

Steps for creating G code in Pro/E:

- 1: Model reference model and work piece as separate parts in Pro/E
- 2: Create new "Manufacturing" file
- 3: Assemble reference model (button on upper right hand column of buttons)
- 4: Assemble work piece (button just below reference model button)
- 5: Create new coordinate system for G54 (On Haas tm1, x + is the long direction of the table pointed right. y+ is the short direction pointed to the back of the machine. z+ is pointed up.)
- 6: Select machine: Resources\Work Center – Accept all defaults and click OK
- 7: Create new operation: Steps\Operation – Select coordinate system and retract plane
- 8: Create NC sequences and tools as needed with tool path buttons in the tool bar
- 9: Play path for entire operation – Right click on operation name in model tree, select "play path"
- 10: Save .ncl file from "play path" control window (Make sure this is the play path for entire operation not a single sequence)
- 11: Create G code: Tools\CL Data\Post Process – Select the first Post Processor from the list, UNCX01, and give the program a 4 digit number when prompted
- 12: Open .tap file in notepad, and G54 to the top of the code

Standard tools:

- 1: Indicator
- 2: 3/4" end mill
- 3: 1/2" end mill
- 4: 1/4" end mill
- 5: 3/16" end mill
- 6: 1/8" end mill
- 7: Center Drill
- 8: Open
- 9: Open
- 10: Open