

# PROCESS DOCUMENT

PROJECT: TITAN-139M

---

## OVERVIEW:

This outline will detail each step of the manufacturing process for the TITAN-139M first series part. Begin by creating the part design in CAD, programming it in CAM, and then machining it to specification on a CNC machine tool. Resources for this project, which include the dimensioned print, setup sheet, tool library, and other files, are available in the [TITAN 5-AXIS Learning Series](#) on the [TITANS of CNC: Academy](#).

## STEP 1: DESIGN IN CAD

1. Navigate to [academy.titansofcnc.com](https://academy.titansofcnc.com) in your browser
2. Locate and select the TITAN 5-AXIS Learning Series on the homepage
3. Select the TITAN-139M banner from the list
4. Choose the TITAN-139M CAD tutorial for either [Solidworks](#) or [Fusion 360](#)
5. Download the Related Files for the CAD module
  - a. Dimensioned Print
  - b. Eagle (.DXF File - Fusion 360 or Solidworks)
  - c. Logo (.DXF File - Fusion 360 or Solidworks)
6. Launch Solidworks or Fusion 360
7. Follow along with the tutorial in your software to design the part

## STEP 2: PROGRAM IN CAM

1. Choose the TITAN-139M CAM tutorial for either [Mastercam](#) or [Fusion 360](#)
2. Download the Related Files for the CAM module
  - a. Dimensioned Print
  - b. Setup Sheet
  - c. 5-Axis Tool Library (download and replace this file for each part in this series)
  - d. OP1 Vise model (Parasolid)
3. Launch Mastercam or Fusion 360
4. Follow along with the tutorial in your software to program the part
5. Simulate OP1 and OP2 to check for machine crashes and/or part gouges
6. For Mastercam users:

- a. Verify every toolpath in OP1 is using the correct WCS.
- b. Verify every plane used in OP1 is set to Manual and "0" for the Work Offset. This will ensure only G54 is output for every plane.
- c. Select every toolpath in OP1 and right-click in the toolpath menu. Select "edit selected operations" then select "change NC file name..." A new dialog box will appear, in the dialog box type TITAN 139M OP1 then select "Ok". This changes the file name for all the toolpaths in OP1 to the same name.
- d. Select every toolpath in OP1 and right-click in the toolpath menu. Select "edit selected operations" then select "change program number..." A new dialog box will appear, type 1001 then select "Ok". This changes the program number for all the toolpaths in OP1 to the same number.
- e. With every toolpath in OP1 selected, post the code.
- f. Verify every toolpath in OP2 is using the correct WCS.
- g. Verify every plane used in OP2 is set to Manual and "0" for the Work offset. This will ensure only G54 is output for every plane.
- h. Select every toolpath in OP2 and right-click in the toolpath menu. Select "edit selected operations" then select "change NC file name..." A new dialog box will appear, type TITAN 139M OP2 then select "Ok". This changes the file name for all the toolpaths in OP2 to the same name.
- i. Select every toolpath in OP2 and right-click in the toolpath menu. Select "edit selected operations" then select "change program number..." A new dialog box will appear, type 1002 then select "Ok". This changes the program number for all the toolpaths in OP2 to the same number.
- j. With every toolpath in OP2 still selected, post the code.
- k. The code for OP1 and OP2 are now ready to be transferred to the machine.

### STEP 3: FIRST OPERATION CNC

1. Choose the TITAN-139M CNC tutorial for either the [DN Solutions DVF5000](#) or the [Haas UMC 750](#)
2. Download and print the Related Files for the CNC module
  - a. Process Document
  - b. Dimensioned Print
  - c. Setup Sheet
  - d. Final Inspection Report
3. Watch the tutorial and follow along to setup and run the part on the machine
4. Setup Vise assembly on machine table
  - a. Install Vise assembly to machine table. It is extremely important to use the exact fixturing setup as you used in the CAM simulation to prevent a crash. If you have a different vise and riser than what is used in the CAM tutorial, please load the correct models for your assembly in the CAM file and simulate the program with your setup.

- b. Before teaching any work offsets, ensure the table is at B0 on the machine position
  - c. Ensure the G54 B-Axis work offset is set to 0. In other words, ensure there is no value in the offset.
  - d. Indicate Vise true with the X-Axis using an indicator or probe
  - e. After the vise is indicated true, set the G54 C-Axis work offset
5. Build tools per the Setup Sheet and CAM program.
  - a. It is critical that the holders and stickout length of the tools match the Setup Sheet and CAM program. If you must deviate from what is provided, then you need to create the new holder in the CAM file and simulate the program with that holder to ensure it does not crash.
6. Install tools in Machine and teach the length of each tool
7. Verify stock material is correct size per the Setup Sheet
8. Install material in the center of the vise per the Setup Sheet and torque to proper specifications
9. Probe the stock to set G54 X0 and Y0 to the center of the material
10. Probe the top of the stock to set G54 Z0 to the top of the material
  - a. Go to the offset page and highlight G54 Z-Axis offset
  - b. Shift the Z0 negative .020" per the Setup Sheet
11. Setup a Vise Stop on either side of the vise
12. Remove material from the vise
13. Install both OP1 and OP2 programs in the Machine
14. Select OP1 program as the main program to run
15. Dry-Run the program without material in the vise to ensure there are no crashes
16. Re-install the material in the vise, locating it against the Vise Stop
17. Take the machine out of Dry-Run mode
18. Run the program
  - a. Run the first part with the rapid override turned down to the lowest setting
19. After program is complete, verify the dimensions using the proper inspection equipment
  - a. If necessary, make adjustments to the tool lengths using the tool wear offset
20. If you are making multiple parts, remove the part from the vise and install the second part against the Vise Stop and run the program. If not, move onto OP2.

### **STEP 3: SECOND OPERATION CNC**

1. Clean the vise to ensure there are no chips or debris between the jaws
2. Ensure the table is sitting at B0
3. Probe the bottom of the vise to set G54 Z0 per the Setup Sheet.
4. Flip part and clamp in the vise on the finished faces from OP1
5. Indicate one of the finished faces from OP1 using an indicator or probe to ensure the C-Axis is set properly with the finished faces. If not, rotate the C-Axis until it is indicated true and then reset C0 on the G54 work offset

6. Probe the raw stock using a web routine to set X0 Y0 in the center of the part. Note: later you will stop the machine after the facing operation to re-probe the finished surfaces.
7. Setup a Vise Stop on the right side of the vise
8. Remove the part from the vise
9. Select the OP2 program as the main program to run
10. Ensure all of the tools used in this program are installed in the machine and the length for each tool is set
11. Dry-Run the program without material in the vise to ensure there are no crashes
12. Re-install the material in the vise, locating it against the Vise Stop
13. Take the machine out of Dry-Run mode
14. Ensure the Optional Stop button is ON before beginning the program
15. Run the first tool in the program which should be the facemill then stop at M01
  - a. Run the first part with the rapid override turned down to the lowest setting
16. After facemill runs, press reset to stop the program.
17. Re-probe the part to set X0 and Y0 off of the finished faces now that the facemill has removed the "hat" of excess material and the probe can safely reach the finished surfaces from Op1
18. Restart the program at the correct location for the next tool. Alternatively, you can restart at the top of the program.
19. After program is complete, verify the dimensions using the proper inspection equipment
  - a. If necessary, make adjustments to the tool lengths and diameters using the wear offset
20. The part is now complete and the inspection form can be filled out and turned in to the instructor.