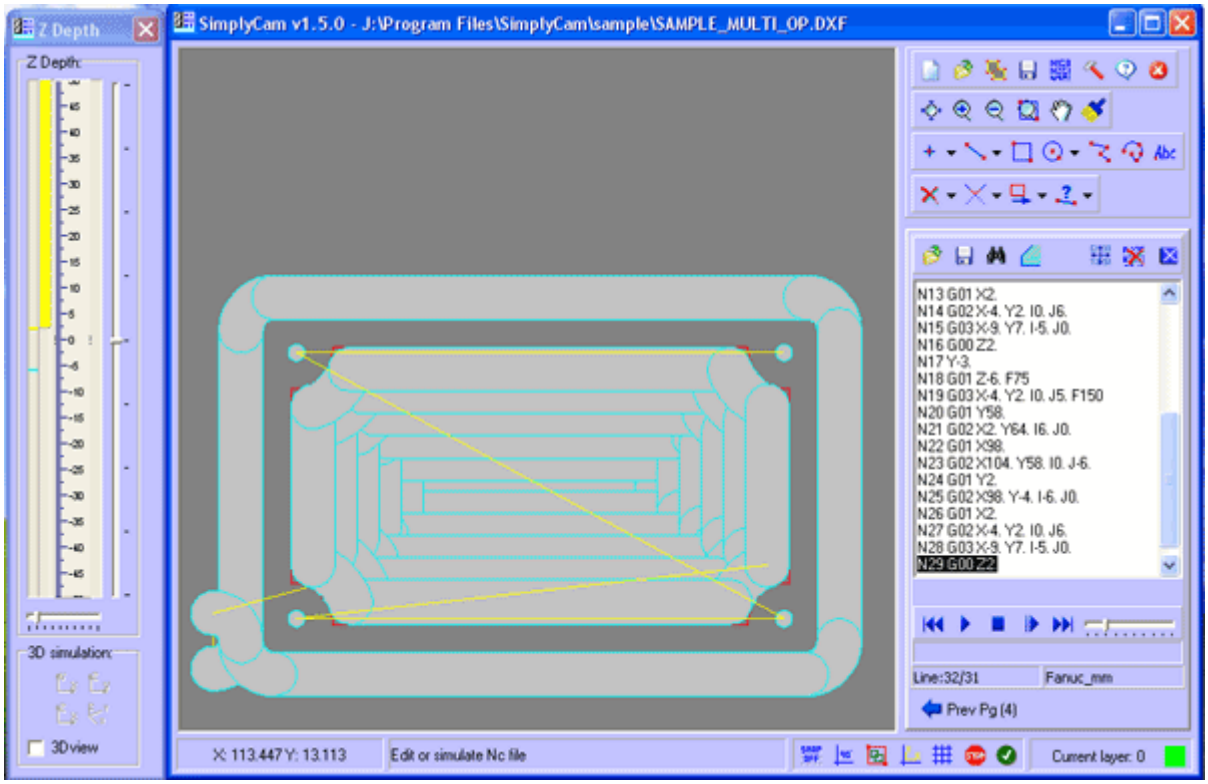
SimplyCam

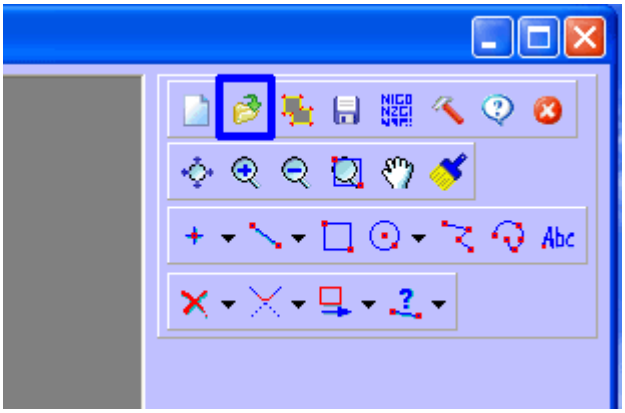
Tutorial 4 - Open Dxf file and create multiple toolpaths (Contour, Pocket and Drill).

In this tutorial you will open a Dxf file and create the toolpath that cut the external of the part, another toolpath to remove the material contained in a closed profile, and a tool-change with the drill operation.



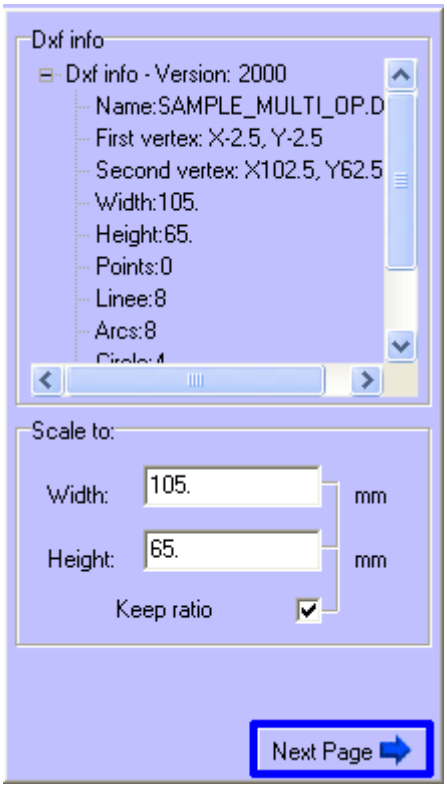
Caution:
CNC machines are potentially dangerous. The post-processor can output code unsuitable for your machine's control. Check the Nc file before sending it to a CNC machine.

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

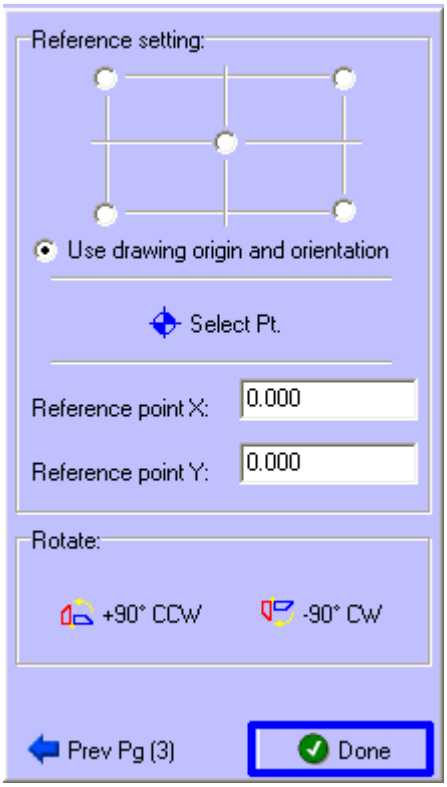


2. Select in "..\SimplyCam\Sample\" folder the **SAMPLE_POCKET.DXF** file.

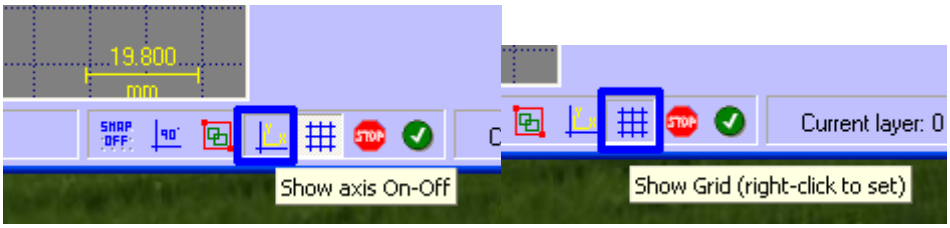
3. The Info panel will appear with the info and dimension of image. Press "Next Page" button.



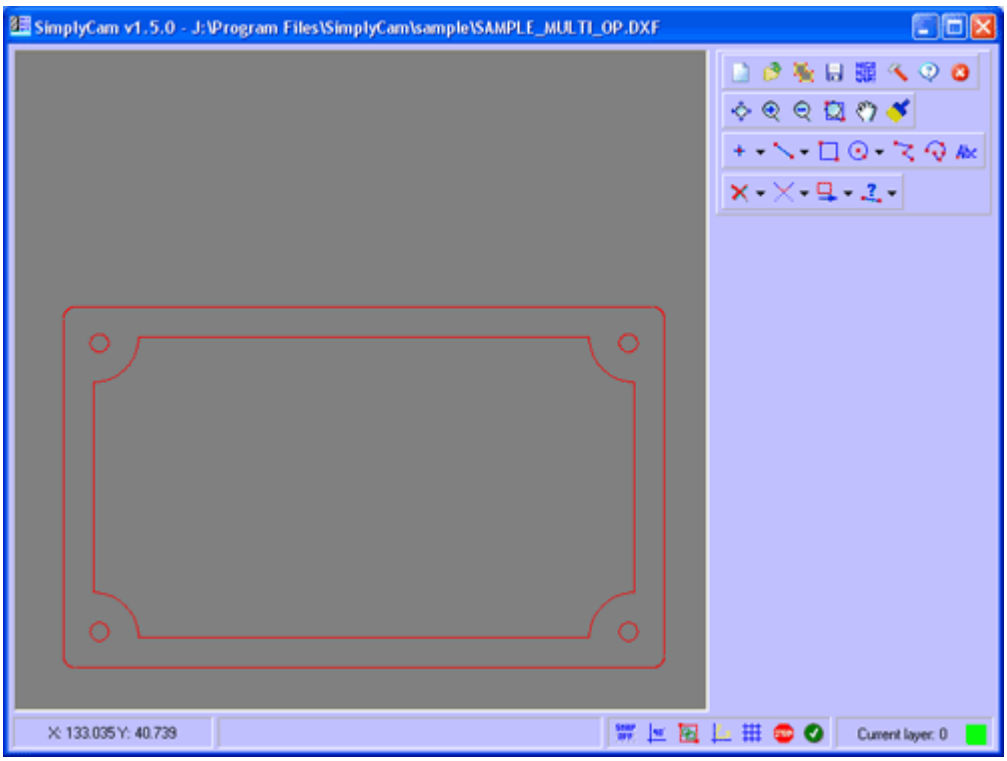
4. Define the reference point of the drawing and press "Done" button.



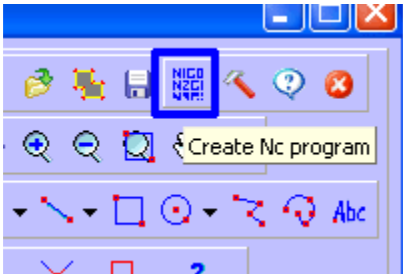
5. Turn in the "Off" state the button "Show grid" and "Show Axis".



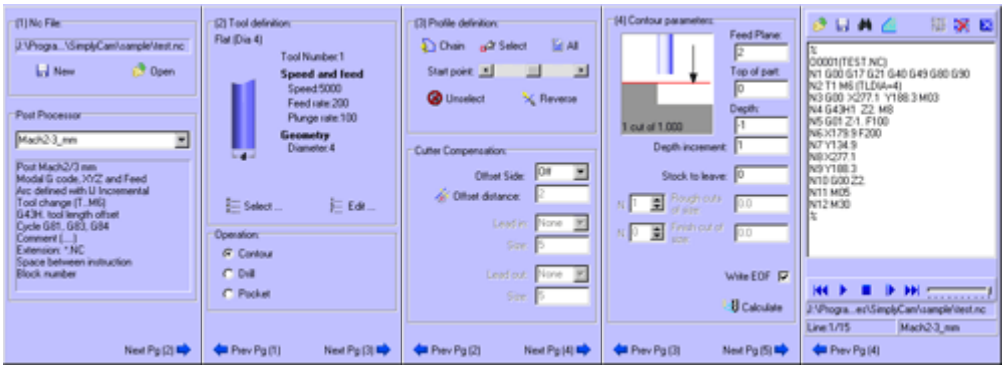
6. The drawing is displayed in graphic area without the grid, the axis direction, the origin and the scale info.



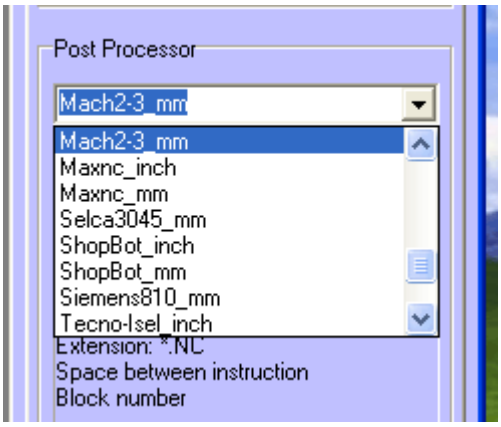
7. Press "Nc Program" button to go in toolpath section.



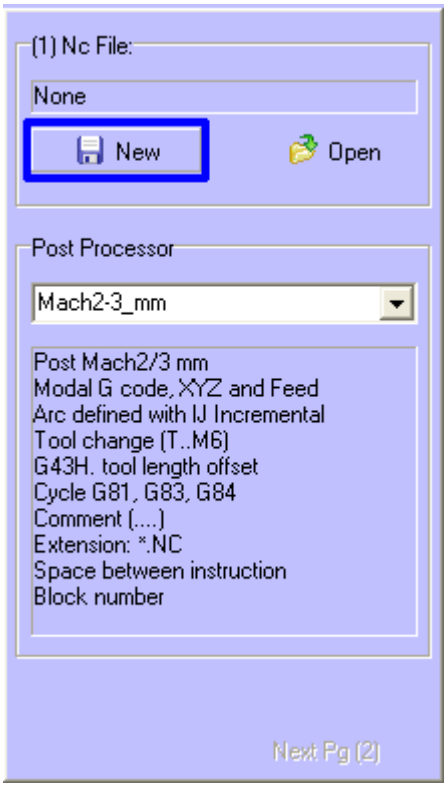
This section contain 5 pages:
Page 1: File/Postprocessor definition
Page 2: Tool/Operation definition
Page 3: Profile/Cutter compensation selection
Page 4: Cutting parameter setting
Page 5: Nc file simulating or editing



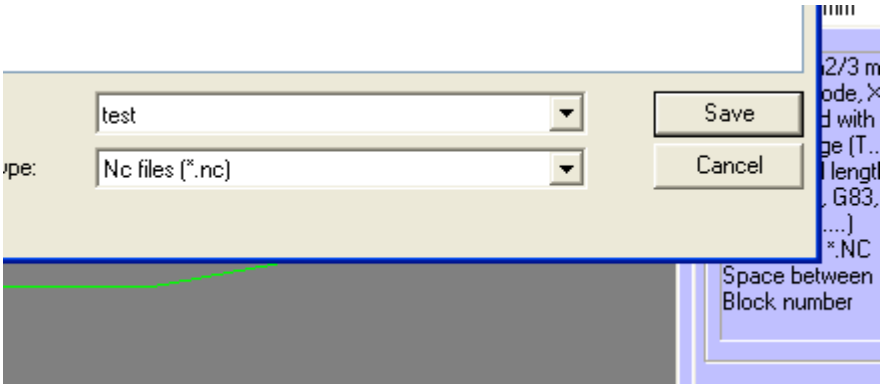
8. Select the postprocessor of your Cnc machine.



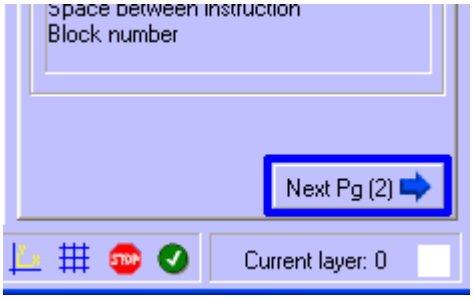
9. Press "New" button to create new toolpath.



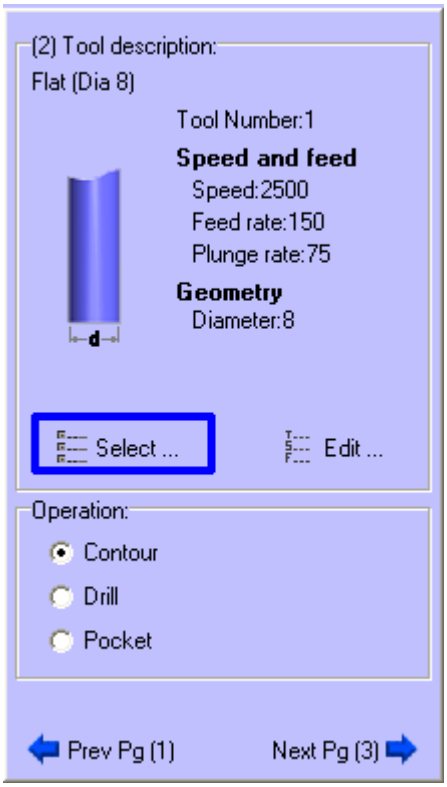
10. Type "test" in the Windows file dialog.



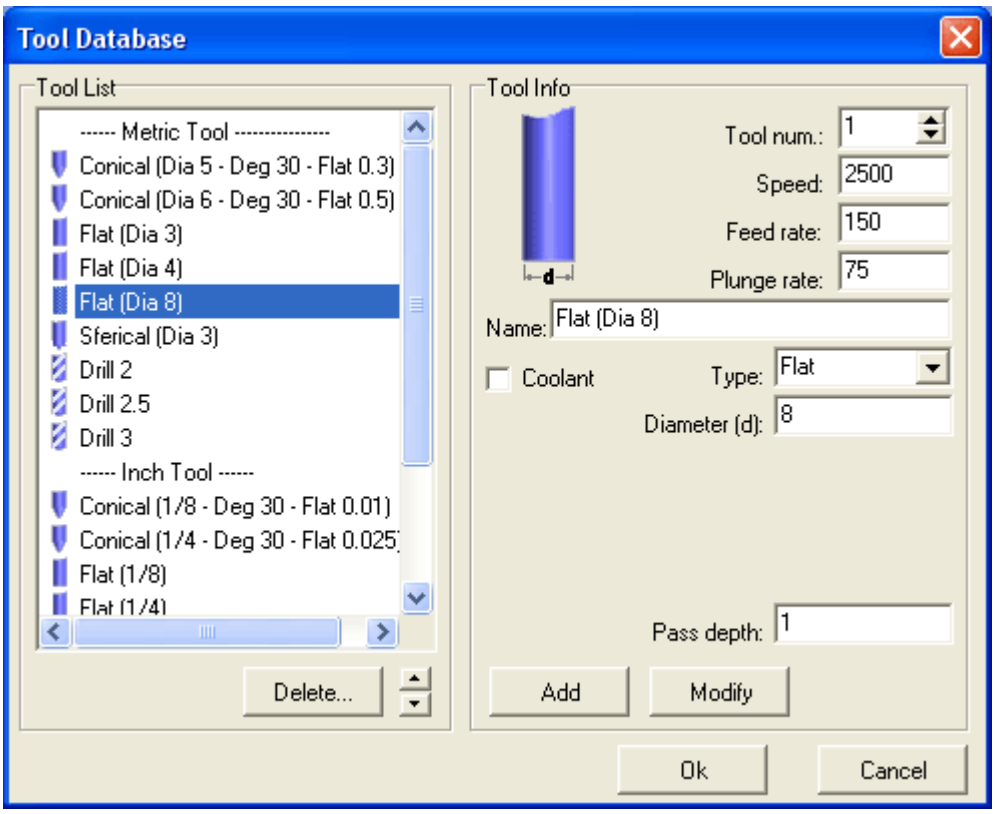
11. Press "Next Pg(2)" button.



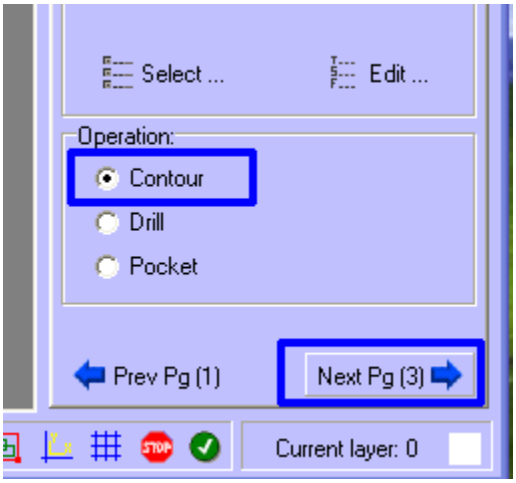
12. In Tool section press "Select" button to retrieve a tool from a tool library or create a new tool.



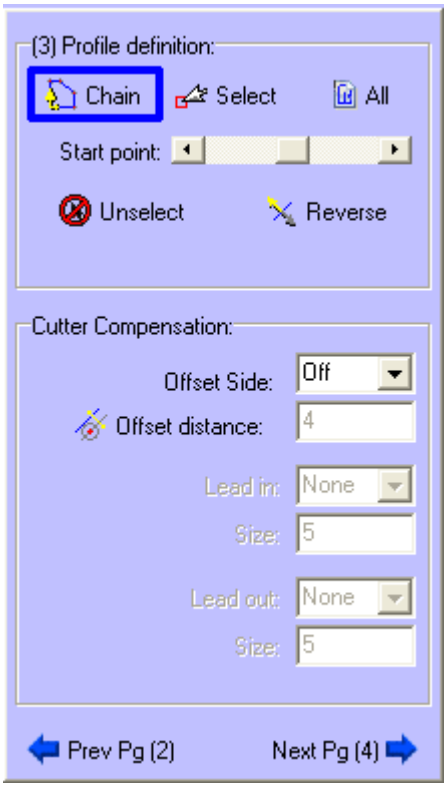
13. Click in the Tool List to select the Flat tool with **diameter 8**. Set feeds and speed of the tool. Press the "OK" button.



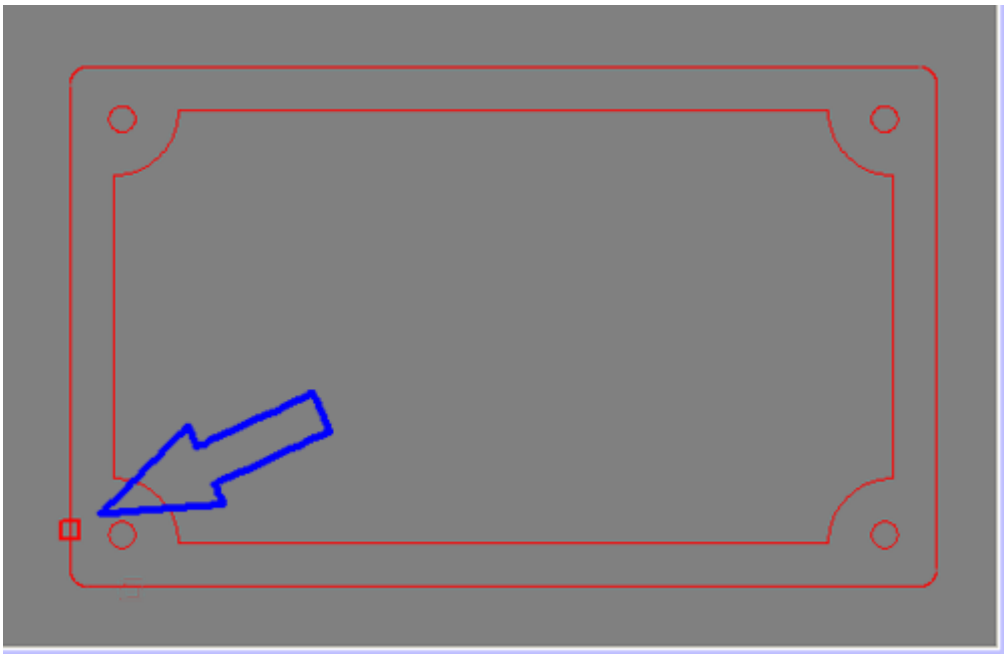
14. In the Operation section, select "Contour" and press "Next Pg(3)".



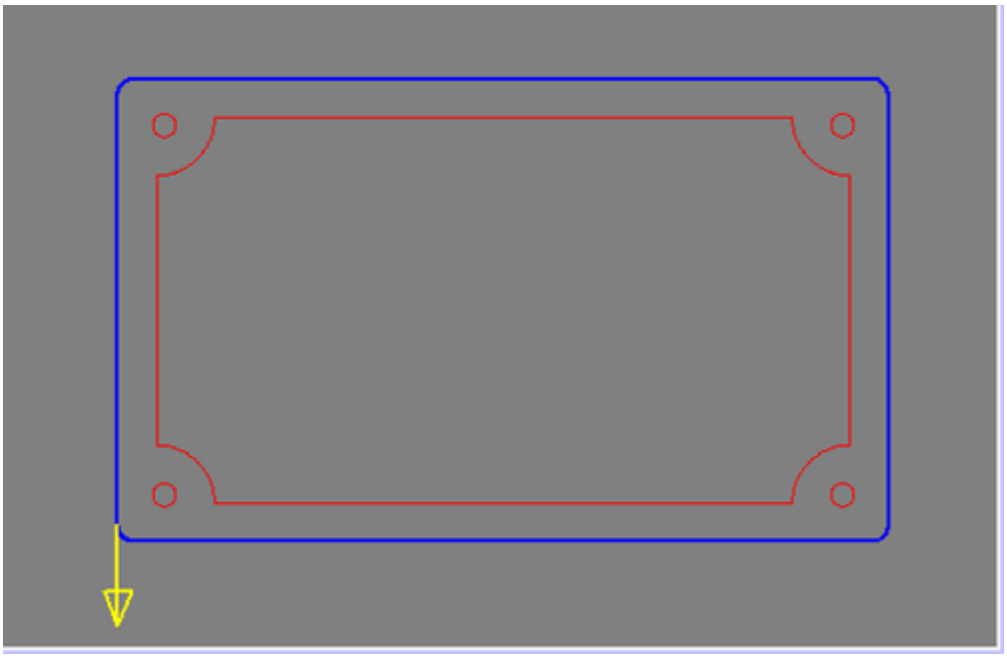
15. The "Profile definition" page appears. Press the "Chain" button.



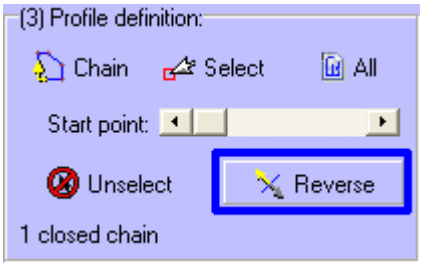
16. Pick the geometry near to start point as indicate by red square.



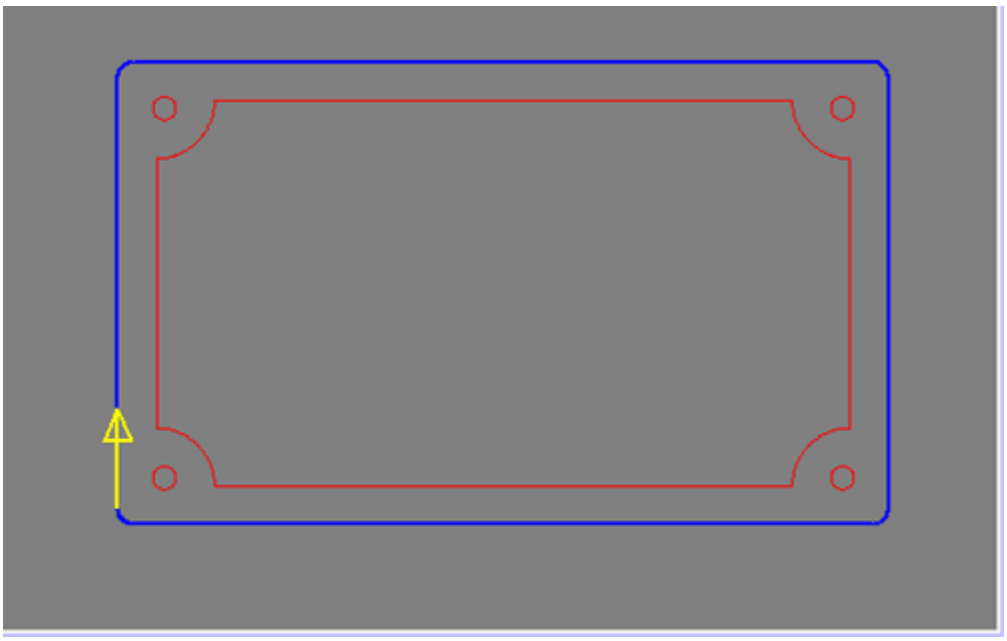
17. The yellow arrow appears to indicate the start point of contour and the direction of contour.
    The blue boundary is the chained profile.



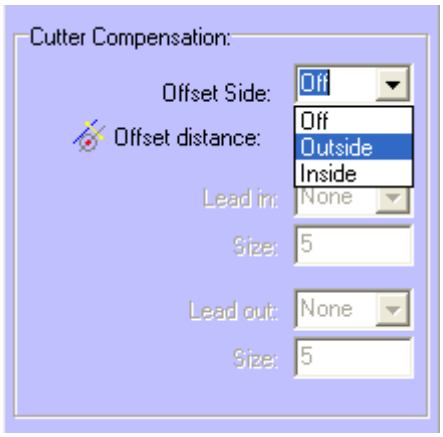
18. In the "Profile definition" press the "Reverse" button and press "Next Pg(3)".



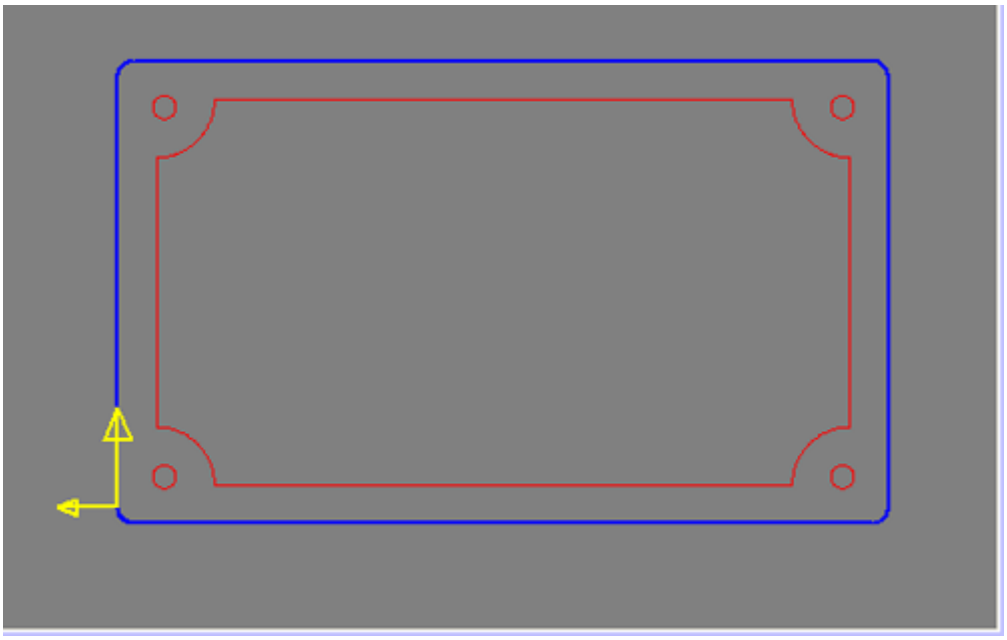
19. The new direction of the chain is showing.



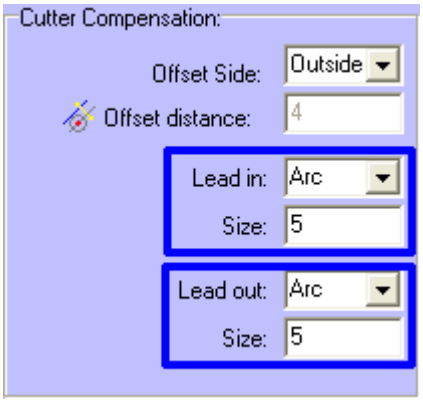
20. In the "Cutter compensation" section, set the "Offset Side" on OutSide.



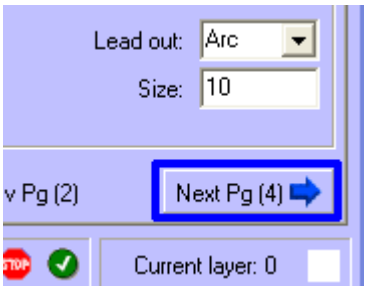
21. Another yellow arrow, smaller then previous, appears to indicate the direction of cutter compensation.
    The long arrow = Toolpath direction
    The small arrow = Cutter compensation direction.



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

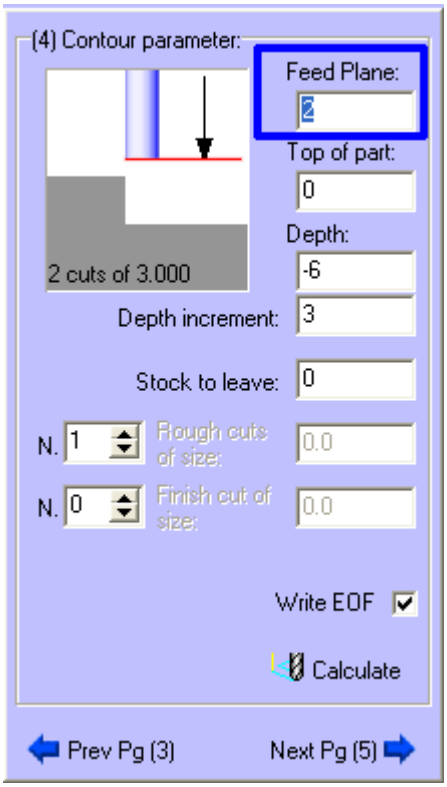


23. And then press "Next Pg(4)" button.

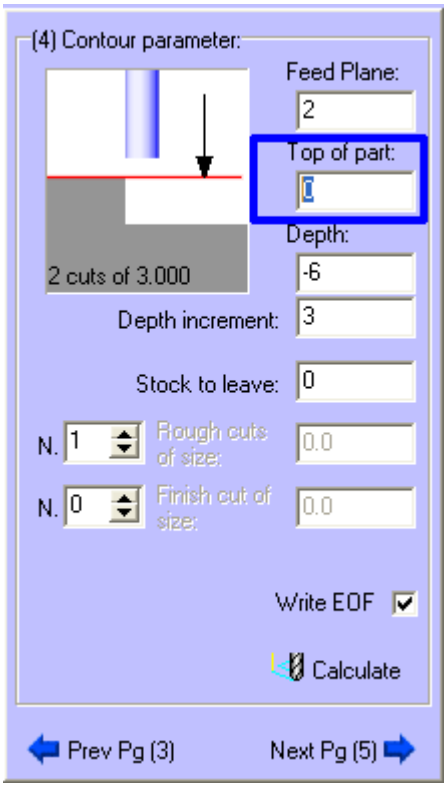


24. The "Contour parameter" page appear. Set up the following parameters:

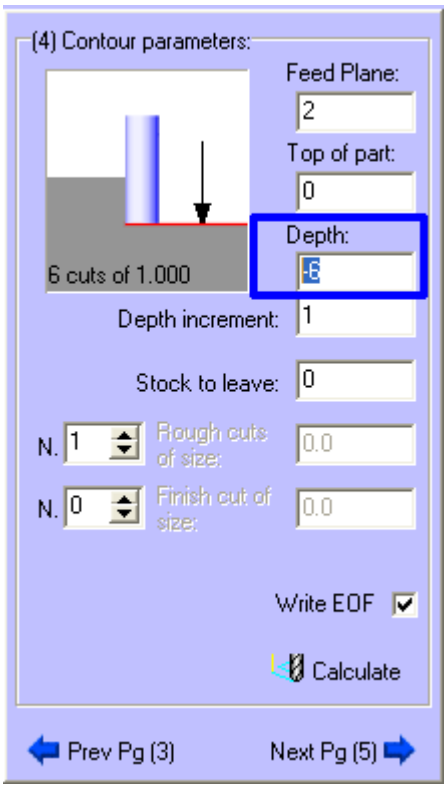
25. **Feed plane:** set the height that the tool rapids to (G0) before changing to the feed rate (G1) to enter in the part (absolute).



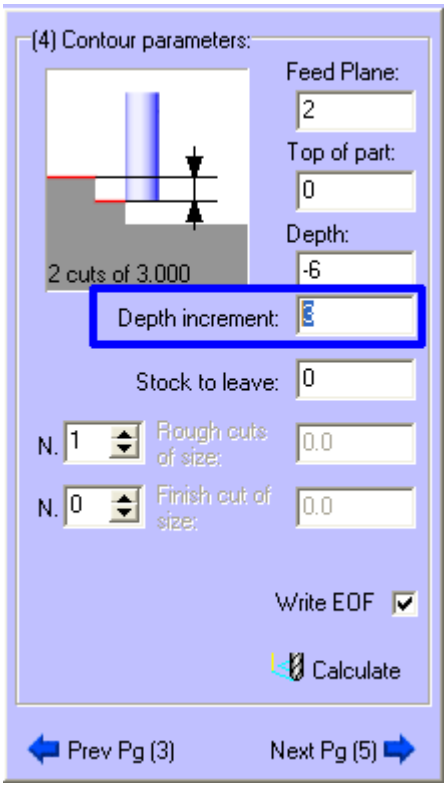
26. **Top of part:** set the height of the piece in the Z axis (absolute).



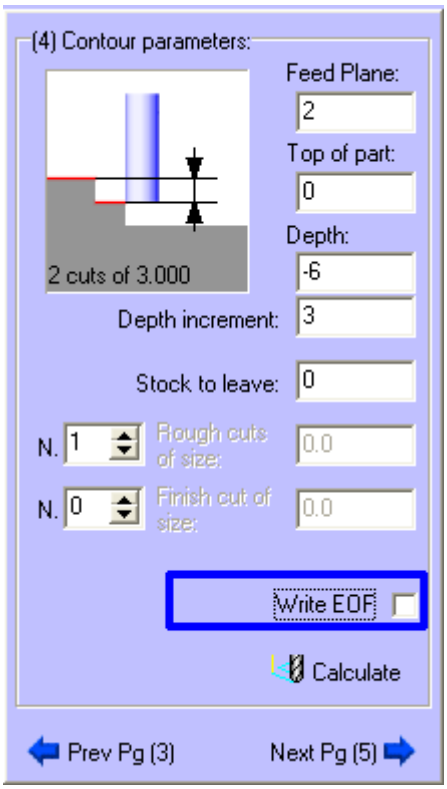
27. **Depth:** set the final machining depth (absolute).



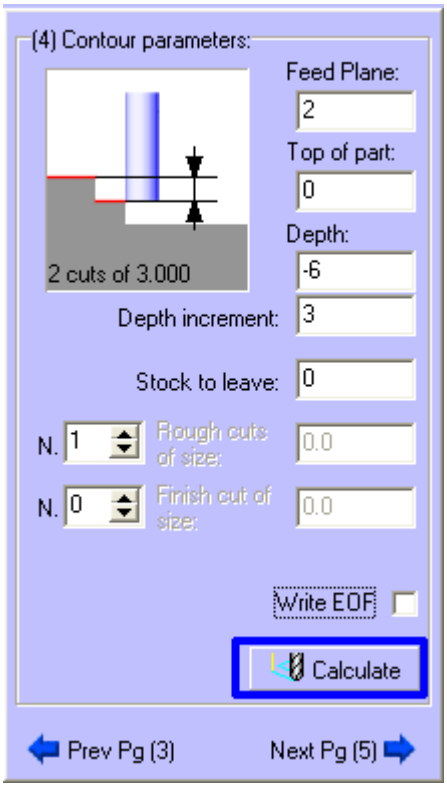
28. **Depth increment:** set the maximum amount of material to remove for each Z cut.



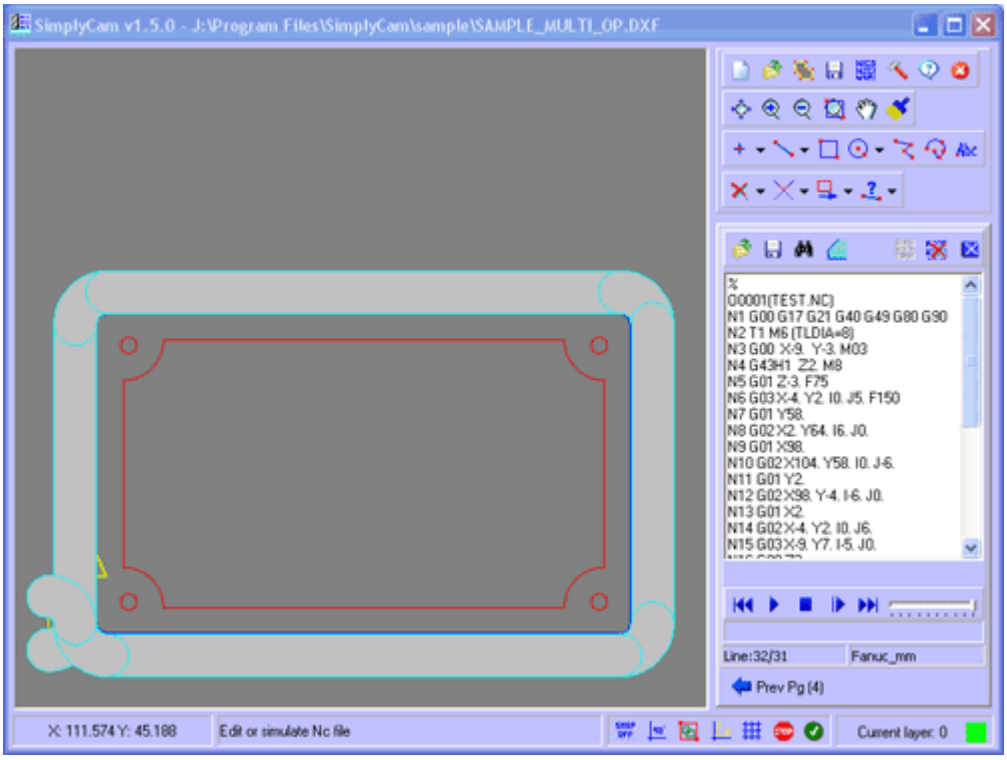
29. **Write EOF:** Turn "Off" this parameter for not write in the Nc file the "End Of File" section (typically M2 or M30) .



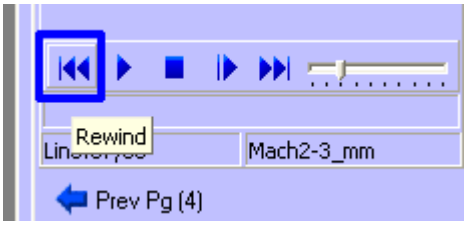
30. Press the "Calculate" button to machining the chained geometry with the cutting parameters.



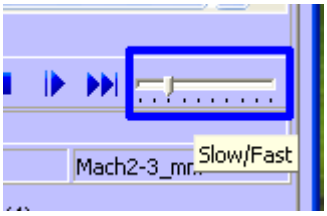
31. The "Nc File" page is displayed and in the graphic area the toolpath is simulated.



32. Press the "Rewind" button.

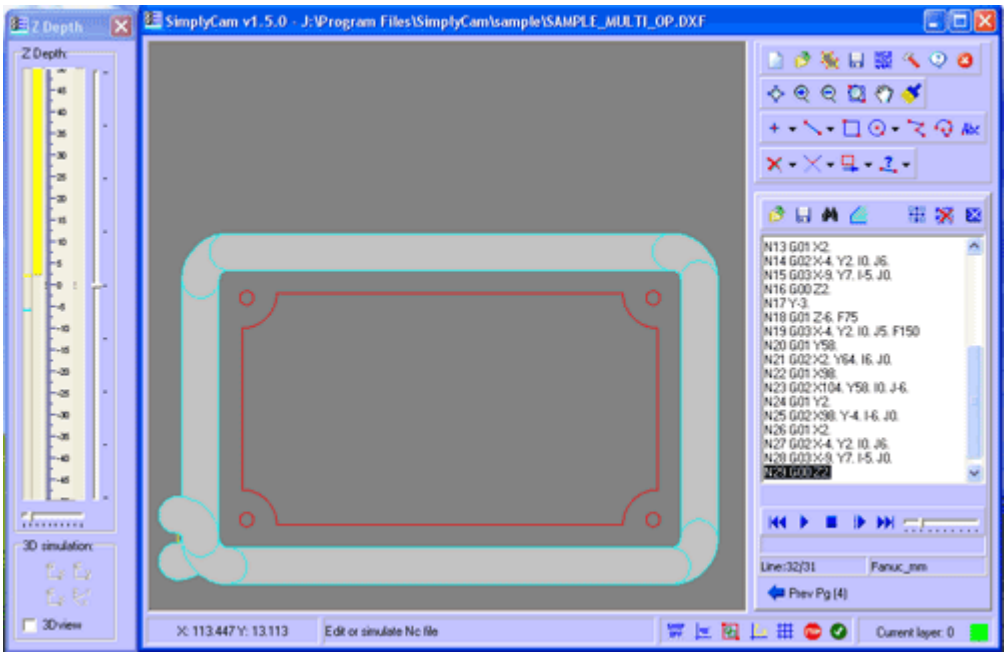


33. Move the slider near to slow position.

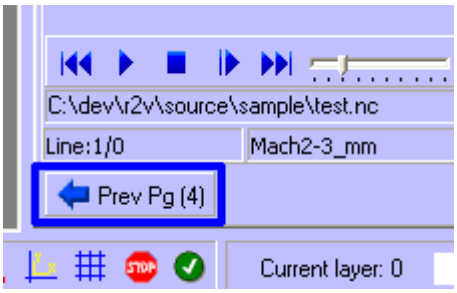


34. Press the "Play" button to simulate the toolpath (Yellow=Rapid, Cyan=Feed) in the graphic area.

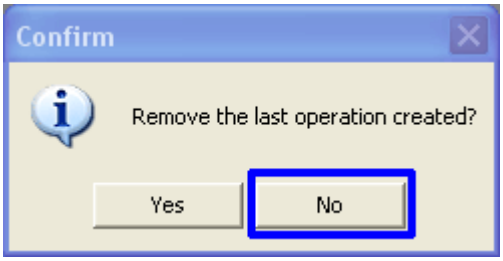
35. The first toolpath (Contour) is simulated.
The "Z Depth" panel indicator reflect the actual Z tool position (Yellow=Rapid, Cyan=Feed).



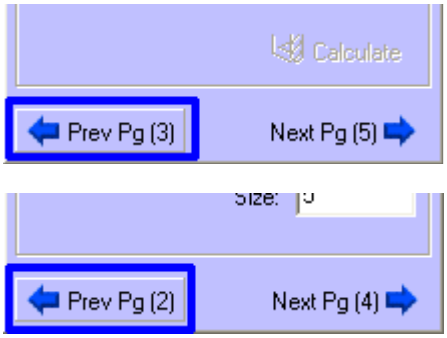
36. Press the "Prev Pg(4)" button.



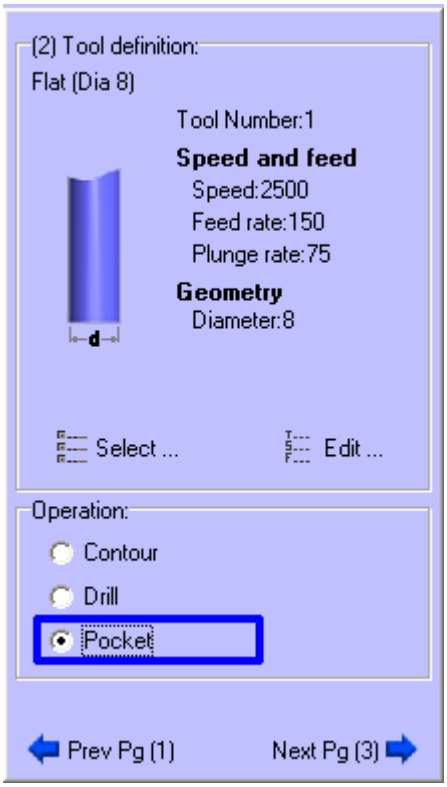
37. "Remove the last operation created?"
No, the contour toolpath is correct, we want to preserve it.



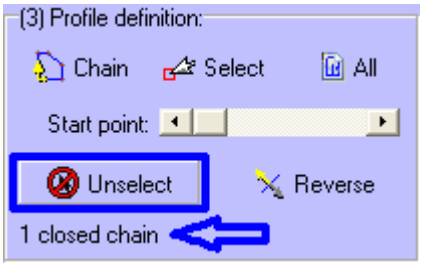
38. Press the "Prev Pg(3)" button and Press the "Prev Pg(2)" button.



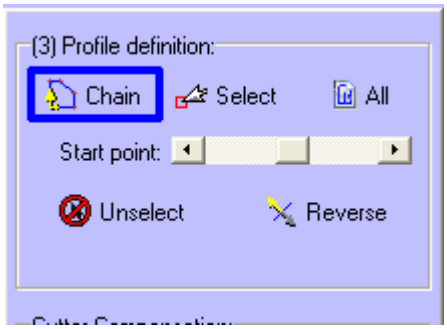
39. In the Operation section, select "Pocket" and press "Next Pg(3)".



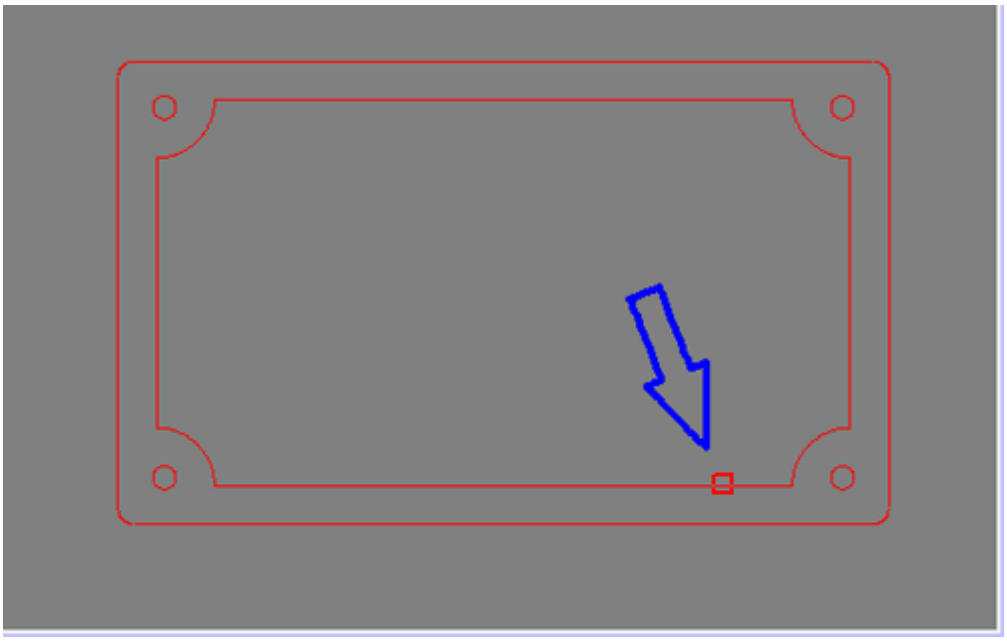
40. The "Profile definition" page appear.
Press the "Unselect" button to remove the last chain.



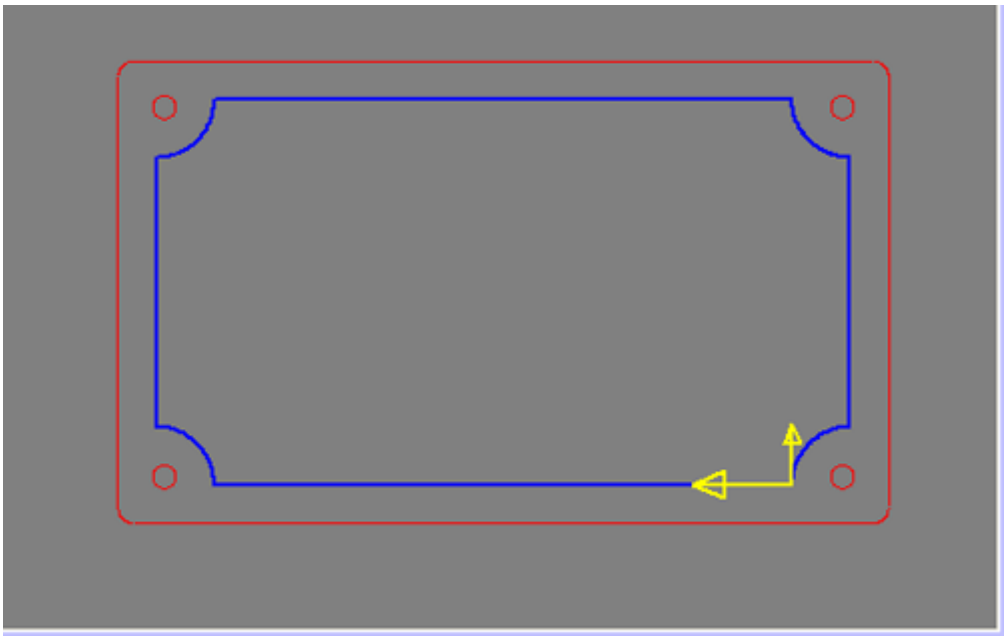
41. Press the "Chain" button.



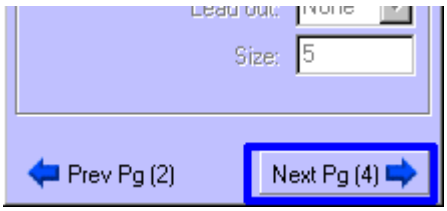
42. Pick the geometry near to start point as indicate by red square.



43. 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.

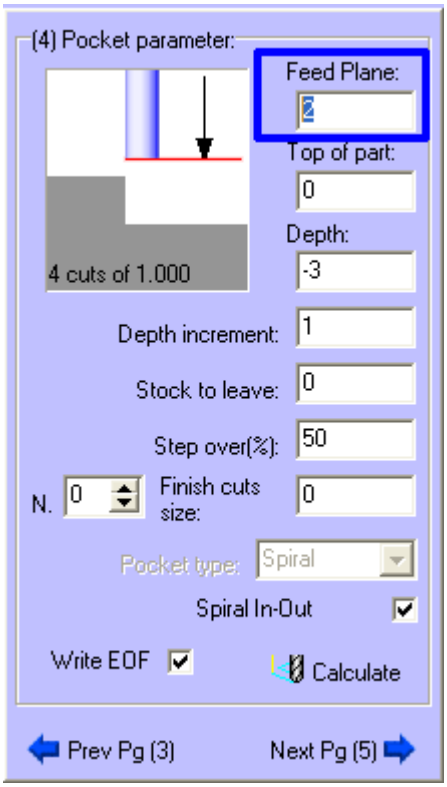


44. Press the "Next Pg(4)" button.

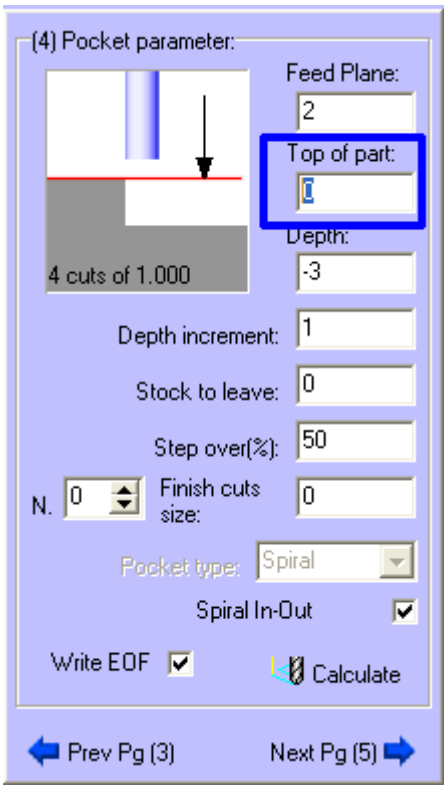


45. The "Pocket parameters" page appears. Set up the following parameters:

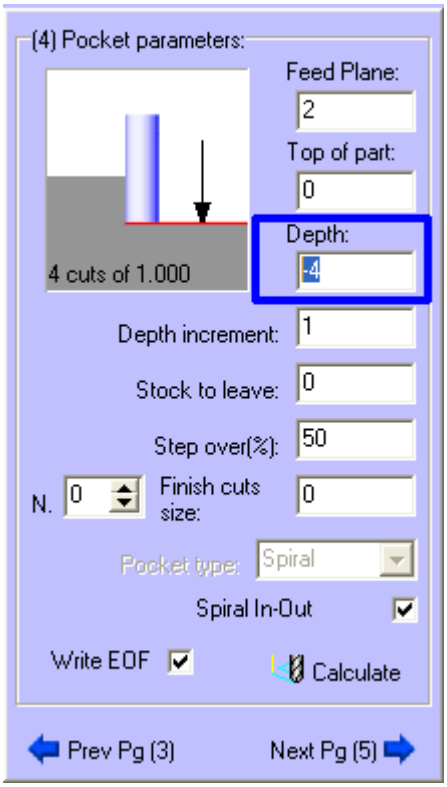
46. **Feed plane:** set the height that the tool rapids to (G0) before changing to the feed rate (G1) to enter in the part (absolute).



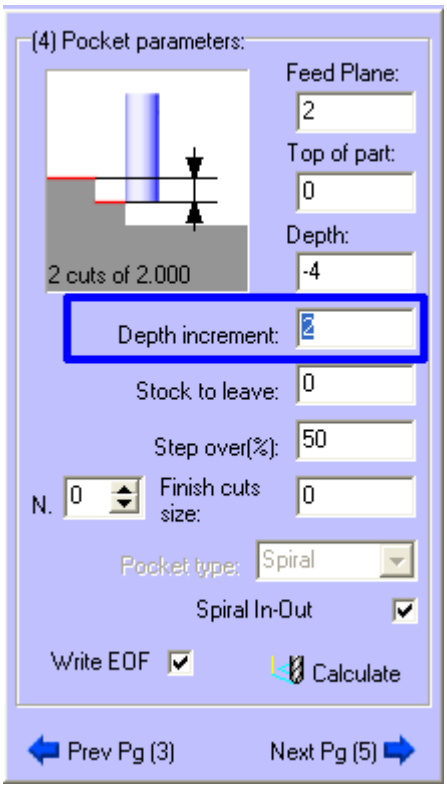
47. **Top of part:** set the height of the piece in the Z axis (absolute).



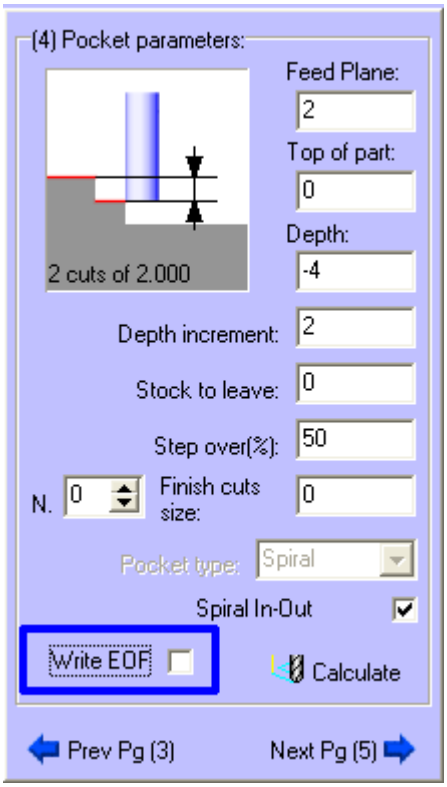
48. **Depth:** set the final machining depth (absolute).



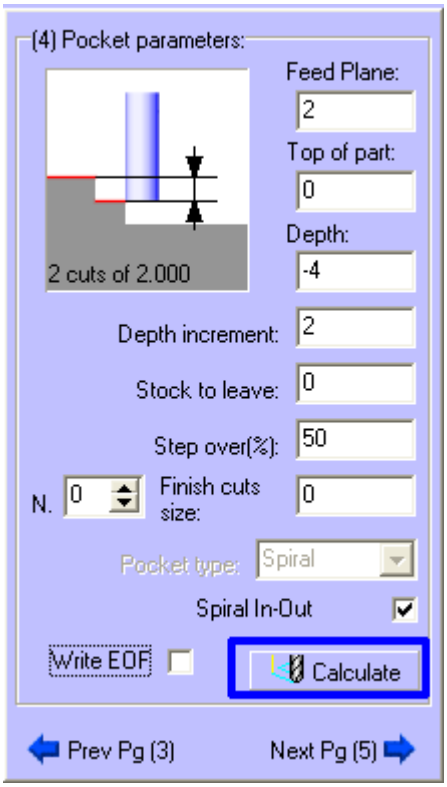
49. **Depth increment:** set the maximum amount of material to remove for each Z cut.



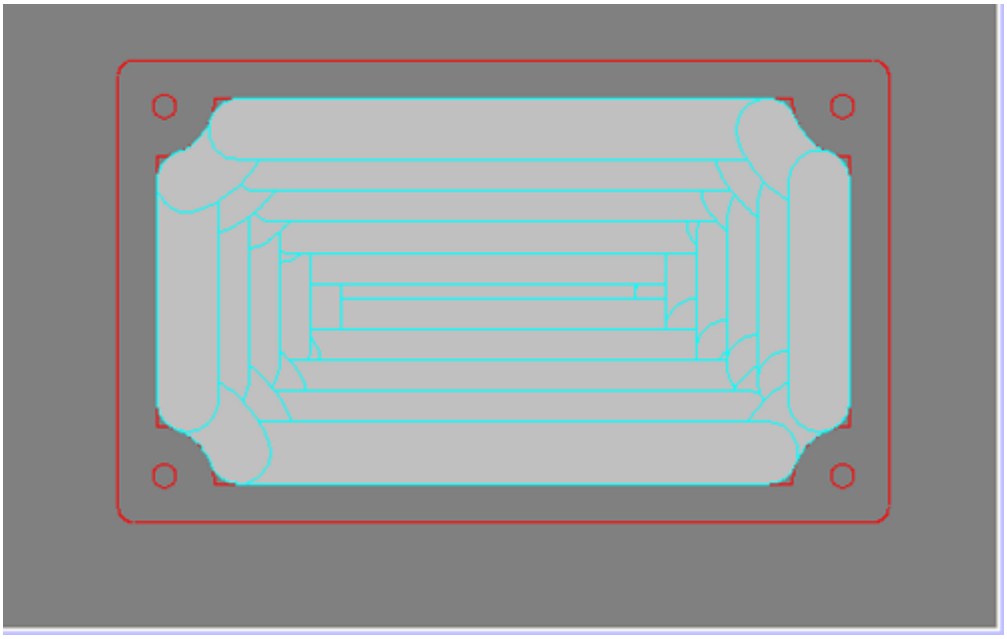
50. **Write EOF:** Turn "Off" this parameter for not write in the Nc file the "End Of File" section (typically M2 or M30) .



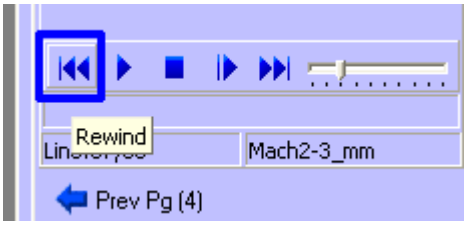
51. Press the "Calculate" button to machining the chained geometry with the cutting parameters.



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

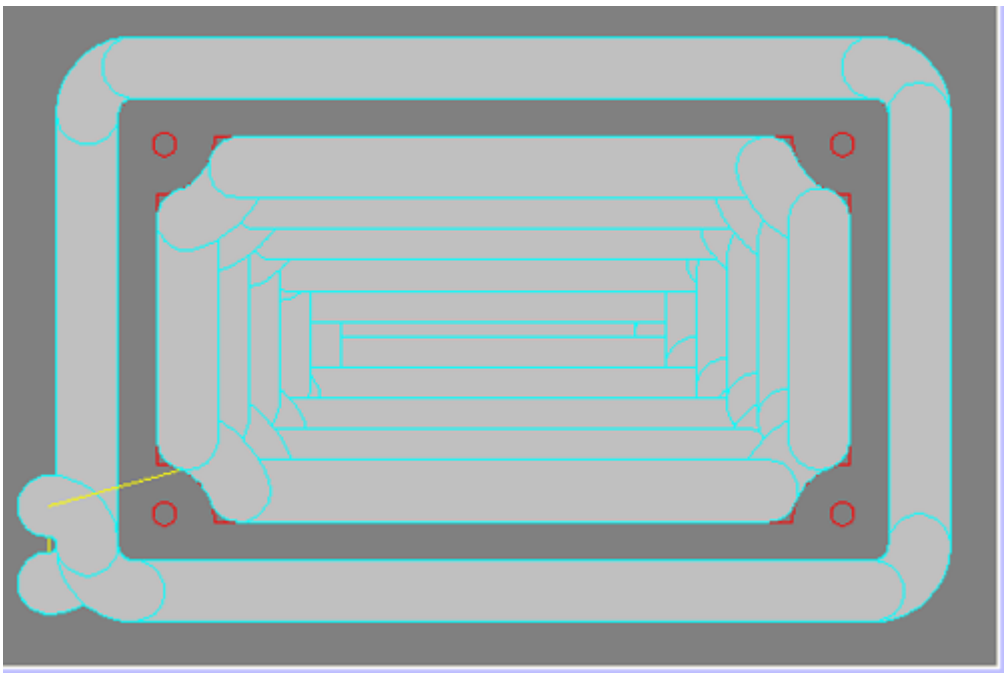


53. Press the "Rewind" button.

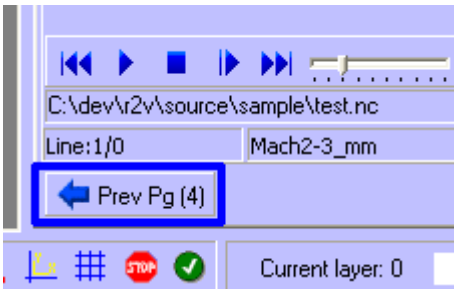


54. Press the "Play" button to simulate the toolpath (Yellow=Rapid, Cyan=Feed) in the graphic area.

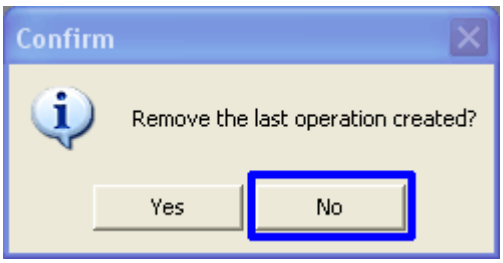
55. In the graphic area the first Contour toolpath and the second Pocket toolpath are simulated.



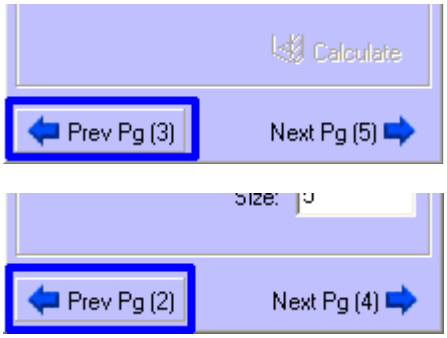
56. Press the "Prev Pg(4)" button.



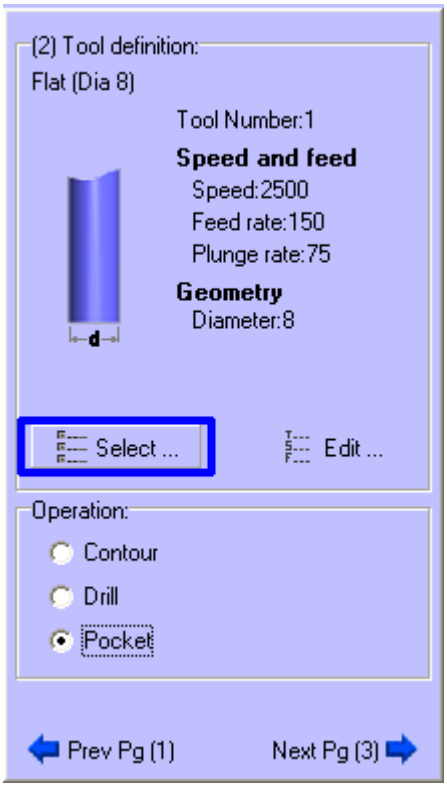
57. "Remove the last operation created?"
No, the pocket toolpath is correct, we want to preserve it.



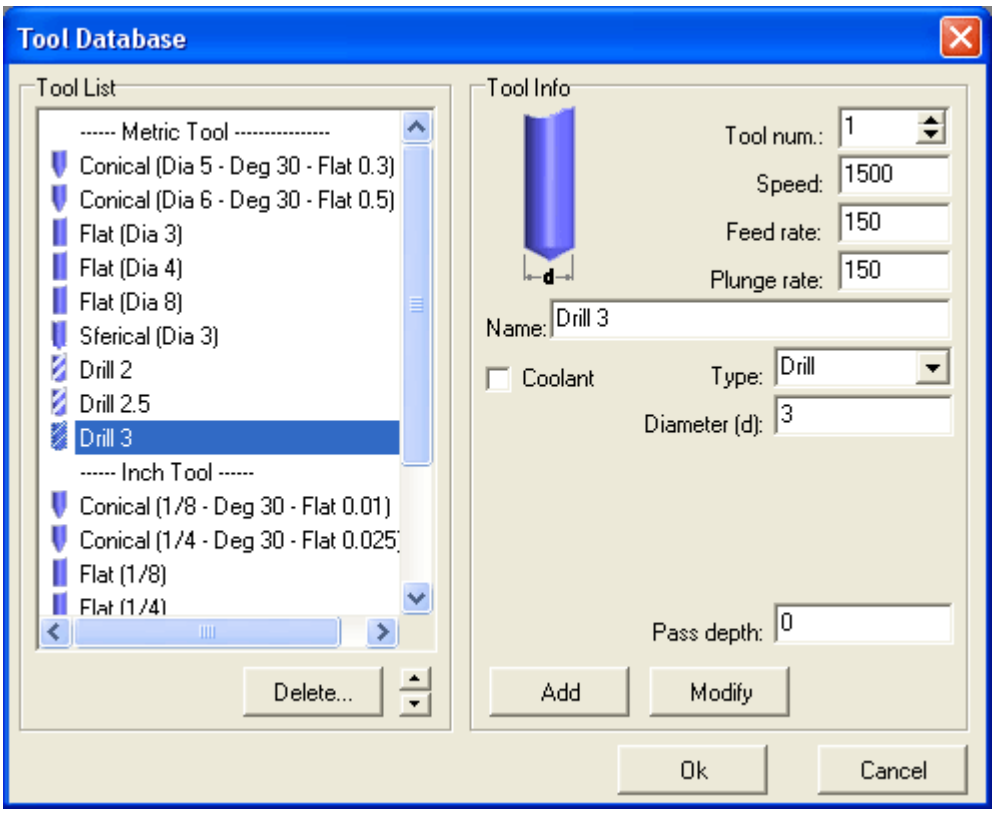
58. Press the "Prev Pg(3)" button and Press the "Prev Pg(2)" button.



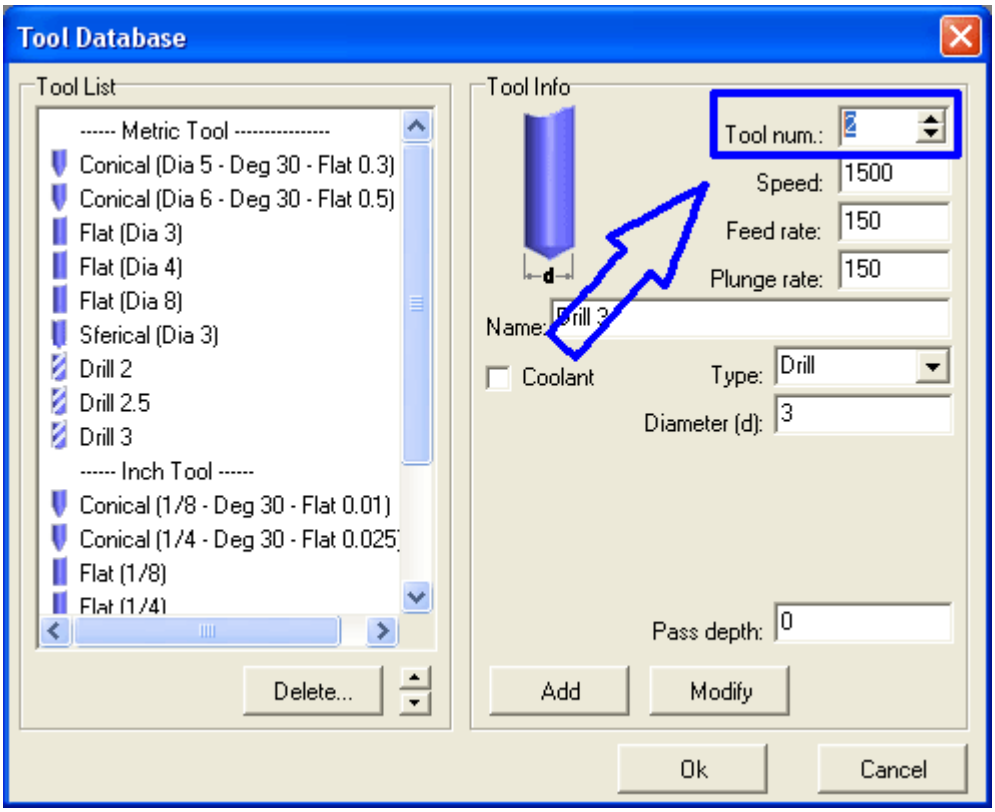
59. In Tool section press "Select" button to retrieve a tool from a tool library or create a new tool.



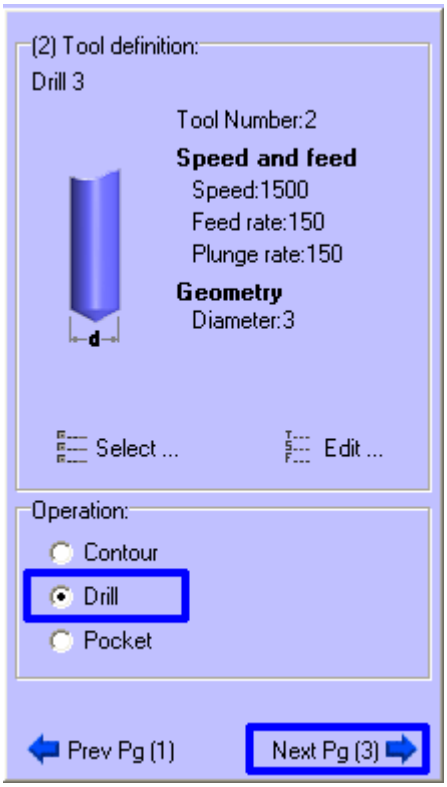
60. Click in the Tool List to select the Drill tool with **diameter 3**. Set feeds and speed of the tool. Press the "OK" button.



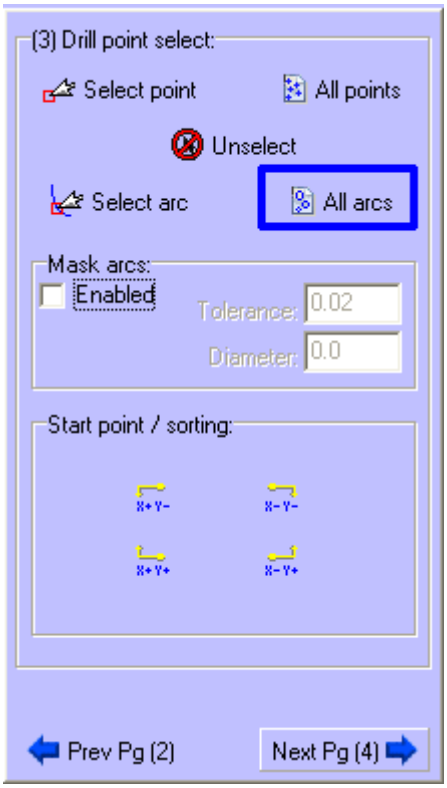
61. Change the number of tool in **2**.
    This new number, different from previous, force SimplyCam to generate in next operatation the tool change.
    Press the "OK" button.



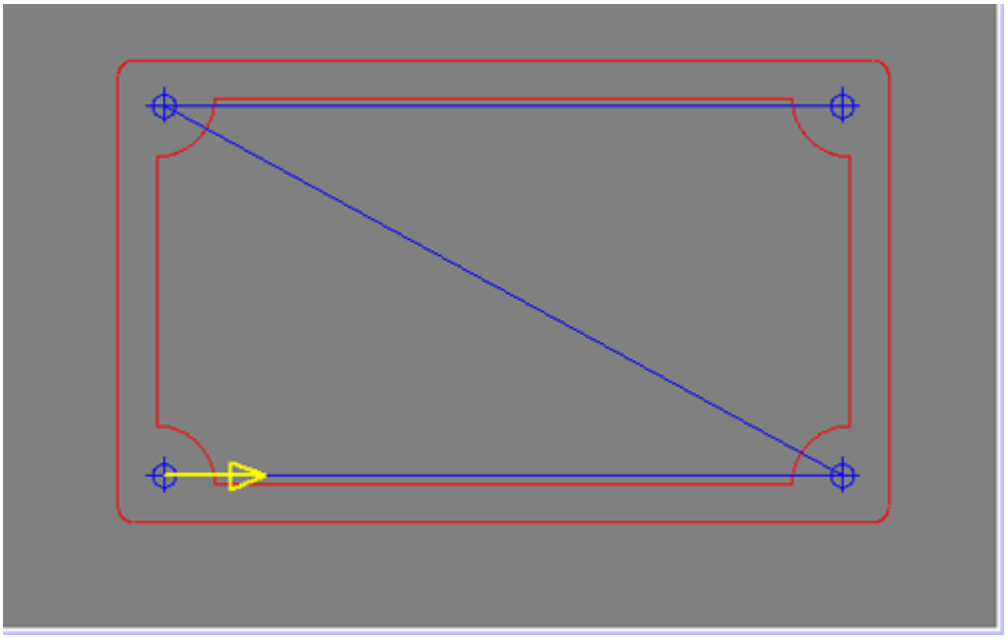
62. In the Operation section, select "Drill" and press "Next Pg(3)".



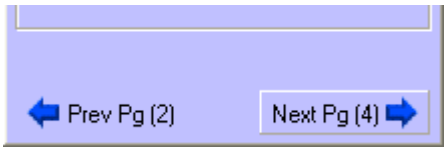
63. The "Drill points select" page appear.
Press the "All arcs" button to select all the closed arcs in geometry.



64. The yellow arrow appears to indicate the start point and the direction of drilling.



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



66. The "Drill parameters" page appears. Set up the following parameters:
Note: the sequence of the drilling cycle is listed below:

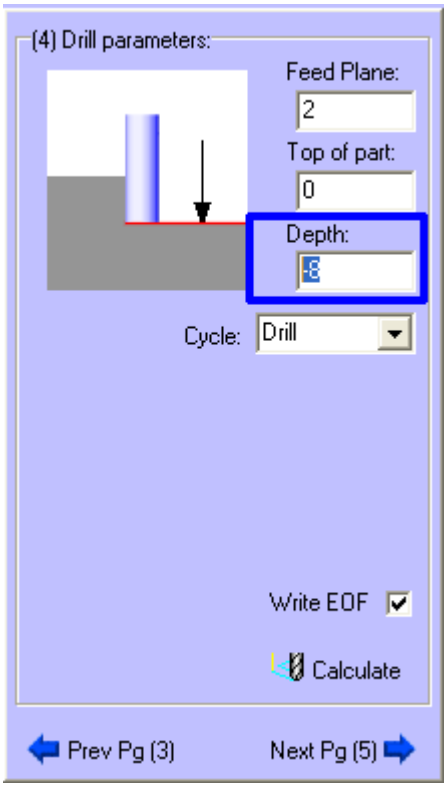
Drill cycle:
- Rapid to the hole center with XY axis
- Rapid Z to feed plane (Reference height)
- Feed Z to the Z depth
- Rapidly retract Z to feed plane (reference height)
- Rapid to another hole center and repeat the sequence

Peck Drill cycle:
- Rapid to the hole center with XY axis
- Rapid Z to feed plane (Reference height)
- Feed Z down with one depth increment
- Rapidly retract Z to feed plane
- Rapid Z down to clearance up from the previous drilled depth
- Feed Z down with one depth increment
- Repeat the last 2 step until the hole bottom is reached
- Rapidly retract Z to feed plane (reference height)
- Rapid to another hole center and repeat the sequence

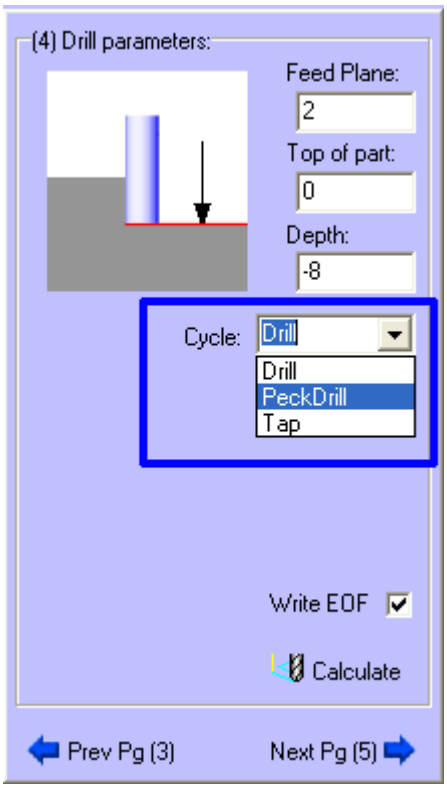
67. **Feed plane:** set the height that the tool rapids to (G0) before changing to the feed rate (G1) to enter in the part (absolute).

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

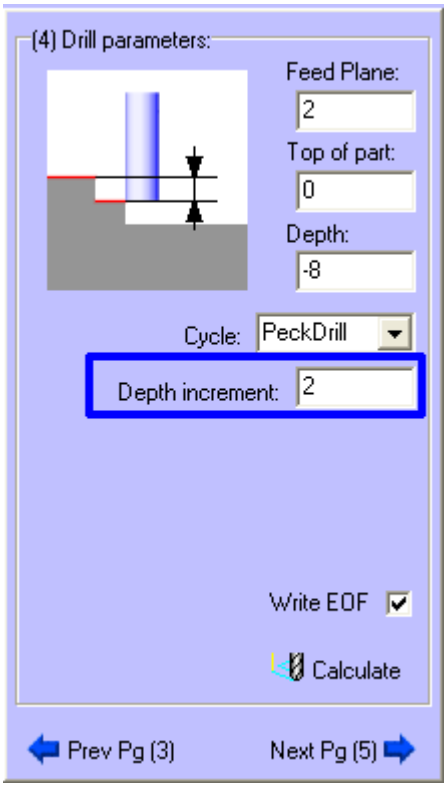
69. **Depth:** set the final depth of drilling operation (absolute).



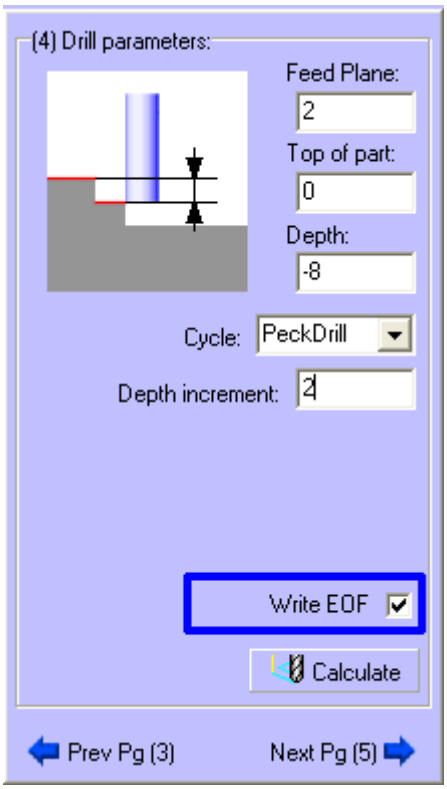
70. **Cycle:** select the drilling cycle.



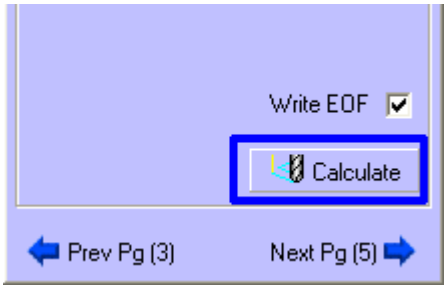
71. **Depth increment:** set the amount of material to remove for each Z cut.



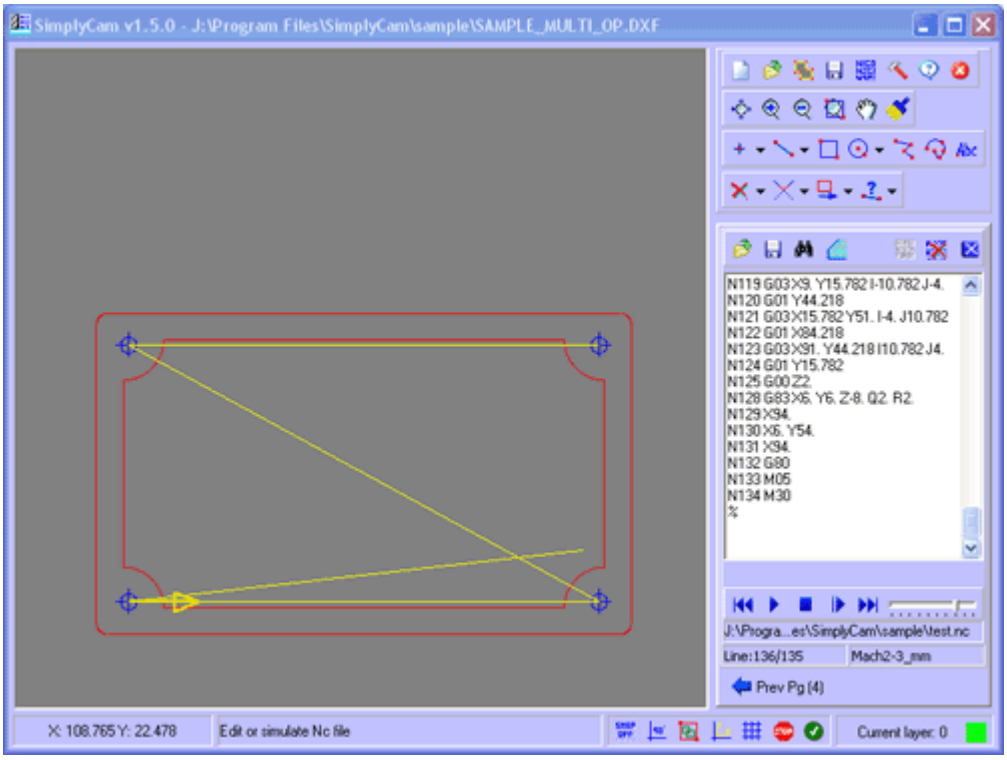
72. **Write EOF:** Turn **"On"** this parameter for write in the Nc file the "End Of File" section (typically M2 or M30). This is the last operation of the example.



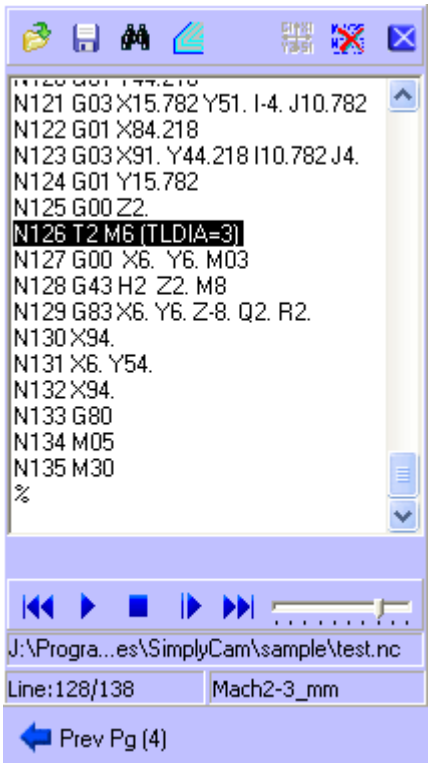
73. Press the "Calculate" button to drilling the definite points with the cutting parameters.



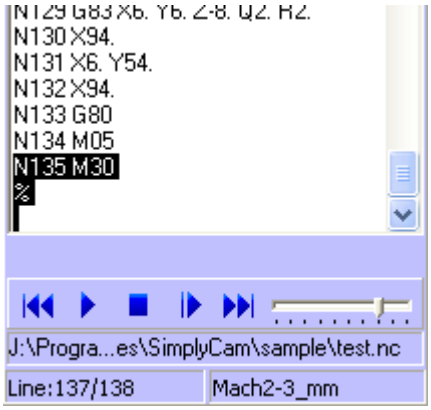
74. The "Nc File" page is displayed and in the graphic area the drillng toolpath is simulated.



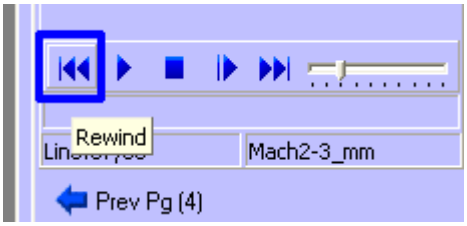
75. The Tool-change section is added in Nc file



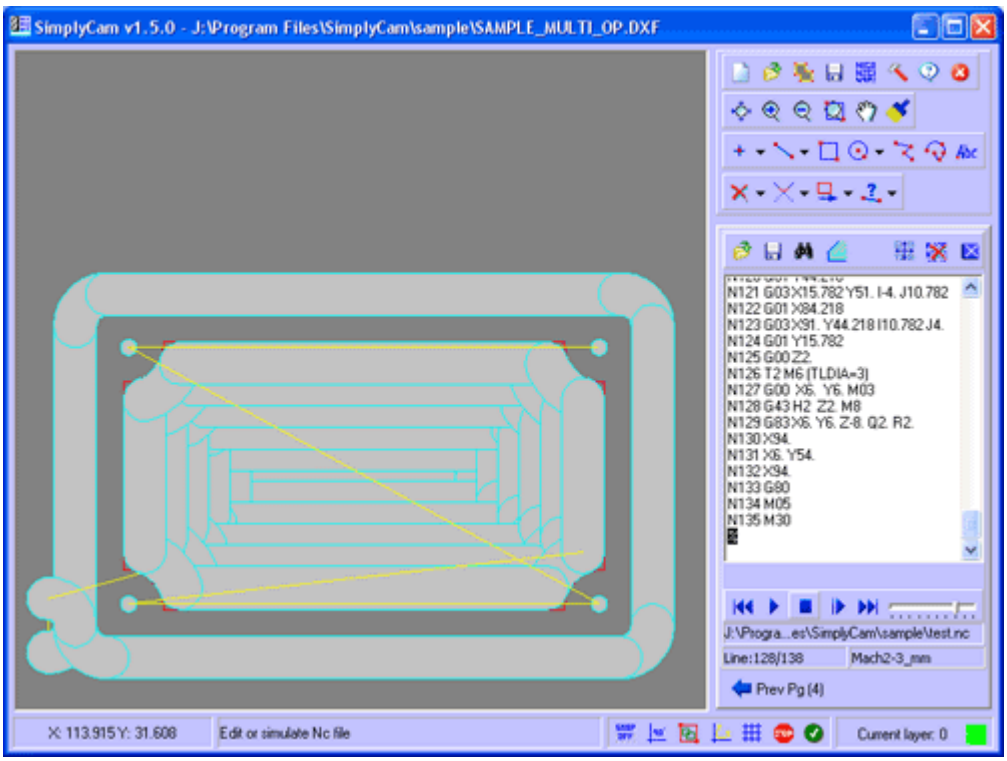
76. And the EOF (end of file) sequence is added at end of the Nc file.



77. Press the "Rewind" button.



78. Press the "Play" button to simulate the contour, pocket and drill toolpaths in the graphic area.



79. You have successfully created the multi-toolpath example 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.