



I bought Pro/NC... Now What?!?

Todd Liebenow

Coldfire Enterprises



www.coldfire-e.com

• Copyright © 2007 Coldfire Enterprises

Agenda

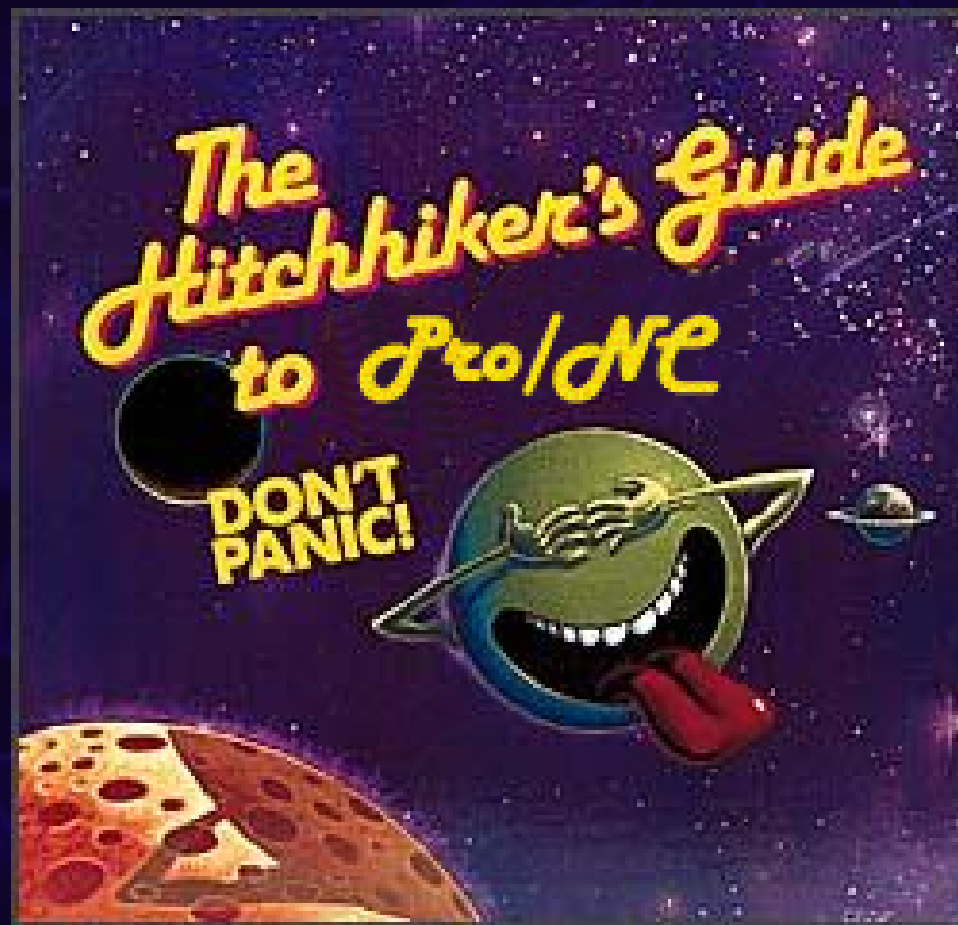
- 3 steps
 - Foundation
 - Workflow
 - Documentation
- Supplemental information
- Q & A (time permitting)

First, CONGRATULATIONS!

The reasons you bought Pro/Engineer, and Pro/NC probably include:

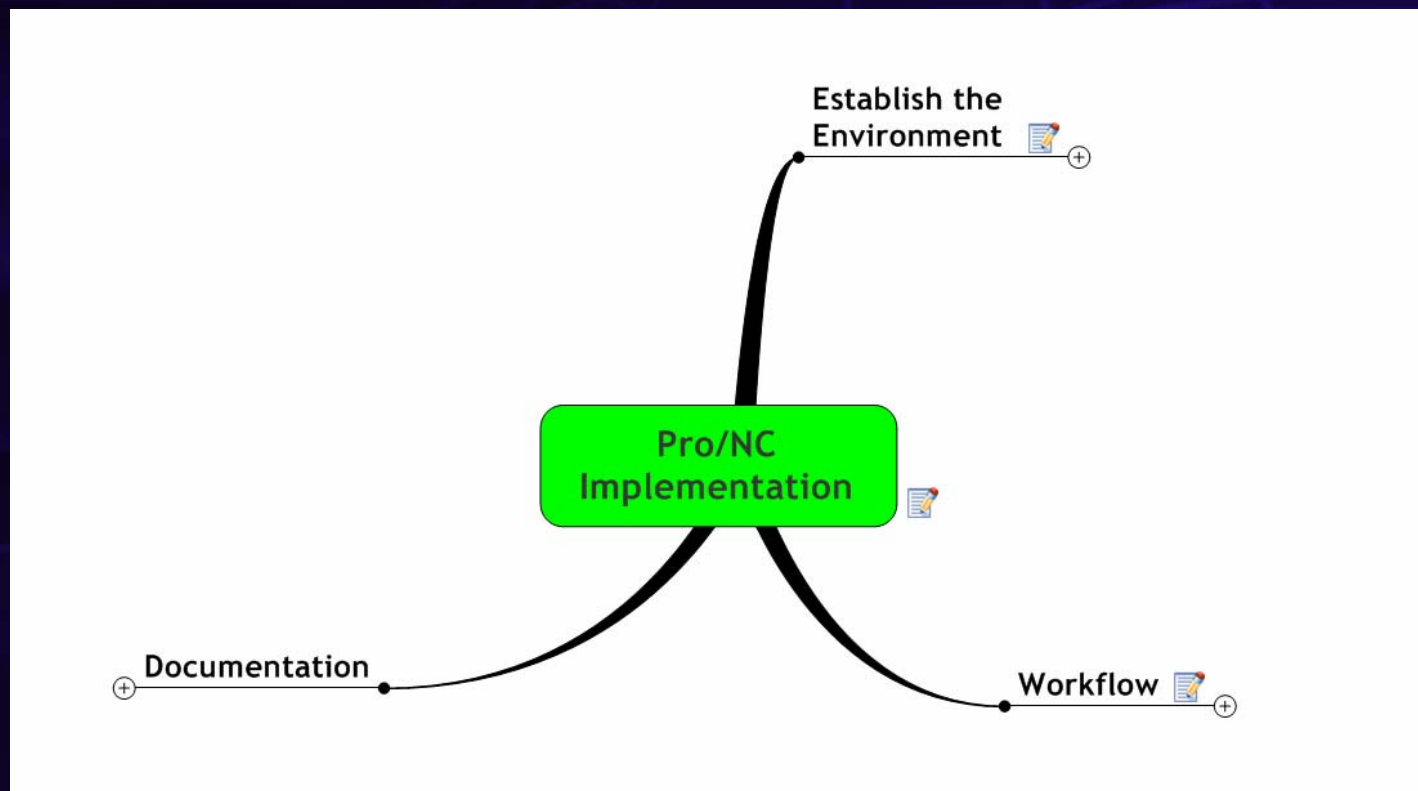
- *Fully integrated and associative Machining applications*
- *NC Programming Functionality*
 - Mill
 - Turn
 - Wire EDM
 - Post-processing
 - Verification
 - Process planning and documentation
- *Robust automated toolpath creation*
- *The most versatile NC programming solution available*
 - Machines anything
- *Enables truly concurrent design through manufacturing*

Next, Don't PANIC!



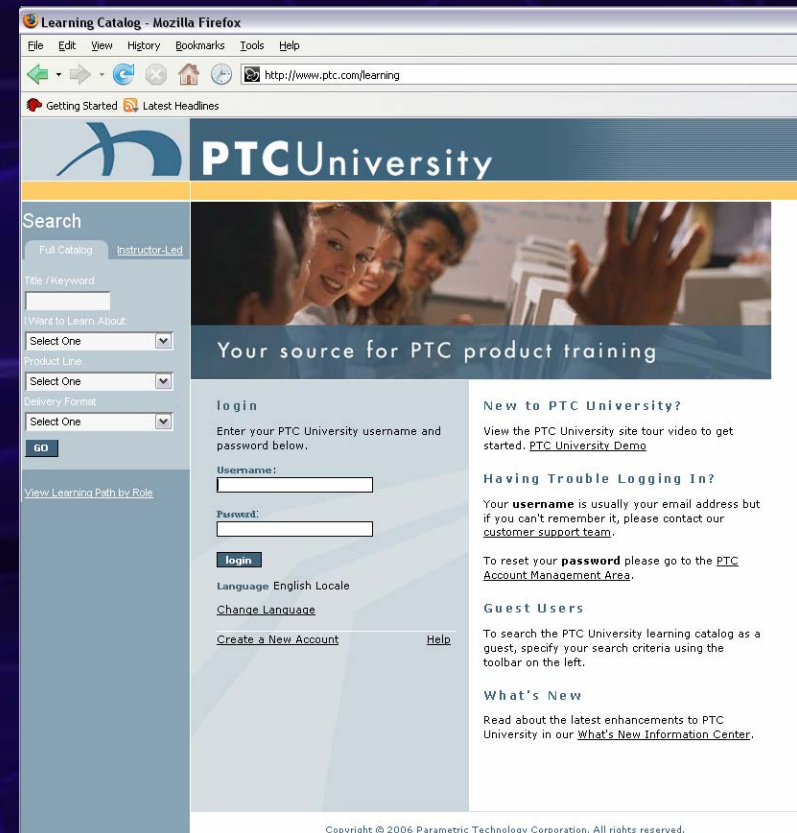
The Plan

- 3 phase approach to complete implementation



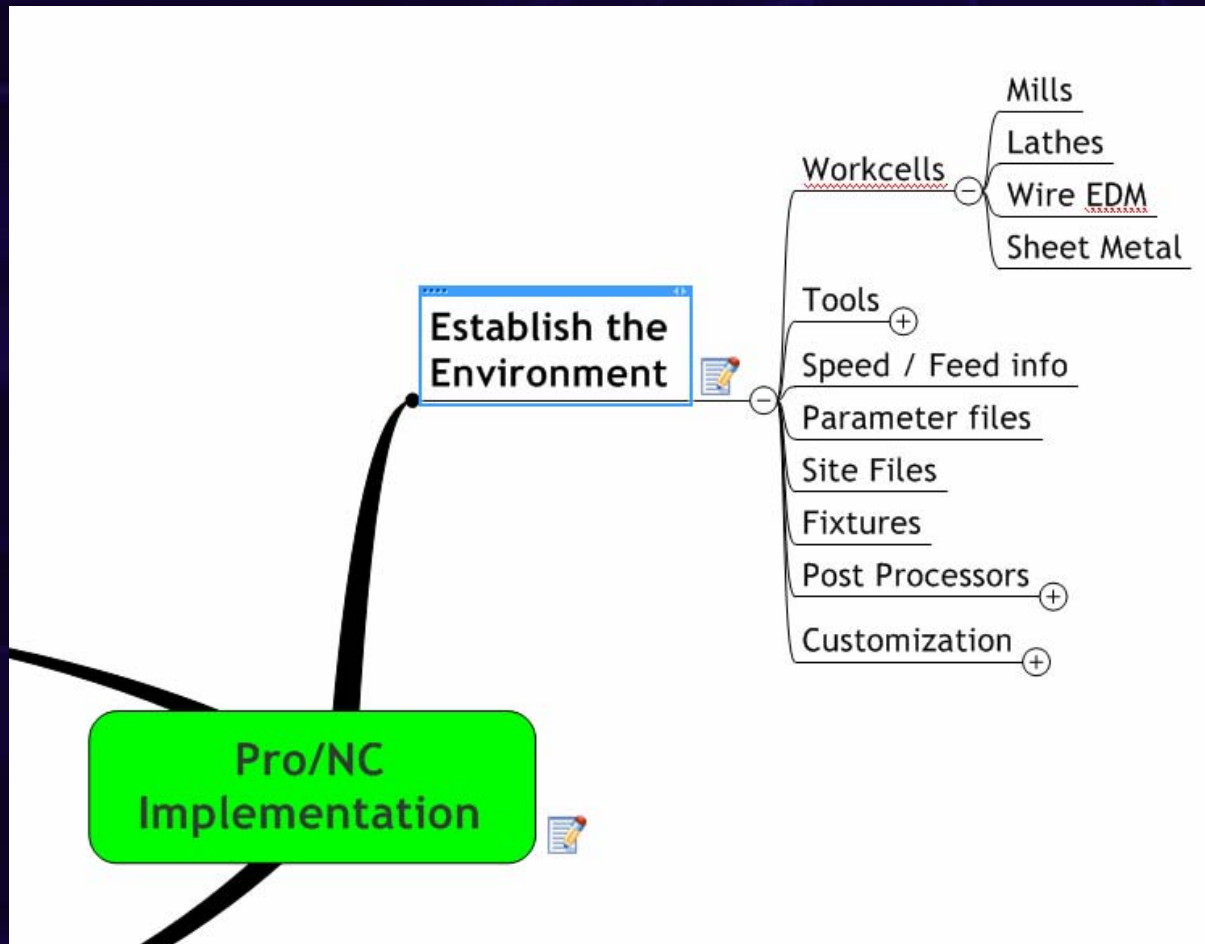
Training – It's not optional!

- Some choices include:
 - PTC University
 - www.ptc.com/learning
 - Certified Training Partners
 - www.ptc.com/partners/training/index.htm



Establish the Environment

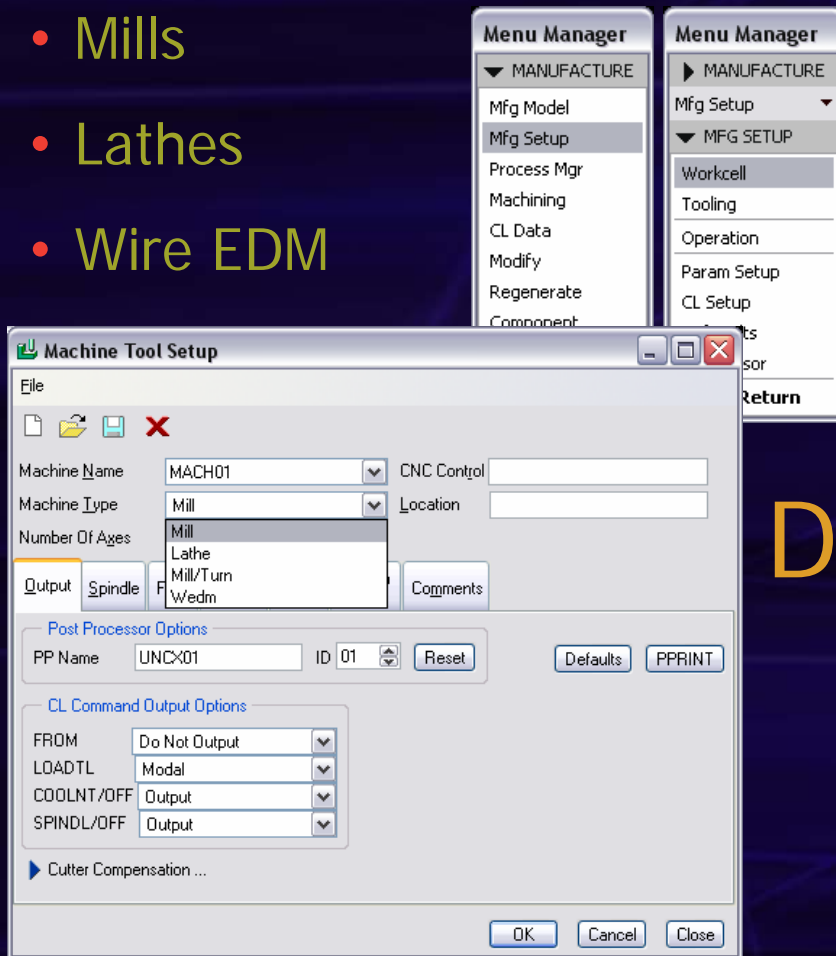
- Step 1: Set up the foundation



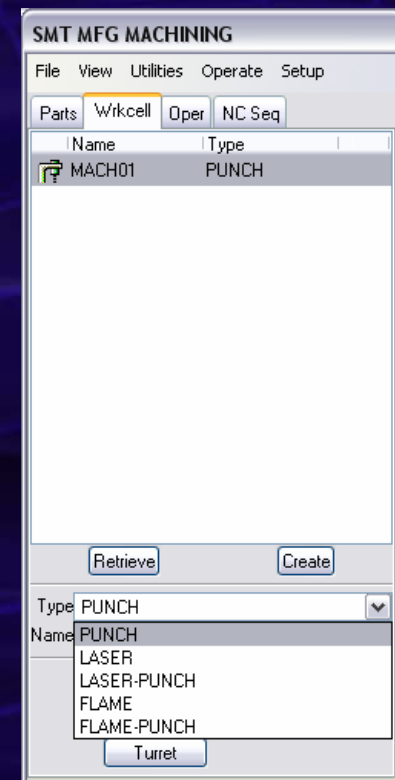
Workcells

Config.pro option: `pro_mf_workcell_dir <drive?:\path\to\workcell\directory>`

- Mills
- Lathes
- Wire EDM



- Sheet Metal

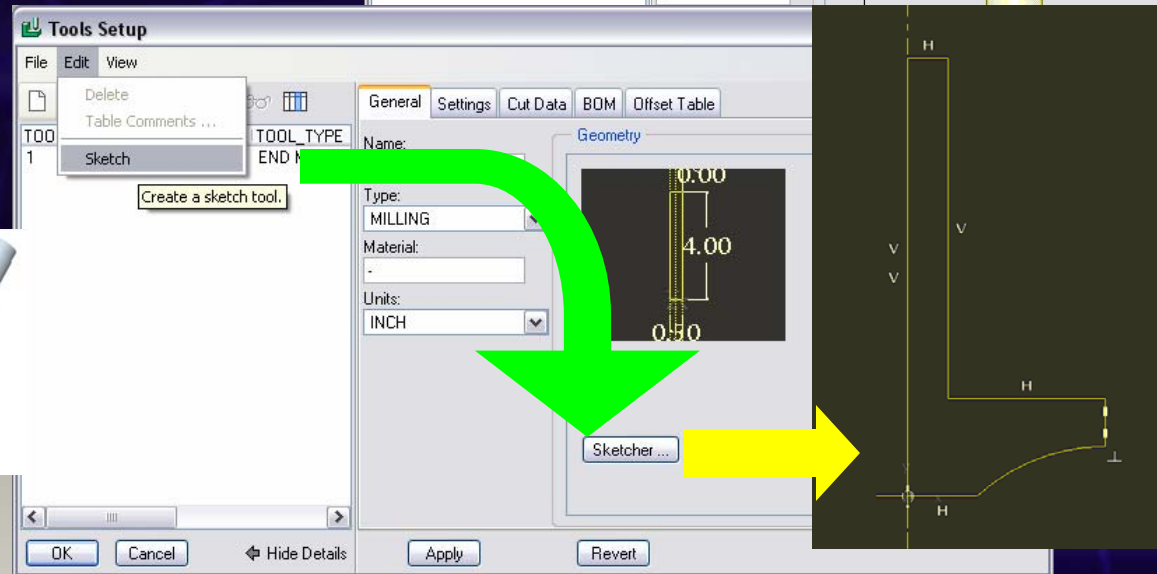
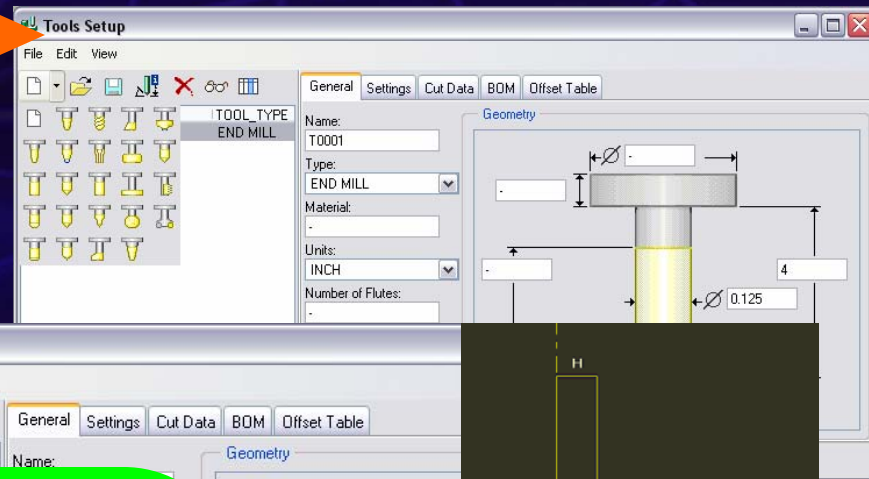


Don't forget
to **SAVE**
them!

Tools

Config.pro option: `pro_mf_tprm_dir <drive?:\path\to\tooling\directory>`

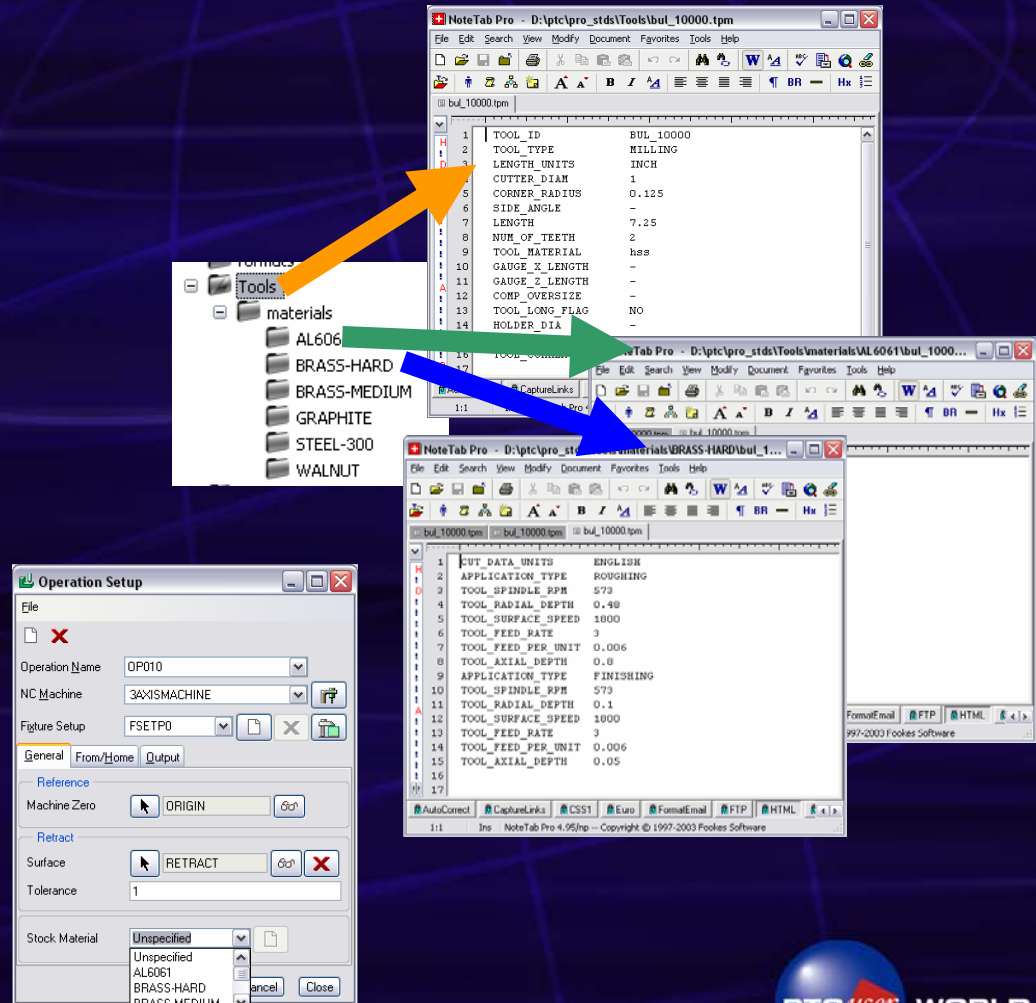
- Parameter based
- Sketched
- Solid



Speed / Feed info

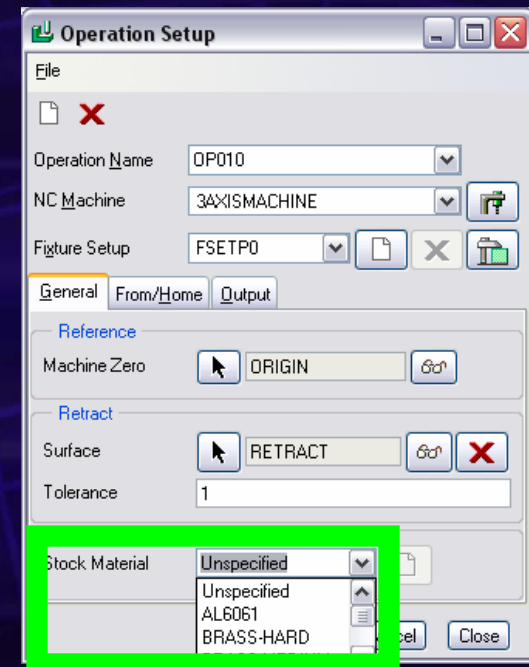
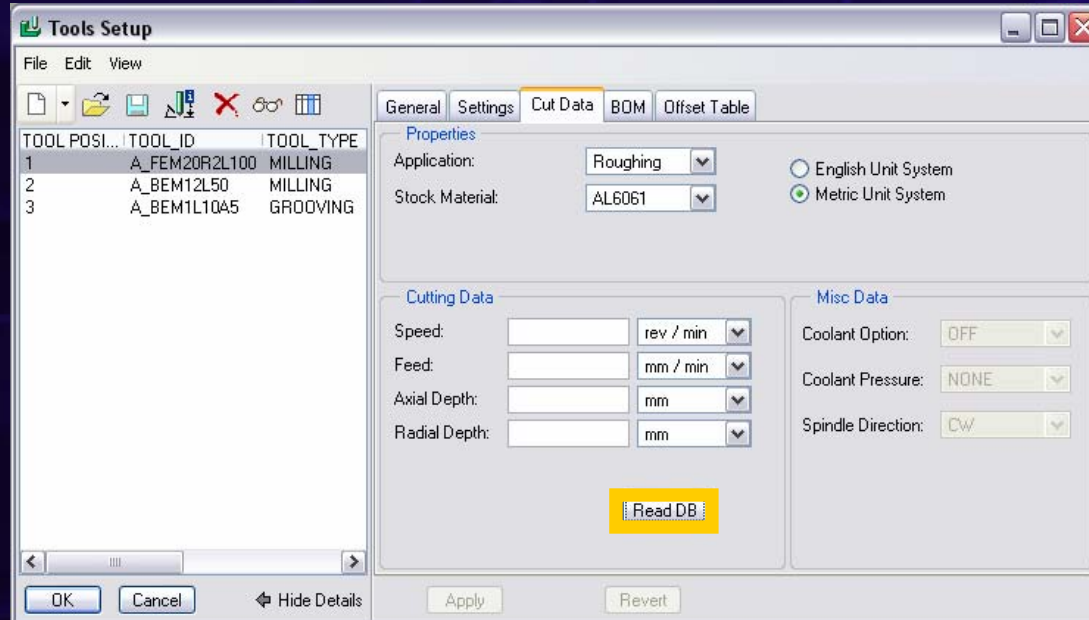
Config.pro option: `pro_mf_tprm_dir <drive?:\path\to\tooling\directory>`

- Pre Wildfire 3.0:
 - Use a directory structure / flat ASCII files (*.tpm)
 - *.tpm for tool geom
 - *.tpm for speed / feed
 - One folder / material



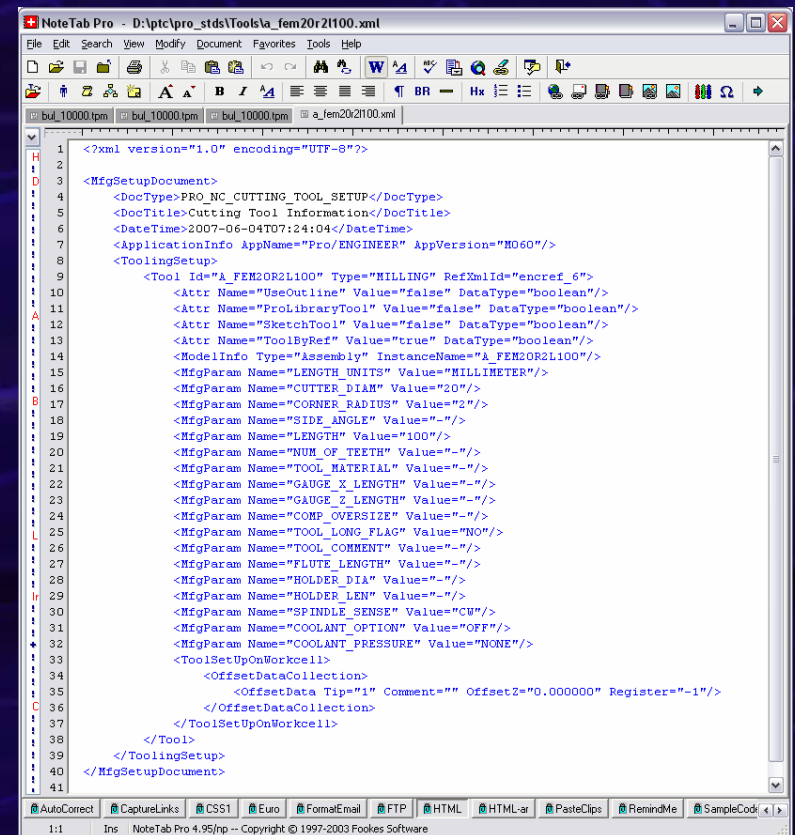
Speed / Feed info (con't)

- Select material in Operation drop down menu, then...
- Read DB option in tooling dialog
- One method for entering S / F data



Speed / Feed info (con't)

- Wildfire 3.0 and beyond:
 - Store the machining tool parameter file in XML format, simplifying the interface to the external tool database management system. This single file, combining tool geometry and cutting technology, can also be easily managed by PDMLink.



The screenshot shows the NoteTab Pro application window with the file `D:\ptc\pro_std\Tools\fa_fem20r2l100.xml` open. The XML content is as follows:

```
<?xml version="1.0" encoding="UTF-8"?>
<MfgSetupDocument>
  <DocType>PRO_NC_CUTTING_TOOL_SETUP</DocType>
  <DocTitle>Cutting Tool Information</DocTitle>
  <DateTime>2007-06-04T07:24:04</DateTime>
  <ApplicationInfo AppName="Pro/ENGINEER" AppVersion="M060"/>
  <ToolingSetup>
    <Tool Id="A_FEM20R2L100" Type="MILLING" RefXid="encref_6">
      <Attr Name="UseOutline" Value="false" DataType="boolean"/>
      <Attr Name="ProLibraryTool" Value="false" DataType="boolean"/>
      <Attr Name="SketchTool" Value="false" DataType="boolean"/>
      <Attr Name="ToolByRef" Value="true" DataType="boolean"/>
      <ModelInfo Type="Assembly" InstanceName="A_FEM20R2L100"/>
      <MfgParam Name="LENGTH_UNITS" Value="MILLIMETER"/>
      <MfgParam Name="CUTTER_DIAM" Value="20"/>
      <MfgParam Name="CORNER_RADIUS" Value="2"/>
      <MfgParam Name="SIDE_ANGLE" Value="-"/>
      <MfgParam Name="LENGTH" Value="100"/>
      <MfgParam Name="NUM_OF_TEETH" Value="-"/>
      <MfgParam Name="TOOL_MATERIAL" Value="-"/>
      <MfgParam Name="GAUGE_X_LENGTH" Value="-"/>
      <MfgParam Name="GAUGE_Z_LENGTH" Value="-"/>
      <MfgParam Name="COMP_OVERSIZE" Value="-"/>
      <MfgParam Name="TOOL_LONG_FLAG" Value="NO"/>
      <MfgParam Name="TOOL_COMMENT" Value="-"/>
      <MfgParam Name="FLUTE_LENGTH" Value="-"/>
      <MfgParam Name="HOLDER_DIA" Value="-"/>
      <MfgParam Name="HOLDER_LEN" Value="-"/>
      <MfgParam Name="SPINDLE_SENSE" Value="CW"/>
      <MfgParam Name="COOLANT_OPTION" Value="OFF"/>
      <MfgParam Name="COOLANT_PRESSURE" Value="NONE"/>
      <ToolSetUpOnWorkcell>
        <OffsetDataCollection>
          <OffsetData Tip="1" Comment="" OffsetZ="0.000000" Register="-1"/>
        </OffsetDataCollection>
      </ToolSetUpOnWorkcell>
    </Tool>
  </ToolingSetup>
</MfgSetupDocument>
```

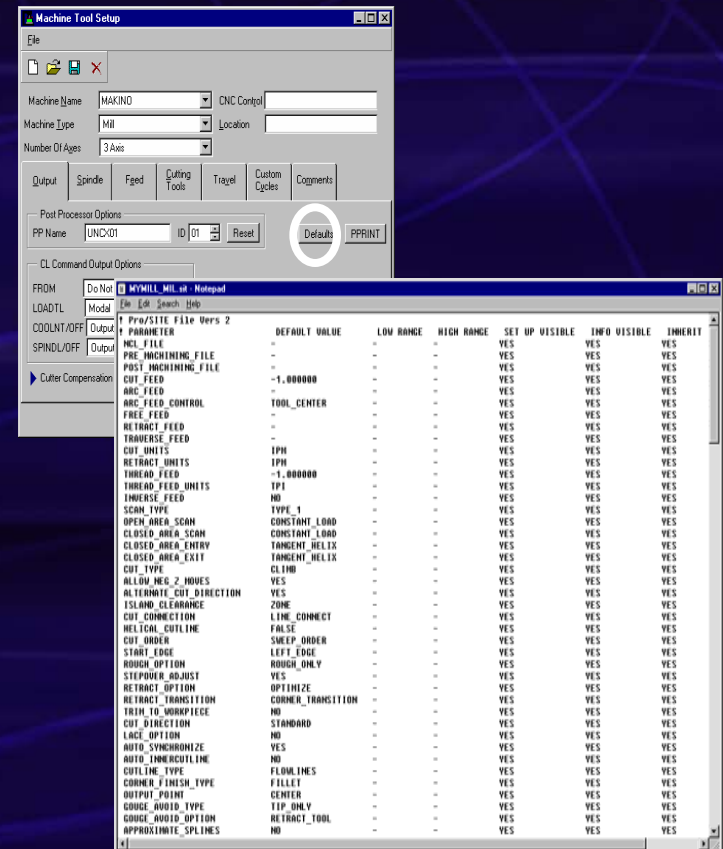
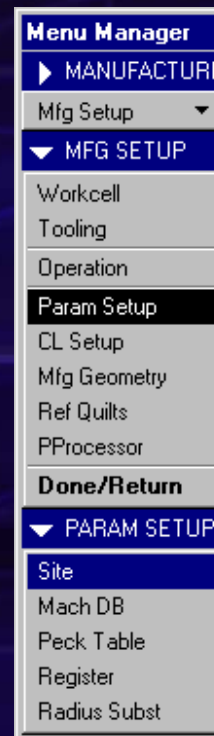

Parameter files

- Loading Parameters from a File
 - Store and retrieve parameters from your disk:
 - Store specific manufacturing strategy
 - Search path defines by Pro_mf_param_dir config.pro option (default is working directory)
 - Parameter(s) loaded from a file will overwrite current setting
 - Parameter file can contain from one to all the parameters for a specific toolpath
 - Good for automation :
 - **Create a Parameter with 1 single line:**
 - **Create a mapkey to load the parameter file**
 - **Create as many as you need**



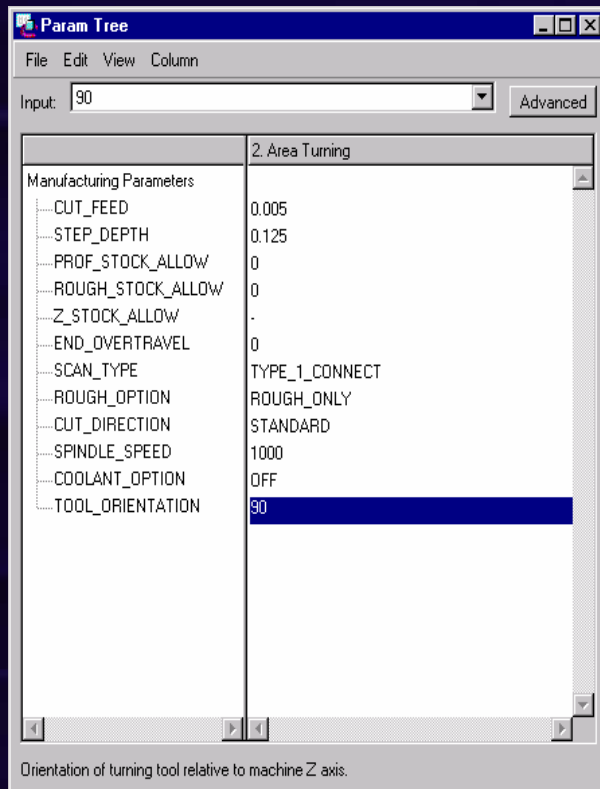
Site Files

- Site File allows you to setup defaults for all the manufacturing parameters
 - Created in Pro/NC or with a text editor
 - Loaded or stored with the Workcell
 - Parameter and Site files have certain '-1' options which need to be filled in every time
 - Minimum and Maximum values can be controlled
 - Visibility can be controlled (NC Sequence and Info)
 - Inheritance can be controlled (site changes - NC sequence changes)

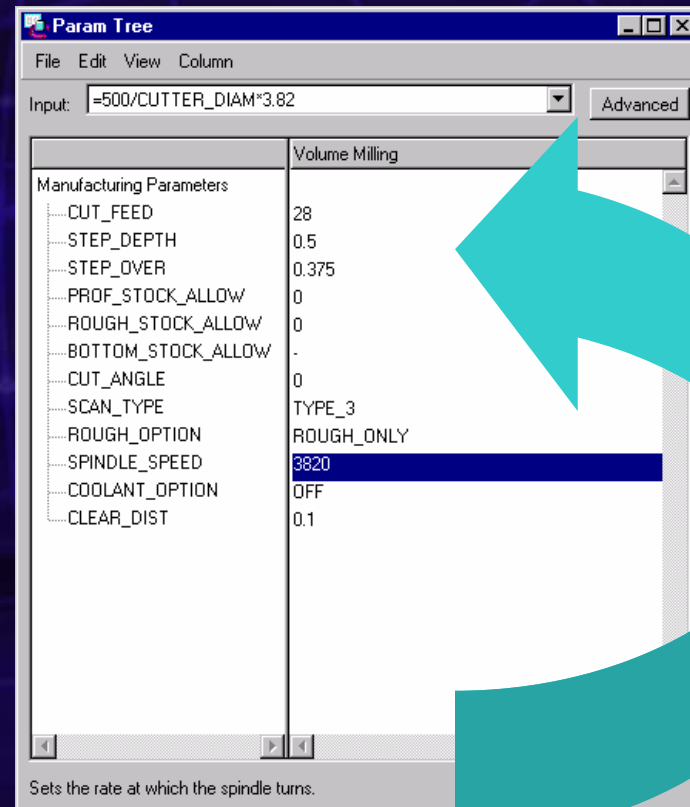


Site Files (con't)

- Values can be 'hard-coded'
- Driven via relations



-or-

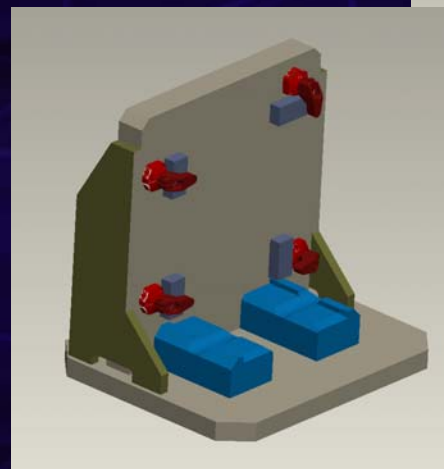
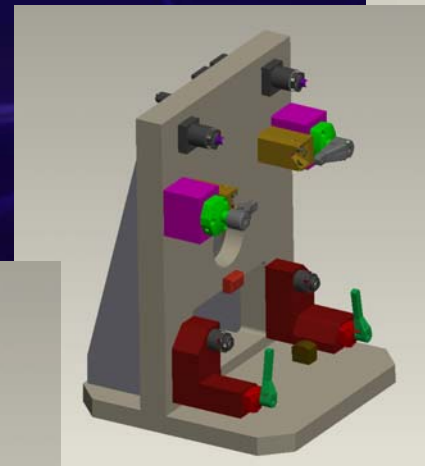
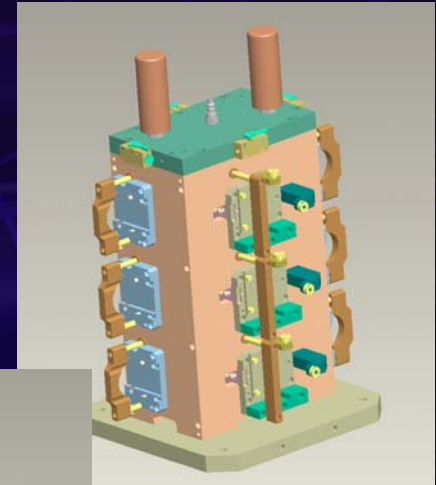


Spindle speed=500/cutter_diam*3.82

Relations can be used in parameters, too!

Fixtures

- Are optional, but can be useful for visualization and collision avoidance
- May consist of assemblies or a single part

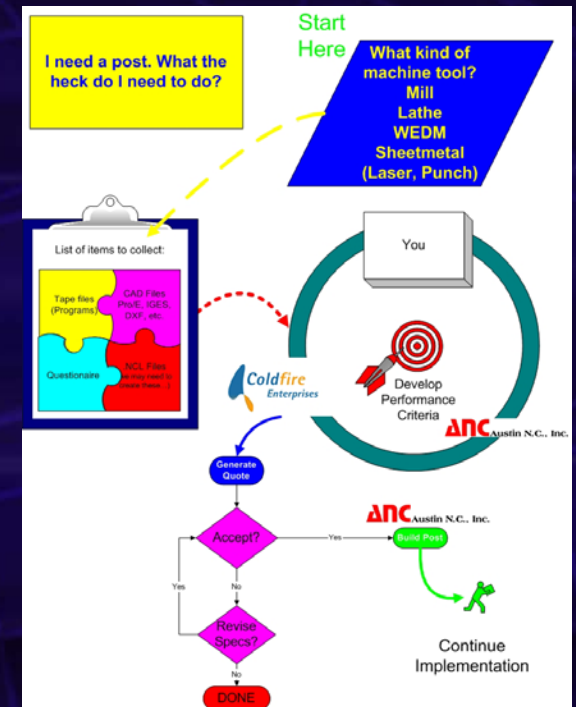
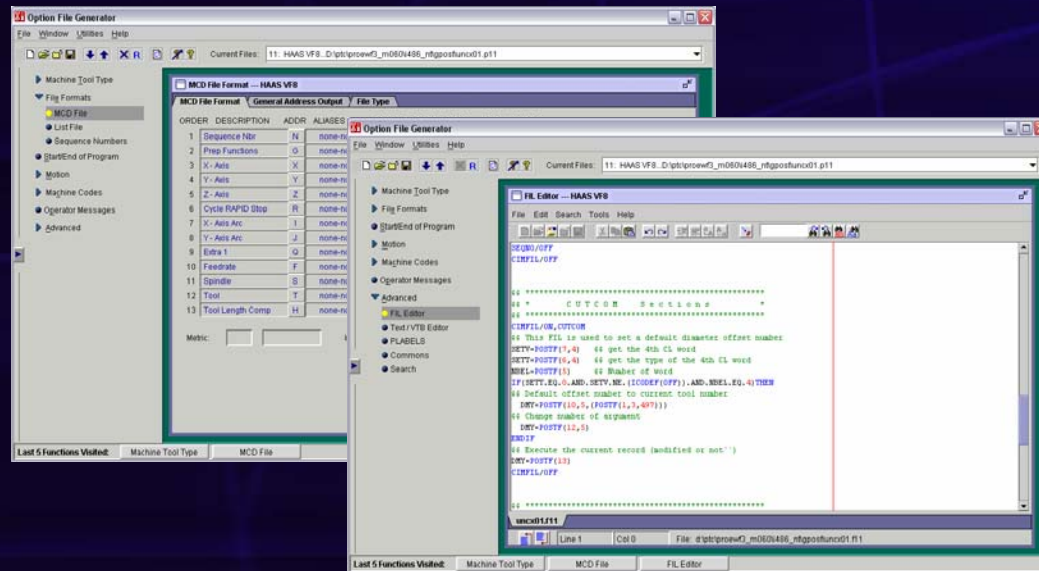


Post Processors

- Mills
- Wire EDM
- Lathes
- Sheet Metal

NC G-Post

- Best-in-Class Post Processor Development tool
- FREE! Bundled with Pro/NC

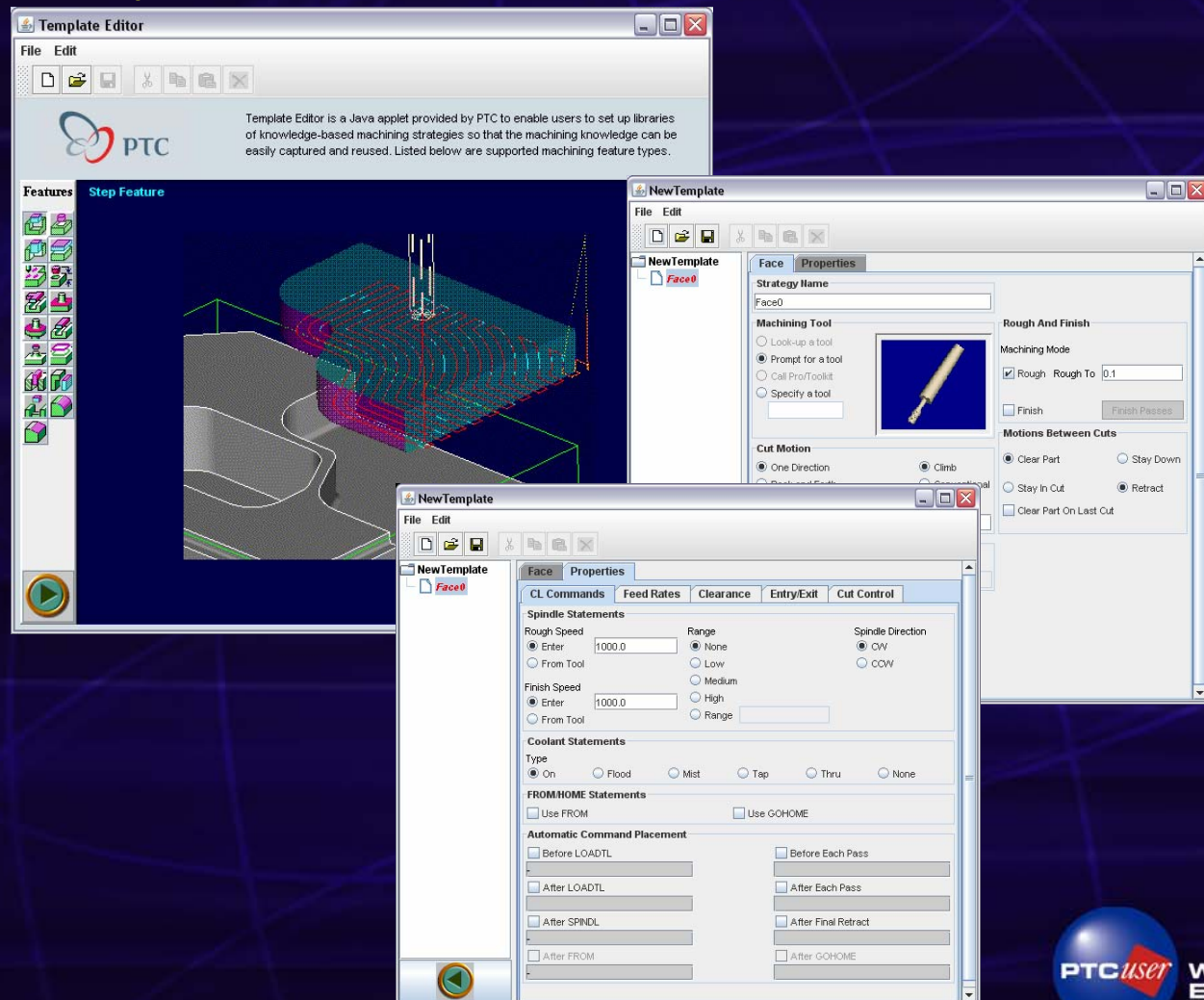


- Do it yourself, or
- Have one built

Customization

- Templates available in Expert Machinist:

- Step
- Pocket
- Profile
- Face
- Hole making
- Through Pocket
- Channel
- Slab
- Flange
- Slot
- Boss top
- O-Ring Groove
- Through Slot
- Undercut
- Rib top
- Round
- Chamfer

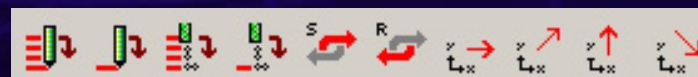
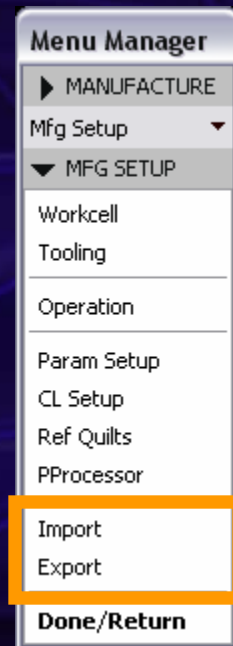
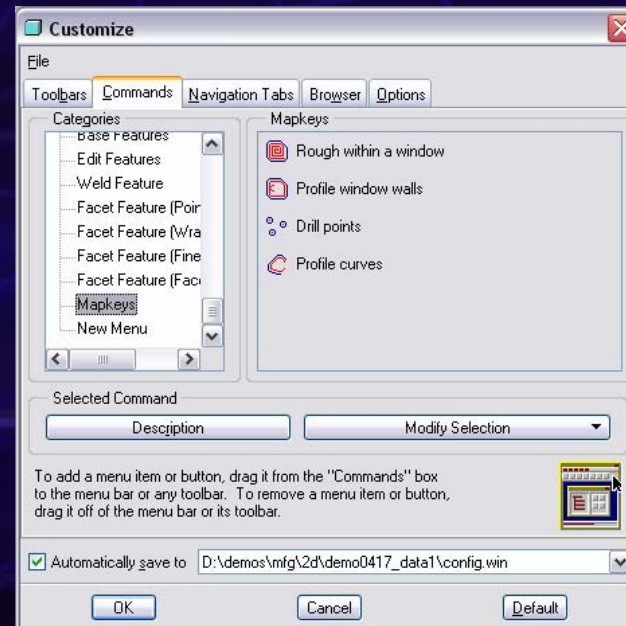
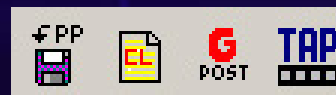


Customization (Con't)

Stuff you can build:

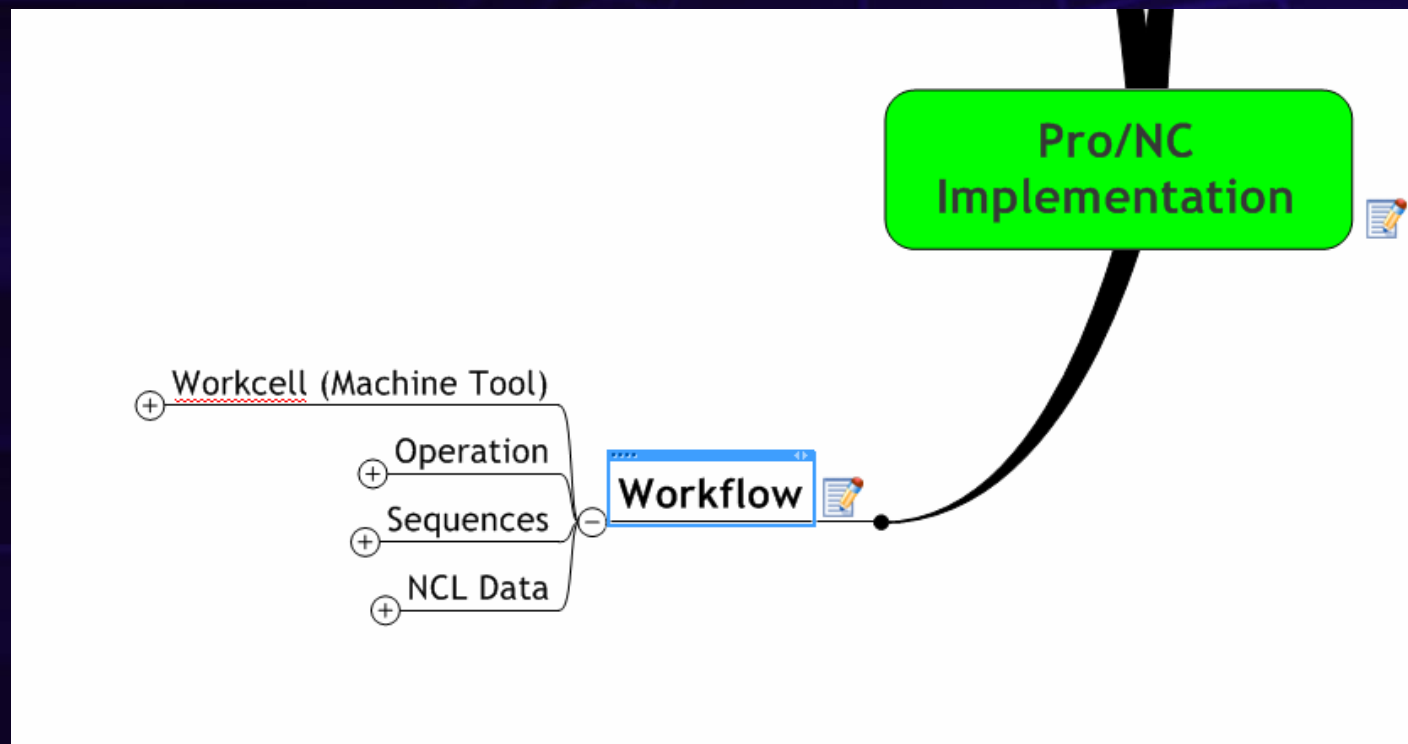
Start Models

- Fixtures
- Operations
- Workcells
- etc.
- Mapkeys
- Menus / Toolbars
- Icons
- Templates (XML)
- M/UDF's
 - Holes
 - Threads
 - Pockets
- Other Apps
 - Toolkit
 - Weblink
 - External Applications



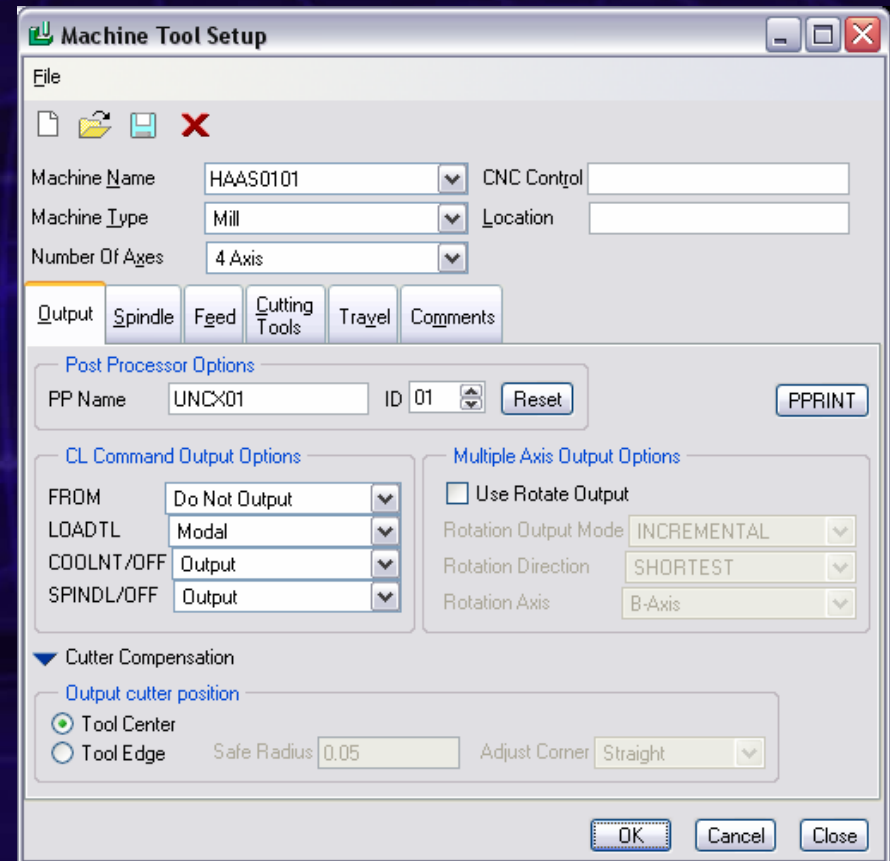
Define the workflow (or using the tools)

Step 2: Create the toolpaths



Workcell (Machine Tool)

- Type
 - Mill
 - Lathe
 - Wire EDM
 - Sheet Metal
- Number of Axes
- Output
- Spindle
- Feed info
- Cutting Tools
- Travel
- Custom Cycles
- Comments
- PPRINTS
- Cutter Comp. configuration



The image shows a screenshot of the 'Machine Tool Setup' dialog box. The 'File' menu is open, showing icons for file operations. The 'Machine Name' is set to 'HAAS0101', 'Machine Type' is 'Mill', and 'Number Of Axes' is '4 Axis'. The 'CNC Control' and 'Location' fields are empty. The 'Output' tab is selected, showing 'Post Processor Options' with 'PP Name' 'UNCX01' and 'ID' '01'. The 'CL Command Output Options' section has 'FROM' set to 'Do Not Output', 'LOADTL' to 'Modal', 'COOLNT/OFF' to 'Output', and 'SPINDL/OFF' to 'Output'. The 'Multiple Axis Output Options' section has 'Use Rotate Output' unchecked, 'Rotation Output Mode' set to 'INCREMENTAL', 'Rotation Direction' set to 'SHORTEST', and 'Rotation Axis' set to 'B-Axis'. The 'Cutter Compensation' section has 'Tool Center' selected, 'Safe Radius' set to '0.05', and 'Adjust Corner' set to 'Straight'. The 'PPRINT' button is visible. At the bottom are 'OK', 'Cancel', and 'Close' buttons.

Machine Tool Setup

File

Machine Name: HAAS0101 CNC Control:

Machine Type: Mill Location:

Number Of Axes: 4 Axis

Output Spindle Feed Cutting Tools Travel Comments

Post Processor Options

PP Name: UNCX01 ID: 01 Reset PPRINT

CL Command Output Options

FROM: Do Not Output

LOADTL: Modal

COOLNT/OFF: Output

SPINDL/OFF: Output

Multiple Axis Output Options

Use Rotate Output: ☐

Rotation Output Mode: INCREMENTAL

Rotation Direction: SHORTEST

Rotation Axis: B-Axis

Cutter Compensation

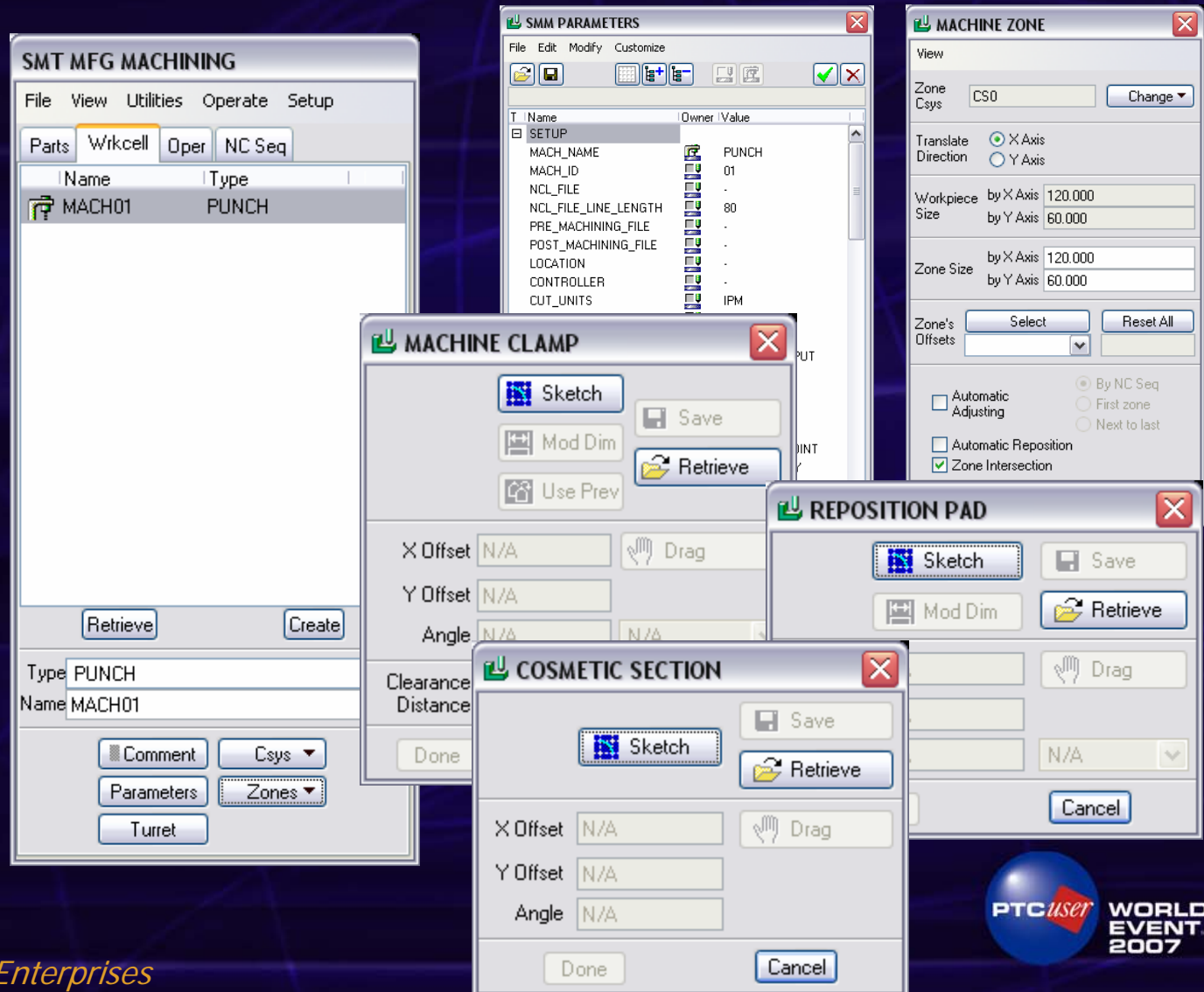
Output cutter position

Tool Center ☒ Tool Edge ☐ Safe Radius: 0.05 Adjust Corner: Straight

OK Cancel Close

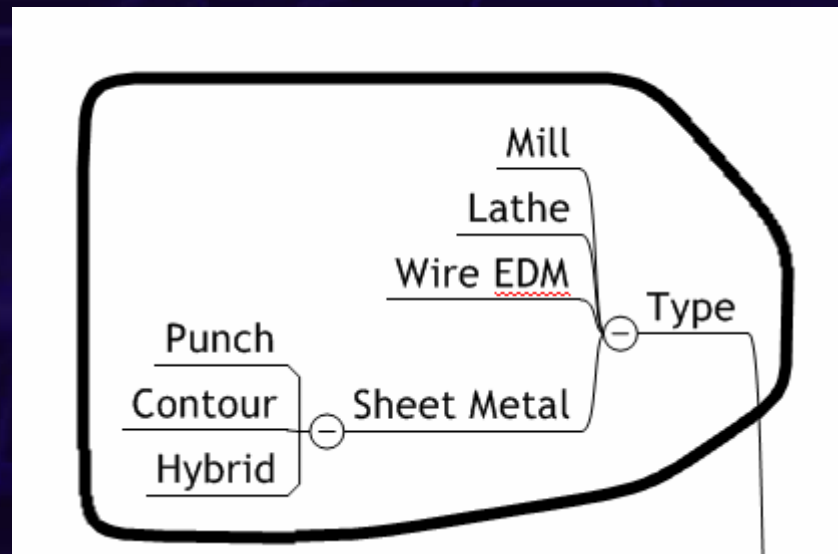
Workcell (Machine Tool) - Sheetmetal

- Sheetmetal
 - Parameters
 - Zones
 - Clamps
 - Turret
 - Pads



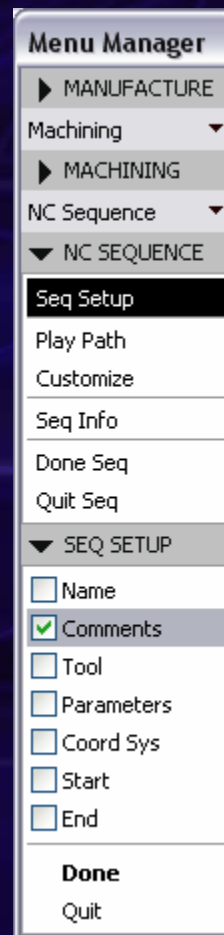
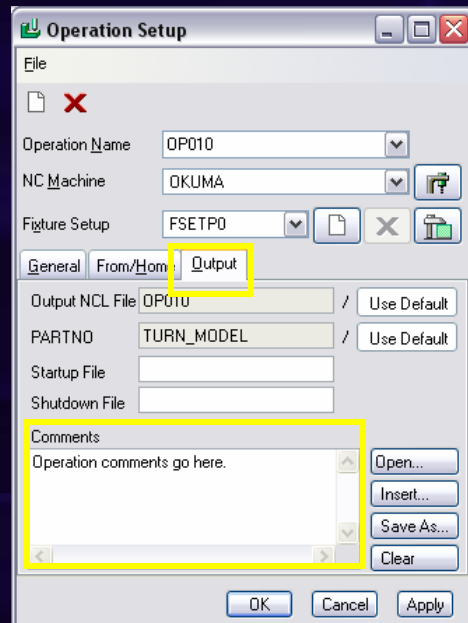
Type

- Mill
- Lathe
- Wire EDM
- Sheet Metal
 - Punch
 - Contour
 - Hybrid

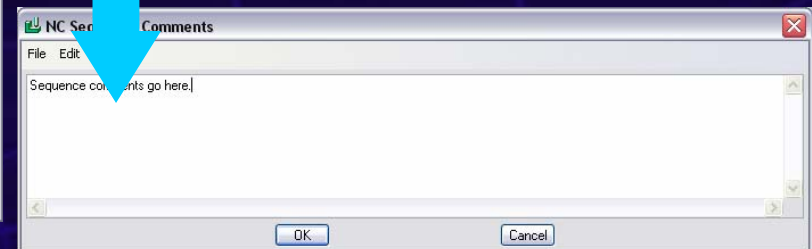


Comments

- Comments can be added at the operation level...

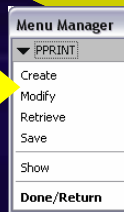
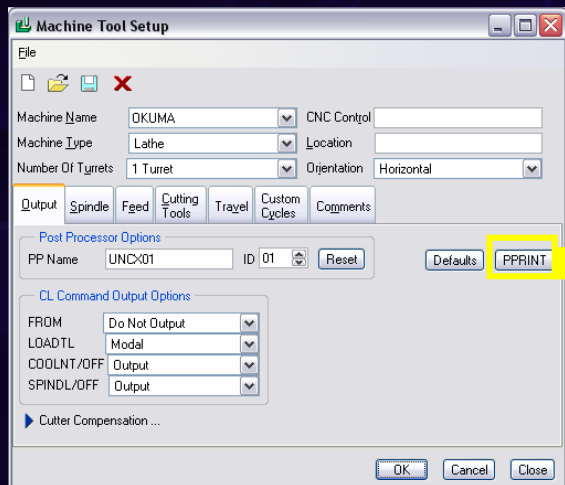


...or at the sequence level



PPRINTS

- Add information to be passed to the post processor, or the operator
- Save PPRINT file for later retrieval



Cutter Compensation configuration

- Defined in Workcell
- Output defined for either:
 - Tool Center
 - Tool Edge

The image shows a screenshot of the 'Machine Tool Setup' dialog box. The 'Output' tab is selected. Under 'Post Processor Options', 'PP Name' is 'UNCX01' and 'ID' is '01'. Under 'CL Command Output Options', 'FROM' is 'Do Not Output', 'LOADTL' is 'Modal', 'COOLNT/OFF' is 'Output', and 'SPINDL/OFF' is 'Output'. The 'Cutter Compensation' section is expanded, showing 'Output cutter position' with 'Tool Center' selected (indicated by a green dot) and 'Tool Edge' unselected (indicated by a blue circle). The 'Safe Radius' is set to '0.05' and 'Adjust Corner' is set to 'Straight'. The 'OK', 'Cancel', and 'Apply' buttons are at the bottom right.

Machine Tool Setup

File

Machine Name: OKUMA CNC Control: Location: Orientation: Horizontal

Machine Type: Lathe

Number Of Turrets: 1 Turret

Output Spindle Feed Cutting Tools Travel Custom Cycles Comments

Post Processor Options

PP Name: UNCX01 ID: 01 Reset Defaults PPRINT

CL Command Output Options

FROM: Do Not Output

LOADTL: Modal

COOLNT/OFF: Output

SPINDL/OFF: Output

Cutter Compensation

Output cutter position

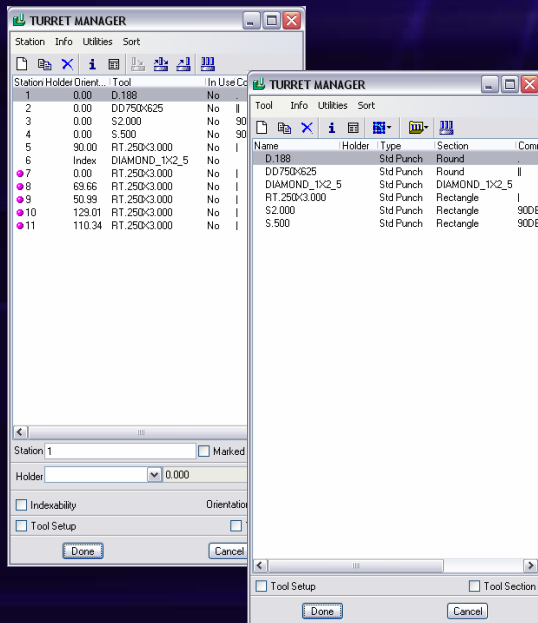
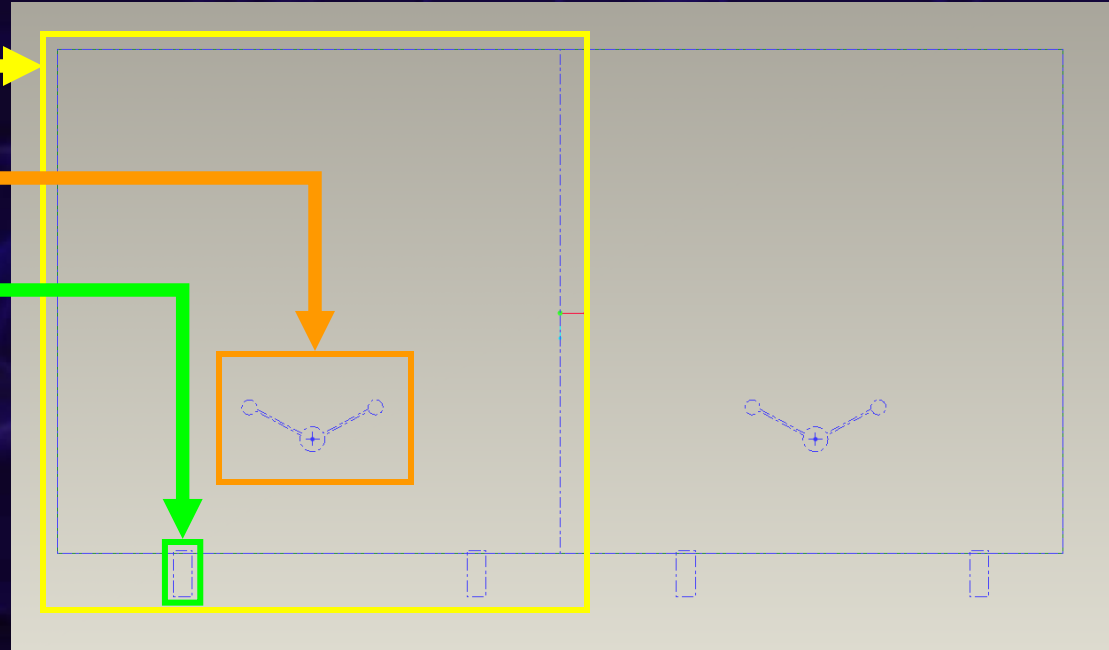
☒ Tool Center

☐ Tool Edge Safe Radius: 0.05 Adjust Corner: Straight

OK Cancel Apply

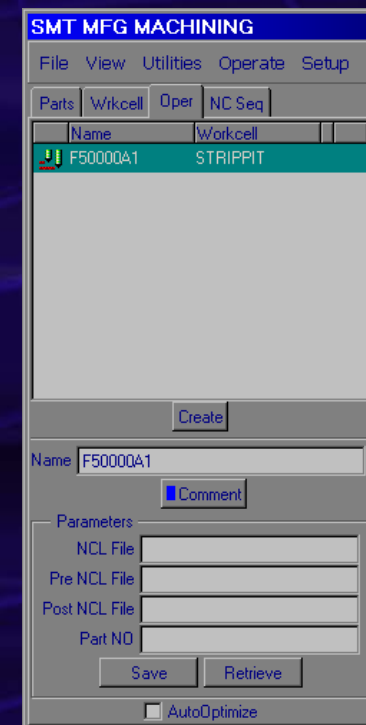
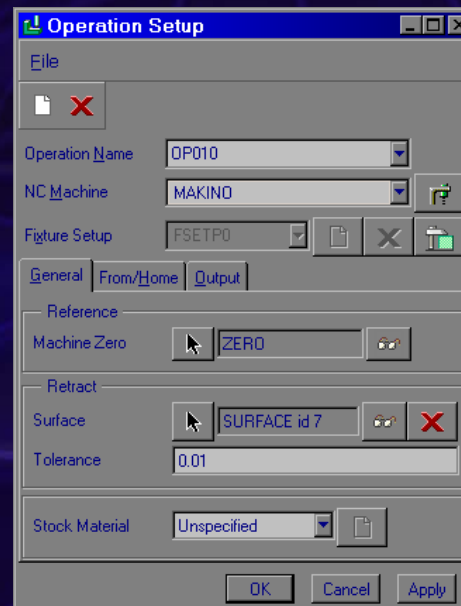
Sheetmetal

- Zones
- Pads
- Clamps
- Turret




Operation

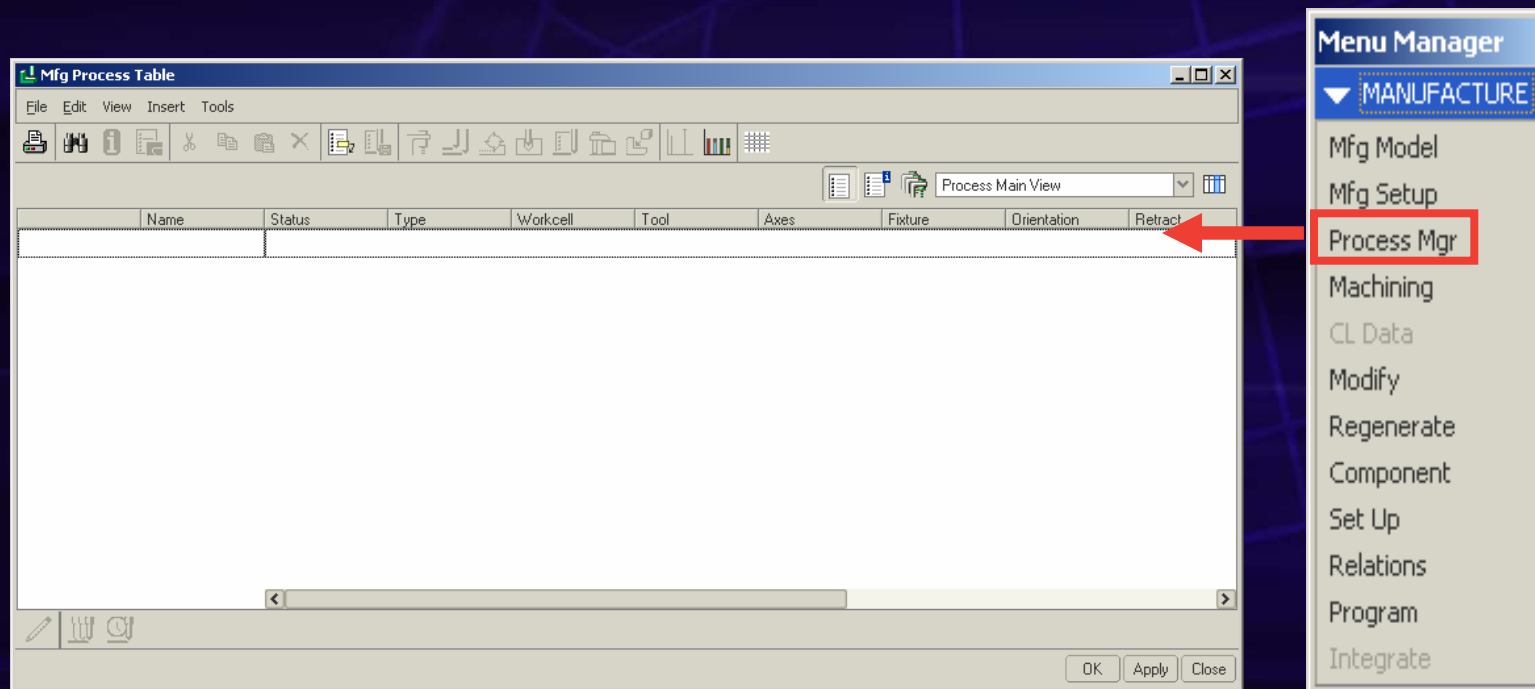
- Machine tool *
- Coordinate system *
- Retract *
- Stock Material
- Fixture Setup
- Mfg Geometry



* required!

Meet the Process Manager...

- The Process Manager can be called from a Pro/NC session
 - Manufacture / Process Manager 
 - This is the central control point for the process creation



Overview of the GUI

The screenshot shows the 'Mfg Process Table' window with a menu bar (File, Edit, View, Insert, Tools) and a toolbar. A table lists manufacturing steps, with step 10 highlighted in blue. Callout boxes point to various GUI elements:

- Create holmaking step**: Points to the 'New' icon in the toolbar.
- Create step from template**: Points to the 'New from Template' icon in the toolbar.
- Create Jig/Fixture**: Points to the 'New Jig/Fixture' icon in the toolbar.
- Create milling step**: Points to the 'New Milling' icon in the toolbar.
- Create Operation**: Points to the 'New Operation' icon in the toolbar.
- Create Workcell**: Points to the 'New Workcell' icon in the toolbar.
- Create Template**: Points to the 'New Template' icon in the toolbar.
- Highlight step references**: Points to the 'Highlight Step References' icon in the toolbar.
- View types**: Points to the 'View Types' icon in the toolbar.
- Selected Step**: Points to the selected row (step 10) in the table.
- Edit Selected Step**: Points to the 'Edit' icon in the toolbar.
- Play Path**: Points to the 'Play Path' icon in the toolbar.
- Time computation**: Points to the 'Time Computation' icon in the toolbar.
- Time Graph**: Points to the 'Time Graph' icon in the toolbar.
- View builder**: Points to the 'View Builder' icon in the toolbar.
- View selection**: Points to the 'View Selection' icon in the toolbar.
- Create NC Features**: Points to the 'Create NC Features' icon in the toolbar.
- Steps view area**: Points to the table area.

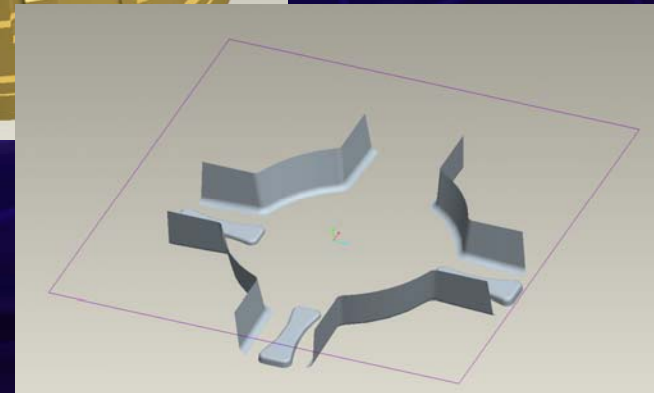
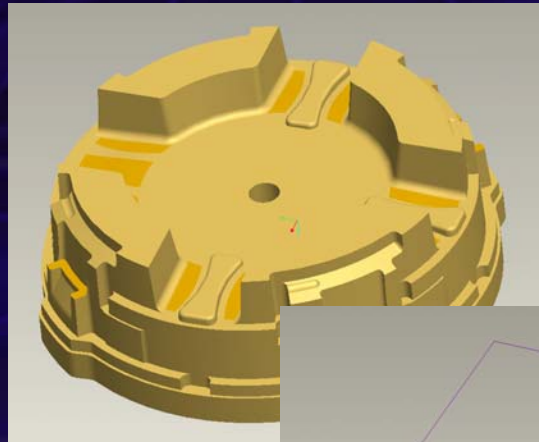
Name	Status	Type	Tool	Workcell	Axes	Fixture	Orientation
FSETP0	-	FIXTURE SETUP					
OP010	-	OPERATION		TOSHIBA			ACS0:F10
1. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		TOP_FAC
2. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		FACE1:FE
3. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		FACE2:F7
4. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		TOP_FAC
5. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		TOP_FAC
6. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		TOP_FAC
7. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		TOP_FAC
8. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		FACE1:FE
9. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		FACE1:FE
10. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		FACE1:FE
11. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		FACE1:FE
12. DRILL_1_INCH	-	STANDARD DR...	DR1000		3 Axes		FACE1:FE

Sequences

- Mfg Geometry (optional)
- Types
- Parameters
- Tools
- Geometry
- Customize

Mfg Geometry (optional)

- Can be used to define surfaces to cut
 - May be different from the reference model
- Use as a 'collection feature' to save time when selecting geometry to cut
- Types
 - Window
 - Surface
 - Volume
 - Hole Group



Sequence Types: Milling

- Volume
- Local Mill
- Surface
- Facing
- Profiling
- Pocketing
- Trajectory
- Aux
- Engraving
- Plunge
- Roughing
- Re-Roughing
- Finishing
- Holmaking
- Thread

Sequence Types: Turning

- Area
- Profiling
- Groove
- Thread
- Holmaking
- Aux

Sequence Types: Wire EDM

- 2 Axis
 - Contouring
- 4 Axis
 - Taper Angle
 - XY-UV Type
- Aux

Sequence Types: Sheetmetal

- Contouring
 - Aux
 - Contouring
 - Slitting
 - Approach Punch
- Punching
 - Edge Nibble
 - Area Nibbling
 - Slot
 - UDF Punch
 - Point Punch
 - Tool Shaping
 - Forming
 - Shearing
 - Aux

Sheetmetal (2)

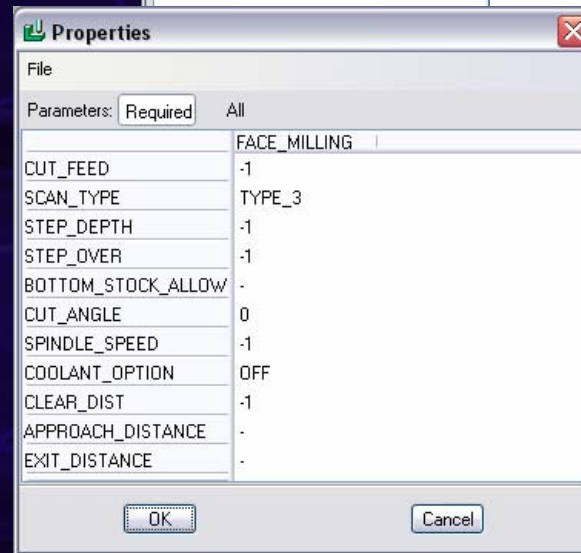
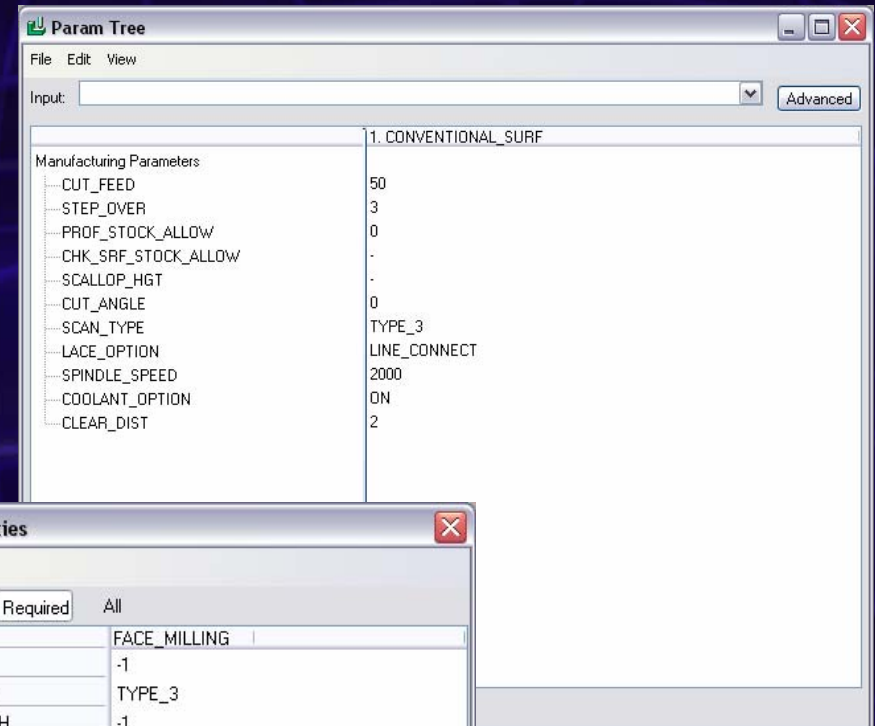
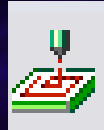
- Modifiers to Sequences include:
 - Lead in / out
 - Shakeaway tabs
 - Check Edges
 - Overhang
 - Multiple Tool (think long obround slots, use both round and rect.)
 - Remove Hits
 - Change Order
 - CL Command

Sheetmetal (3)

- Single part
- Automation (Nesting)
- Populate (Subroutines)
- Optimize
 - Control tool path after all parts are placed on the sheet.
 - Options for minimizing tool travel, tool change, etc.

Parameters

- Parameters are what defines *how* the geometry gets cut
 - *Step depth*
 - *Step over*
 - *Scan type*
 - *Etc.*
- Can have a dramatic effect on sequence
- Are context sensitive, based on sequence



Tools

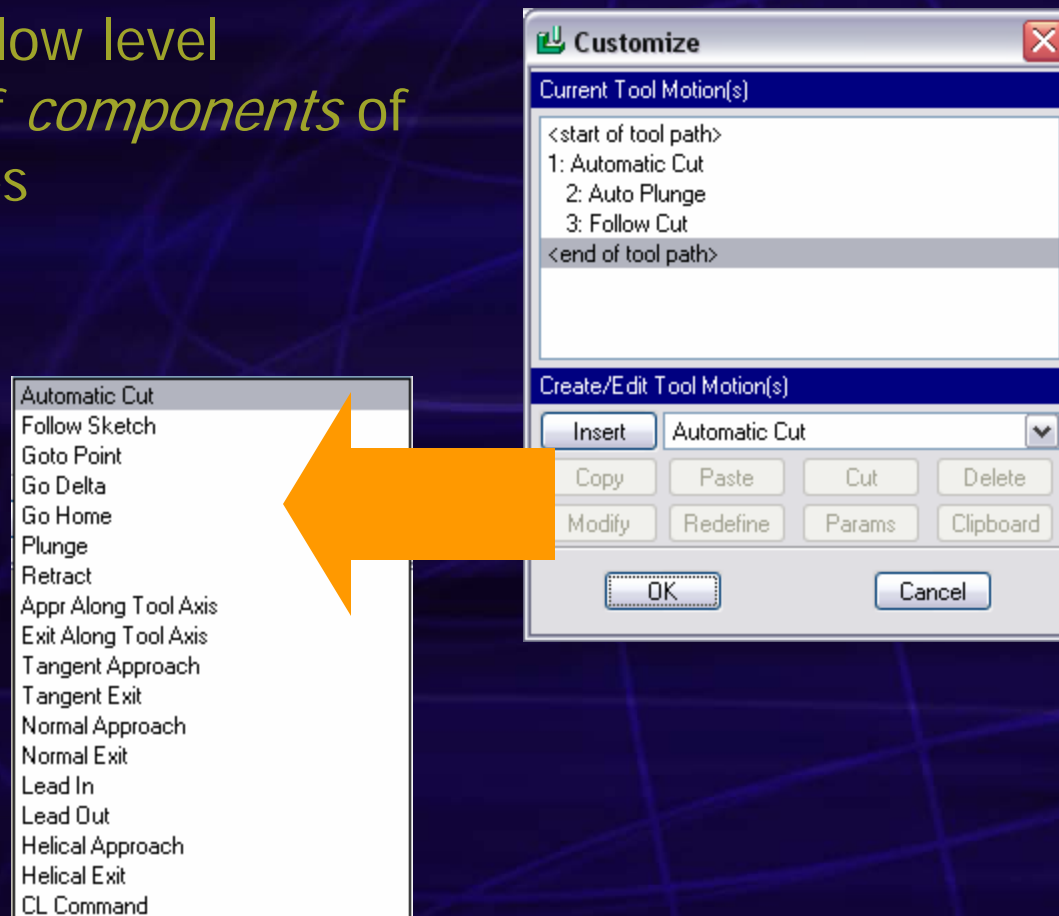
- Can be created on the fly or retrieved from library
- Can load parameters, based on material
 - Remember to specify material in the Operation
- 3 different types
 - Parameter based
 - Sketched
 - Solid part

Geometry

- Optionally selected from
 - Model
 - Sketched (think trajectory)
 - Manufacturing geometry
 - Selected
 - Created 'on the fly'
- Don't forget about imported and 2D legacy data!

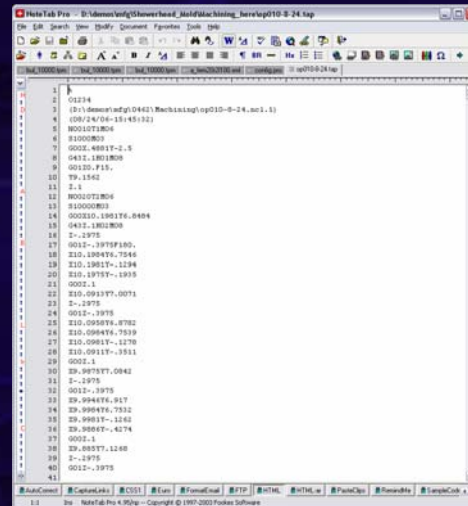
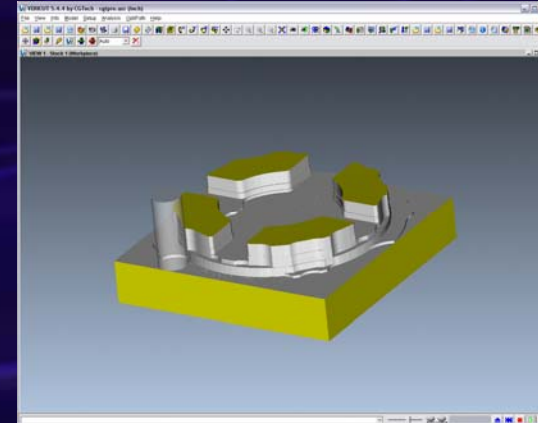
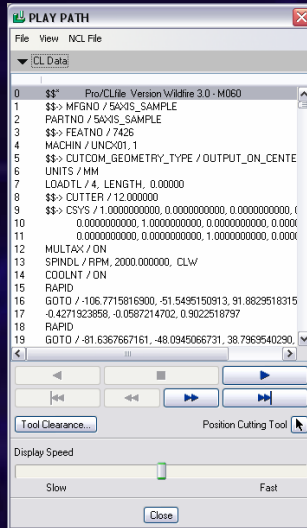
Customize

- Provides low level control of *components* of sequences



NCL Data

- Create
- Visualize
 - NC Check
 - Vericut
- Post Process
 - GPost



Documentation

- Standards
- Best Practices
- Process
- Tips & Tricks
- New Functionality / enhancements
- 'Loop back' to capture new automation as you develop it

config.pro for manufacturing

- Accuracy : it is strongly recommended to work with absolute accuracy enable.
 - **enable_absolute_accuracy** *yes*
 - **accuracy_lower_bound** *number (small enough, 0.001 metric)*
 - It is also important to be sure that all the components and the assembly have the same accuracy
- CL File output :
 - for 3 axis it is recommended to have at least one digit more than the number of digits required in the NC tape.
 - **mfg_xyz_num_digits** *4 for metric (5 for inches)*
 - *For 5 axis it is recommended to have at least 8 digits for I,J,K*
 - **Mfg_ijk_num_digits** *8*
- XML Import / Export : enable export and import on NC process in XML format
 - **output_mfg_xml** *yes*
- Remove arrow in the toolpath display
 - **cl_arrow_scale** *0*



config.pro for manufacturing (con't)

- VERICUT : to improve export performance for mold machining (simple stock)
 - **vericut_stock_tolerance 1 (mm)**
 - **vericut_fixture_tolerance 1 (mm)**
 - **vericut_export_all_tools YES**
- Workpiece transparency default value
 - **mfg_workpiece_transparency .75 (75% transparent)**
- Mill Window : automatic creation of the mill window based in workpiece silhouette
 - **allow_workpiece_silhouette auto**
 - **Other options are**
 - No : no workpiece taken in account (default)
 - Yes : user select workpiece or ref model
 - Auto : workpiece silhouette only

config.pro for manufacturing (con't)

- Make job manager available (optional)
 - nc_jobman_visible yes
- Support for multi-tips tool
 - Allow_multiple_tips_tool yes
- Post-processor directory definition
 - gpostpp_dir *path* (path can be .\ for current directory)
- Important paths to define
 - Mfg_start_model_dir path to manufacturing start parts
 - pro_mf_param_dir NC sequence parameters library path
 - Pro_mf_tprm_dir Tool library path
 - Mfg_template_dir path to manufacturing template directory