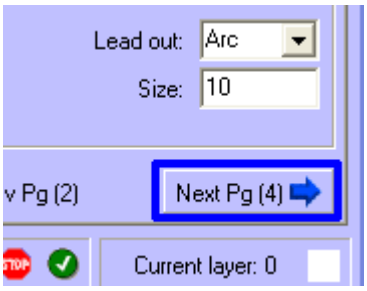
23. Another yellow arrow, smaller then previous, appears to indicate the direction of cutter compensation.
    The long arrow = Toolpath direction
    The small arrow = Cutter compensation direction.



24. Set the "Lead in" and "Lead out".

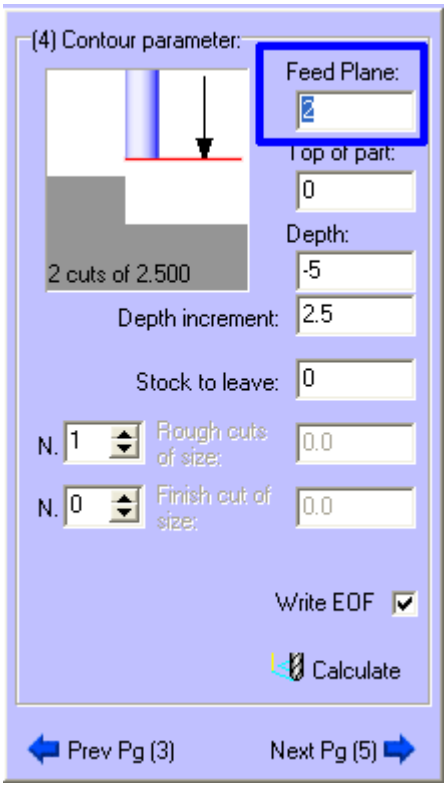


25. And then press "Next Pg(4)" button.

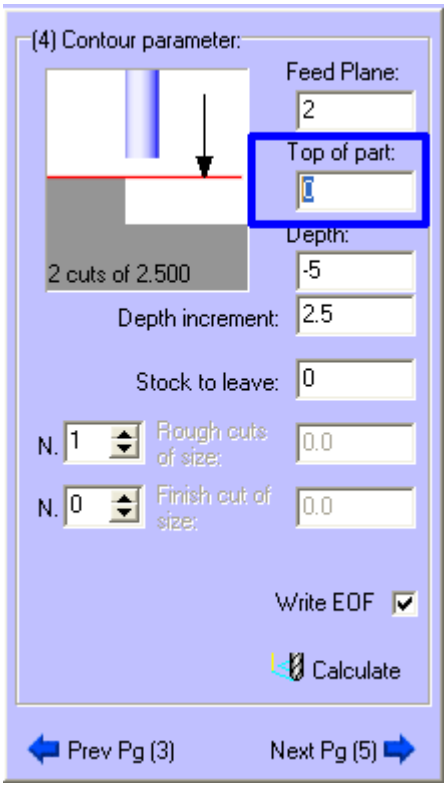


26. The "Contour parameter" page appear. Set up the following parameters:

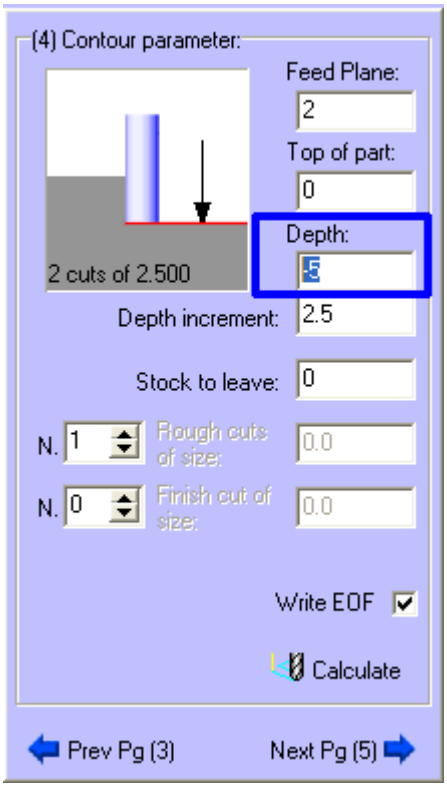
27. **Feed plane:** set the height that the tool rapids to (G0) before changing to the feed rate (G1) to enter in the part (absolute).



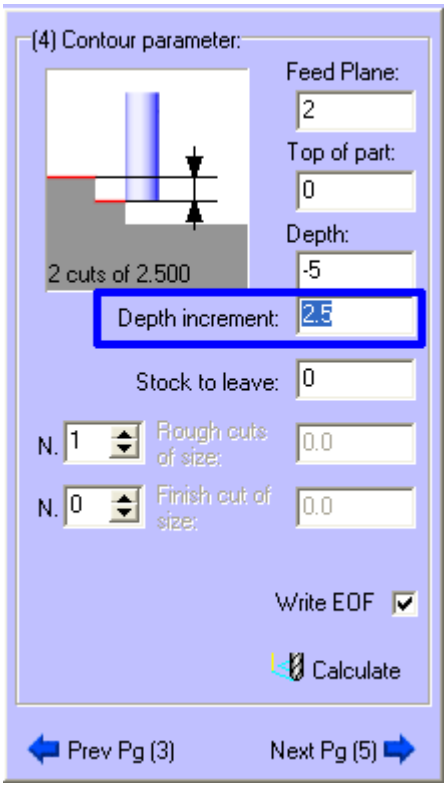
28. **Top of part:** set the height of the piece in the Z axis (absolute).



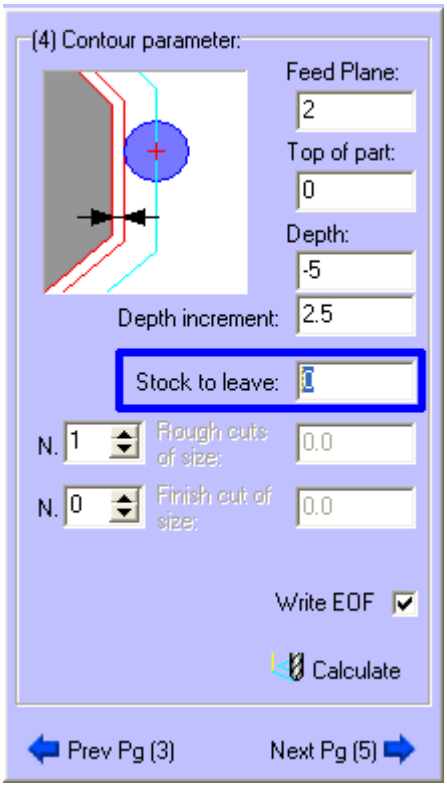
29. **Depth:** set the final machining depth (absolute).



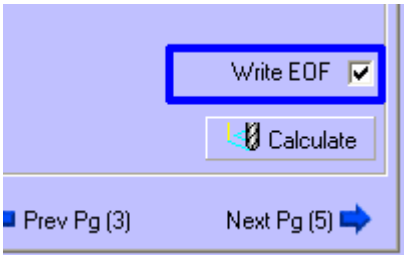
30. **Depth increment:** set the maximum amount of material to remove for each Z cut.



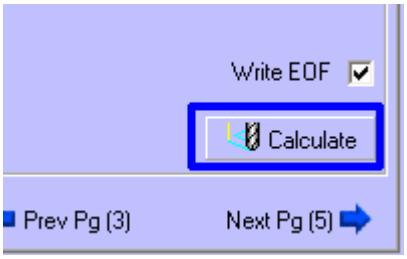
31. **Stock to leave:** set the amount of material to leave on profile; example if you need a finish pass with other tool.



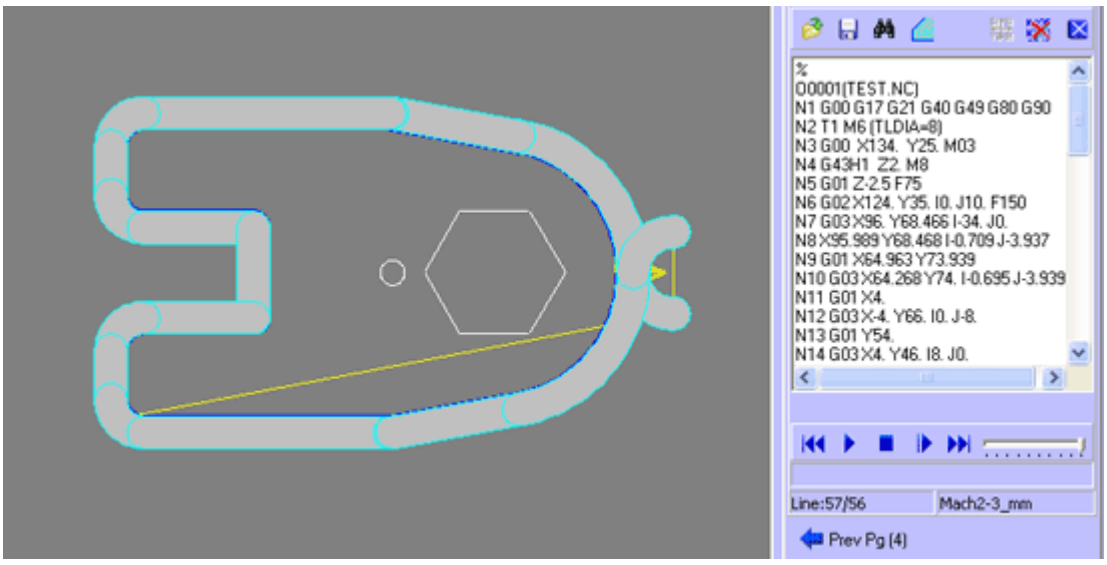
32. **Write EOF:** set this parameter for write in the Nc file the "End Of File" section (typically M2 or M30) .



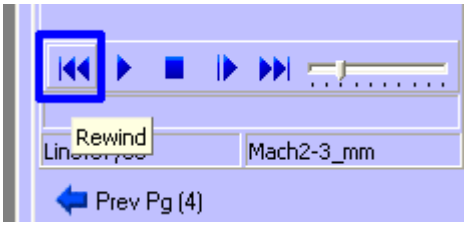
33. Press the "Calculate" button to machining the chained geometry with the cutting parameters.



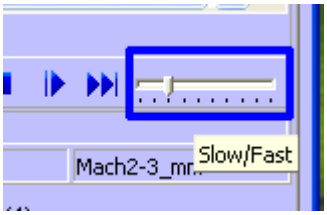
34. The "Nc File" page is displayed and in the graphic area the toolpath is simulated.



35. Press the "Rewind" button.

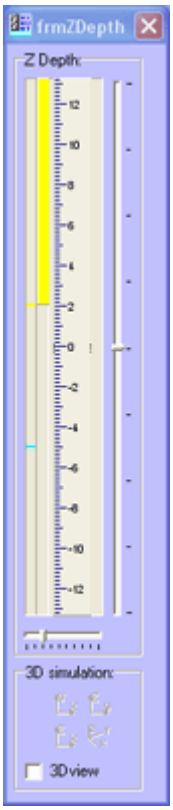


36. Move the slider near to slow position.

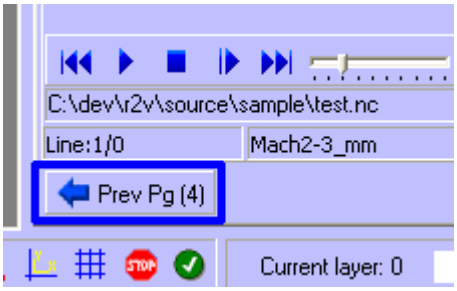


37. Press the "Play" button to simulate the toolpath (Yellow=Rapid, Cyan=Feed) in the graphic area.

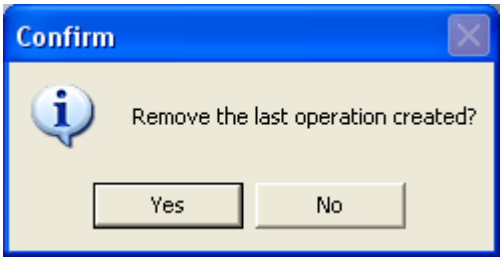
38. The "Z Depth" panel indicator reflect the actual Z tool position (Yellow=Rapid, Cyan=Feed).



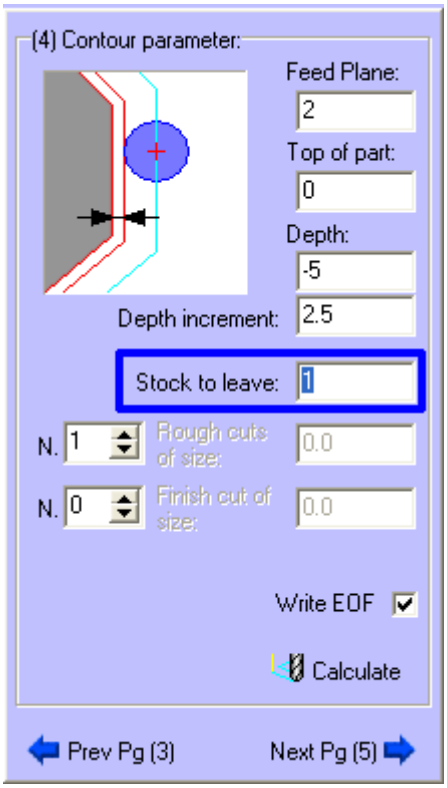
39. Press the "Prev Pg(4)" button.



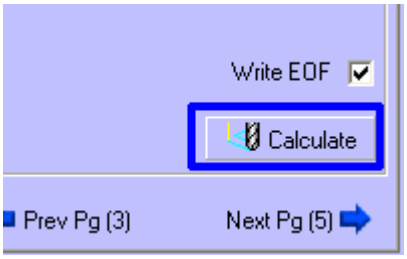
40. "Remove the last operation created?" Yes, we want to modify the toolpath.



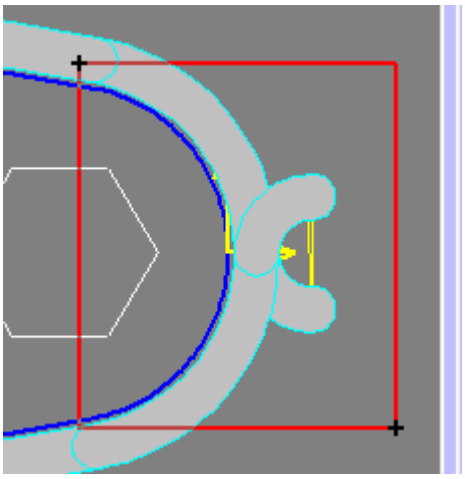
41. Modify the parameter "Stock to leave" to value 1.



42. Press again the "Calculate" button to machining the chained geometry with the new cutting parameter.

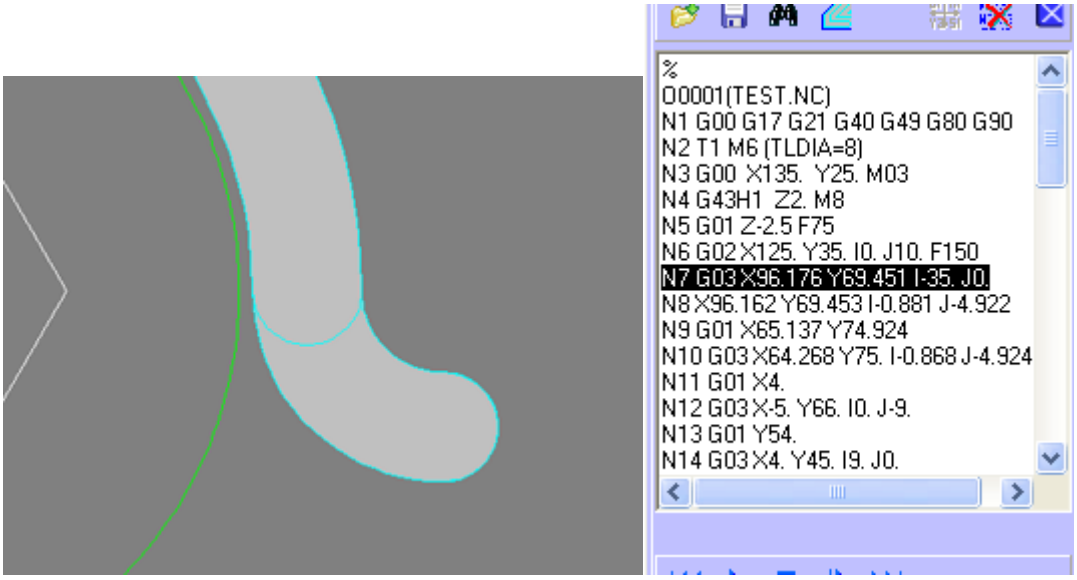


43. Click on the "Zoom Window" button and select in graphic area two points, as indicated in the below image.



44. Press the "Rewind" button and the "Step" button. Press the "Step" button many time....

45. The toolpath is simulate in "step to step" mode, outside the green part.



46. You have successfully created the outside Contour 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.