Pro/ENGINEER®
Wildfire™ 2.0

Pro/NC-SHEETMETAL™
Help Topic Collection

Parametric Technology Corporation

Third-Party Technology Information

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494 USA

Table Of Contents

Foundation Modules

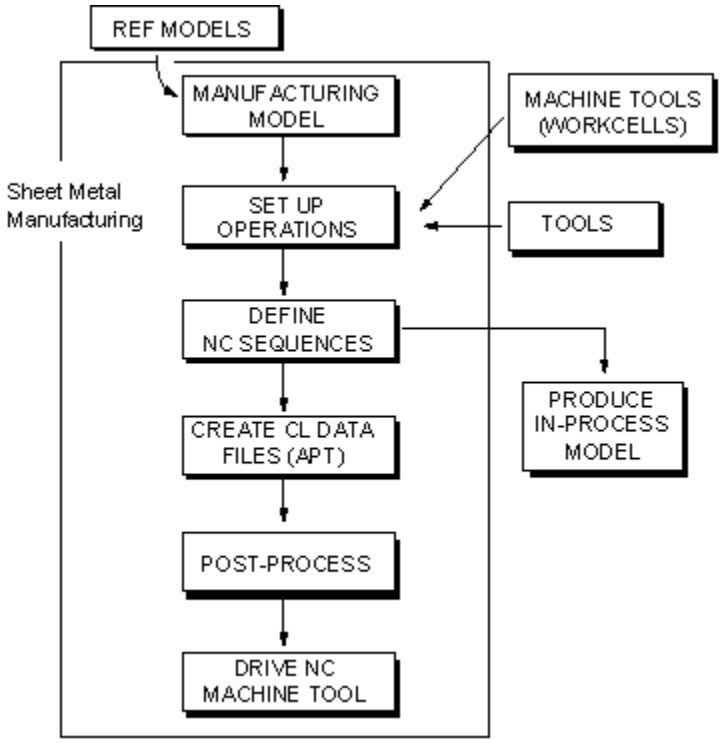
Pro/NC-SHEETMETAL

Using Pro/NC-SHEETMETAL

About Sheet Metal Manufacturing

Sheet Metal Manufacturing creates the data necessary to drive an NC machine tool to machine parts from a sheet workpiece. It does this by providing the tools that let the manufacturing engineer set up the manufacturing model, and then follow a logical sequence of steps to progress from the manufacturing model to ASCII CL data files that can be post-processed into NC machine data. The following illustration summarizes the Sheet Metal Manufacturing process.

The Sheet Metal Manufacturing Process



To Enter Manufacturing Mode

To work in manufacturing mode, you have to retrieve an existing manufacturing model or create a new one. To start up with a new manufacturing model, you need a reference model (or models).

Note: The workpiece is created "on the fly."

Create these in Sheet Metal mode before entering Manufacturing.

Pro/NC-SHEETMETAL Configuration Options

About Configuring Pro/NC-SHEETMETAL

Pro/NC-SHEETMETAL configuration options enable you to customize your sheet metal design environment. For example, you might specify constants for neutral bend lines, enable corner relief notes and punch axis points, set directory locations, or define certain material behavior in your sheet metal design.

See the Pro/SHEETMETAL online Help for descriptions of all applicable sheet metal configuration options.

pro_mf_clamp_dir

Specifies the directory where the manufacturing clamps are found.

You must type the full directory path for **Value**, for example, `c:\program files\ptc\clamp`.

Model Tree

About the Model Tree

You can display a graphical hierarchy of a manufacturing model in the form of a Model Tree window. When you create or retrieve a manufacturing model, the system displays the Model Tree window.

To Display Manufacturing Features in a Model Tree Window

In the Model Tree, click **Tree > Show > Expand All**. The system displays all the manufacturing features in the Model Tree at all levels.

To Display Parent/Child Relationships in the Model Tree Window

In the Model Tree, select a feature. Right-click the feature and select **Info** > **Parent/Child**. **Reference Information Window** appears. The parents and children of the current selected feature are displayed.

To Select Features to Display in the Model Tree Window

1. In the Model Tree, click **Tree > Settings > Tree filters**. The **Model Tree Items** dialog box opens.
2. Under **Feature Types** click the **MFG** tab.
3. Select the feature type(s) you want to show in the Model Tree. The following options are available:

- **Operation**

- **Workcell**
- **Sequence**
- **Mfg Geometry**
- **Material Removal**
- **Show all features**—If you select this option, the checkmarks next to all the options above disappear. The Model Tree will contain all the features (including regular Pro/ENGINEER features) in the manufacturing assembly and in all the components.

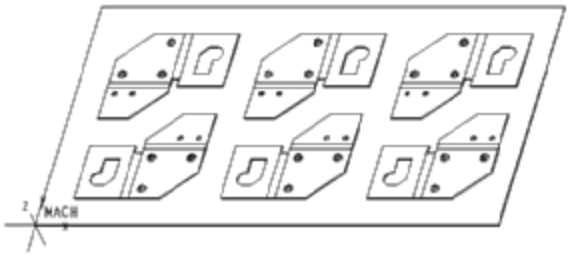
4. Click **OK**.

Manufacturing Model

About Manufacturing Model

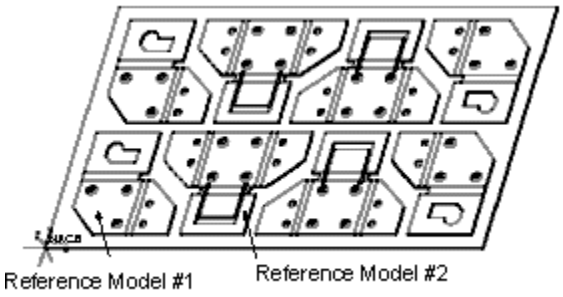
Place the reference model(s) in the desired locations on the workpiece to create a manufacturing model. Use either the nesting functionality or the regular assembly placement commands. (See the following illustration). You can create NC sequences by referencing geometry of design model(s).

Sheet Metal Manufacturing Model



You can use different design models as reference parts within a single manufacturing model. (See the following illustration.)

Sheet Metal Manufacturing Model—Multiple Reference Models



Creating the Sheet Metal Manufacturing Model

The sheet metal manufacturing model is an assembly that consists of a sheet metal workpiece with one or more design (reference) models attached to it.

The sheet metal workpiece represents the raw sheet metal stock that is machined by the NC sequences. It is the base component of the sheet metal manufacturing assembly.

To Create a New Manufacturing Model

1. Click **FILE > New**. The **New** dialog box opens.
2. Select **Manufacturing** from the **Type** list and **Sheetmetal** from the **Sub-Type** list.
3. Type a name in the **Name** box for the new manufacturing model.
4. Click **OK**. The manufacturing model is created.

Using Assembly Commands

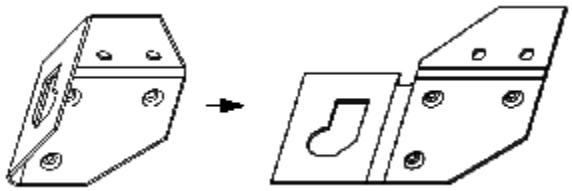
The **Components** menu in the **Parts** tabbed page enables you to assemble reference parts to the workpiece using the regular assembly placement commands instead of the nesting functionality, as well as perform other assembly actions on individual members of the manufacturing model.

Note: You cannot create parts in Sheet Metal Manufacturing mode.

Design Model

The Design (Reference) Model, represents the finished product and is used as the basis for all NC sequences. Select features, surfaces, grain orientation and edges on the design model as references for each NC sequence. Referencing the geometry of the design model sets up a parametric relationship between the design model and the workpiece. When the design model is changed, all associated NC sequences are updated to reflect the change.

The Design (Reference) Model



Before using a sheet metal part as a manufacturing reference model, make sure that it is unbent into the flat state. Add an Unbend All feature if necessary.

To Retrieve an Existing Manufacturing Model

1. Click **FILE > Open**. The **File Open** dialog box opens.

2. Select the desired file.
3. Click **OK**.

To Modify a Manufacturing Model

You can manipulate the workpiece and the reference models as any other part. You can modify their features, redefine their features, suppress their features, etc. To do this, choose **Modify** from the **SMT MFG** menu and use the following options:

- **Mod Part**—Create, delete, suppress features, and modify dimensions of the design model or the workpiece. Use **Pick** or **Query Sel** to select the part to modify.

  The following two options (**Mod Ref** and **Mod Work**) make the process of selection easier.

- **Mod NC Seq**—Create, delete, suppress features and modify dimensions of the NC sequence
- **Mod Pattern**—Create, delete, suppress features and modify dimensions of the pattern.
- **Mod Assem**—Create, delete, suppress assembly features and modify dimensions of the manufacturing assembly.
- **Mod Subasm**—Create, delete, suppress assembly features and modify dimensions of the manufacturing subassembly.
- **Mod Dim**—Modify any and all dimensions (part, assembly, nesting, etc.).

Regenerating a part recalculates the geometry of a part after modifications.

To Regenerate a Manufacturing Model

1. To perform regeneration in Manufacturing mode, click **Regenerate** from the **Parts** tabbed page in the **SMT MFG MACHINING** dialog box. The **PRT TO REGEN** and **SELECT PARTS** menus appeaar.
2. Select parts to regenerate. Select **Automatic**.

   When you select the workpiece for regeneration, the **REGEN TYPE** menu appears with the following options:

   - **Normal**—Regenerate a part normally.
   - **Slow**—Regenerate a part, and displays the part and tool path after regenerating each manufacturing feature.
   - **Step**—Regenerate a part, waits for user input after regenerating, and displays each manufacturing feature.
   - **Supp Fail**—Automatically suppress any features that fail upon regeneration, along with their children.

- **Compute CL**—Specify if you want the CL data to be recalculated at regeneration time.

When you specify normal regeneration, the workpiece regenerates as any other part. The part feature currently regenerates in the following form: `Regenerating part.prt feature 10 out of 20`. This process continues with no user input until all modified features regenerate. At this time, the regenerated part appears.

Slow Regeneration

When you specify slow regeneration, the workpiece regenerates and appears one feature at a time. When you regenerate an NC sequence, the cutter path and tool display. After the first feature regenerates and displays, there is a short pause, then the next feature regenerates.

This process continues until all modified features regenerate.

Step Regeneration

When you specify step regeneration, the workpiece regenerates and appears one feature at a time. After each feature displays, you must enter a carriage return `<CR>` to continue regeneration. Continue this process to step through the regeneration of all modified features.

Regeneration Options

If you select **Supp Fail**, the system automatically suppresses any features and NC sequences that fail upon regeneration, along with their children. To select the **Supp Fail** option, click it once; a check mark (✓) appears. To turn the check mark off, click once more.

If a check mark next to the **Compute CL** option is on, the system recalculates CL data every time it regenerates an NC sequence. To reduce the regeneration time, turn it off. If **Compute CL** is off, only workpiece geometry regenerates. When you choose **CL Data**, **Output**, the system reports any errors that may occur with **CL data**.

To Change the Coordinate System

The coordinate system is defined in the **Wrkcell** tabbed page of the **SMT MFG MACHINING** dialog box. You can, however, switch to another coordinate system at any time during the manufacturing process.

1. Click the **WRKCELL** tabbed page in the **SMT MFG MACHINING** dialog box.
2. Click **Select** from the **Csys** menu.
3. Select the desired coordinate system.

The workpiece automatically regenerates, and the system recalculates CL data for all existing NC sequences with respect to the new coordinate system.

To Change the Units of a Manufacturing Model

1. From the **SMT MFG MACHINING** dialog box, click **Setup > Model Setup > Units**. The **Units Manager** dialog box opens.
2. Change the model units as desired. The following options are available:
   - **Set**—Set the system of units.
   - **New**—Create a customized system units using the **Systems of Units Definition** dialog box.
   - **Copy**—Copy the selected system of units using the **Copy System of Units** dialog box.
   - **Edit**—Edit the selected system of units using the **Systems of Units Definition** dialog box.
   - **Delete**—Delete the selected system of units.
   - **Info**—Obtain information about the selected system of units.
3. Click **Close**.

To Define a Manufacturing UDF

1. From the **SMT MFG MACHINING** dialog box, click **Utilities> Feature > UDF Library**. The **UDF** menu appears.
2. Click **Create**.
3. Specify the name for the group. The **UDF OPTIONS** menu appears.
4. Select an option from the **UDF OPTIONS** menu:
   - **Stand Alone**—The UDF is functional by itself.
   - **Subordinate**—The UDF is driven by the current model.
5. Define the UDF elements as you would when creating a group of part features. Select manufacturing features (NC sequences and material removal features) to be grouped.
6. If a features' group is referenced by selected NC sequences, the **MFG UDF REF** menu appears. Select one of the following options:
   - **Design UDF**—Use a reference group of features to resolve references. If only one design UDF is referenced, it is selected automatically, otherwise select a group to use.
   - **Ref Part**—Use the reference part to resolve references.
   - **None**—Stores no reference information.

   If no design UDF is referenced, the **MFG UDF REF** menu does not appear. You must select a reference part to use. If you do not want to select a reference part, click **Done Sel** without selecting a part.

7. After you successfully define the group, Pro/ENGINEER automatically stores the information.

Note: If you specify a reference design UDF, Pro/ENGINEER does not prompt you for the corresponding placement references; they are resolved automatically.

To Place a Previously Defined Group in Another Manufacturing Model

1. From the **SMT MFG MACHINING** dialog box, click **Utilities > Feature > Group**.
2. Click **Create** from the **GROUP** menu.
3. Click **From UDF Lib** from the **CREATE GROUP** menu.
4. Retrieve the group by name. The **PLACE OPTS** menu appears.
5. Select if the group is to be **Independent** or **UDF Driven**. You are prompted to enter group elements.

   As you answer the prompts for group elements, the **WHICH REF** menu appears. Select one of the following options:

   - **Alternate**—Select a reference for the current element.
   - **Same**—Use the same reference as in the reference part. This option appears only if the group contains a reference part information, and this reference part is present in the current manufacturing assembly.
   - **Skip**—Skip the current prompt without selecting a reference. After you answer all the other prompts, the system enables you to redefine the skipped element.

   Note: The **Skip** option is available only for UDFs created in Release 16.0 and later.

6. After the UDF placement is completed, a namelist menu of all NC sequences included in the group appears. Check off one or more NC sequences whose tool or parameters you want to modify (you can use **Select All**). The **MOD NC SEQ** menu appears with the following options:

   - **Tool**—Changes the tool.
   - **Parameters**—Modifies the NC sequence parameters.

   Click **Done** when finished.

   Note: If you skip some of the prompts, the appropriate user interface is invoked to enable you to redefine the skipped element.

7. The **GRP PLACE** menu then appears. Select one or more of the following options:

   - **Redefine**—Redefines all elements related to the skipped prompts.

- **Show Result**—Previews the group.
- **Info**—Displays information about the group being created in the Information Window.

Choose **Done** from the **GRP PLACE** menu to finalize the group.

8. The group is placed in the new model.

Manufacturing Process

About the Manufacturing Process

A sheet metal manufacturing process consists of the following basic steps:

1. Set up the manufacturing database. It may contain such items as workcells (machine tools) available, tooling, site parameters, etc. If you do not want to set up all your database up front, go directly into the machining process and define any of the items above when you need them.

2. Define an operation. An operation setup may contain the following components:

- Operation name
- Workcell (machine tool)
- Coordinate system for CL output
- Operation comments
- Operation parameters

The system prompts you to specify a workcell and a coordinate system before you can create NC sequences (the coordinate system is usually defined earlier, at the time the reference parts are nested in the workpiece). Other setup components are optional.

3. Create NC sequences under the specified setup. Each NC sequence is a series of tool motions with the addition of specific post-processor words that are not motion-related but required for the correct NC output. The system automatically generates a tool path based on the NC sequence type (e.g., Nibble Edge, Form, etc.), reference geometry, and manufacturing parameters. For Contouring, you can generate various types of tool paths by referencing model geometry. You can also apply more low level control, if you like, by:

- Modifying tool paths generated by the system (e.g., clipping, extending, reversing direction).
- Interactively adjusting the tool path. For Contouring, this includes defining approach, exit, corner conditions, shakeaways, and CL commands. For other sheet metal NC sequence types, you can skip or reorder hits that the system generates. You can also insert non-motion CL commands for any type NC sequences.

4. You are required to define NC sequences for only one instance of a reference part on the workpiece. The Populate functionality enables you to copy these tool paths to all other occurrences of this reference part in the manufacturing model.

For each completed NC sequence, you can create a material removal feature, either by making the system automatically remove material (where applicable), or by manually constructing a regular Pro/ENGINEER feature on the workpiece (Slot, Hole, etc.).

SMT Manufacturing Info Box

Once you choose **Manufacturing** from the Info menu, the **SMT Manufacturing Info** dialog box containing the current manufacturing information appears. You can view specific manufacturing information by selecting items in the filter list.

About SMT MFG MACHINING

Use the **SMT MFG MACHINING** dialog box to create the data necessary to drive an NC machine tool to manufacture parts from a sheet metal workpiece. Functions such as nesting, tool setup, optimizing, and NC sequencing are controlled using the following buttons located in the **SMT MFG MACHINING** dialog box.

Parts

- **Nesting**—Nest Parts manually on the sheet metal workpiece.
- **Automation**—Enter the Autonesting mode.
- **Components**—Assemble parts on the sheet metal workpiece.
- **Regenerate**—Regenerate the manufacturing assembly using different modes.

Wrkcell

- **Csys**—Select new machine Csys data.
- **Parameter**—Specify workcell parameters.
- **Zones**—Define machine zones, pads, clamps, cosmetic features, and so forth.
- **Turret**—Define or modify the turret manager.

Oper

- **Create**—Create a new operation.
- **Retrive**—Retrive an existing operation.

NC Seq

- **Populate**—Populate NC sequences.
- **Auto**—Create Autotool NC sequences.

- **Optimize**—Optimize the CL output.
- **New**—Create a new NC sequence.
- **CL Output**—Open NCL player to output CL file.

To Create a New Workcell

1. Select the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Create** to create a new workcell.
3. Select the workcell type from the **Type** drop-down menu. The following options are available:
    - **Punch**—Perform **Nibble Edge**, **Nibble Area**, **Punch**, **Form**, and **Shear** NC sequences. These NC sequence types are jointly referred to as Punch NC sequences.
    - **Laser**—Perform laser NC sequences.
    - **Laser-Punch**—Perform laser and punch NC sequences.
    - **Flame**—Perform flame NC sequences.
    - **Flame-Punch**—Perform flame and punch NC sequences.
4. Once you have specified the workcell type, you can choose the following options:
    - **Name**—Specify the name of the workcell. The system automatically assigns the default name (MACH01, MACH02, etc.).
    - **Parameters**—Specify the workcell parameters in the **SMM PARAMETERS** dialog box.
    - **Turret**—Set up the turret associated with the workcell in the **TURRET MANAGER** dialog box. This enables you to set stations to be indexable or not, and specify tools and orientation for these stations (optional).
    - **Csys**—Specify a coordinate system for CL data output.
    - **Zones**—Create a machine zone.
    - **Comment**—Create a comment.

    If you do not select any other options, the new workcell has a default name (MACH01, MACH02, etc.), a default set of parameters, and no tools associated with it. All the tools that you subsequently use for the NC sequences performed on this workcell are associated with the workcell.

To Retrieve a Workcell

1. Select the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.

2. Click **Retrieve**. The **Open** dialog box appears with a list of names for the current working directory (or for the library directory, if pro_mf_workcell_dir has been set).
3. Select the name of the workcell from the list.
4. Click **OK**.

To Modify, Redefine or Delete Workcells

1. Select the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Select the workcell to modify.
3. Change the **Name**, **Comment**, or **Parameters** options as desired or click **Operate > Delete** to delete the workcell.

To Modify an Existing Cell

1. Select the **Part** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Redefine** from **NESTING** menu.
3. Cyan rectangles enclose currently defined cells. Select one of the models in the cell to modify it. The **NEST CELL** menu appears with the following options:
   - **Add Part**—Bring a new reference part into the manufacturing model and the current cell (as described above).
   - **Drag Part**—Modify placement of a part currently in the cell. Select a part to drag. The **PART PLACE** menu appears enabling you to change the drag origin or increments. Select the desired options and click **Done**. Then place the part using the mouse buttons.
   - **Delete Part**—Remove a part currently included in the cell from the cell and from the manufacturing model.
   - **Note**: The first part in a cell cannot be removed.
   - **Nest Info**—Provide nesting information (similar to the **Info** option in the **NEST** menu).

To Save a Workcell

1. Select the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Select the workcell to be saved.
3. Click **Operate > Save**. The workcell is saved in the current working directory (or in the library directory, if `pro_mf_workcell_dir` has been set) as `<workcellname>.gph`.

Saving/Retrieving Workcells

Workcells are saved as user-defined features (groups) that contain a single feature (the workcell).

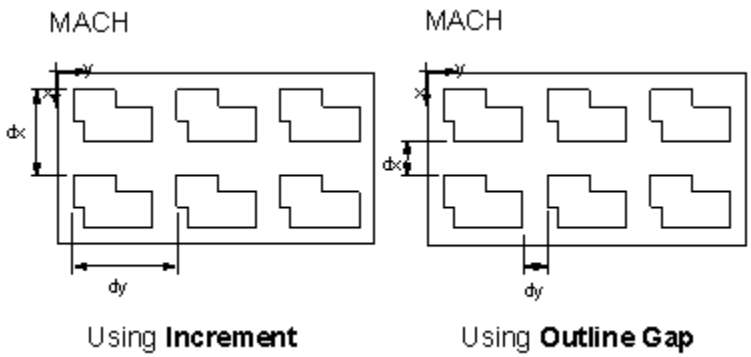
The configuration file option `pro_mf_workcell_dir` enables you to specify a library directory for workcells. There, the workcell files are available to all users for retrieval into their manufacturing processes. The value of the option is the path name (absolute path is recommended) of the directory where the workcell files are stored.

To Change a Pattern of Cells

1. Select the **Part** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Multiply > Define** from the **NESTING** menu.
3. Cyan rectangles enclose currently defined cells. Click **Select** from the **SELECT FEAT** menu. Select one of the models in the cell to multiply it. The **INCR TYPE** menu appears with the following options:
   - **Outline Gap**—Enables you to enter dimensions as gaps between the cell outlines.
   - **Increment**—Uses incremental dimensions between the cells (the way ordinary patterns work) to construct the pattern.
   - **XY Pattern**—Creates a bidirectional pattern.
   - **X Pattern**—Creates a pattern along the x-axis of the Machine Csys only.
   - **Y Pattern**—Creates a pattern along the y-axis of the Machine Csys only.
   - **Fill Sheet**—Automatically calculates at regeneration time the number of instances in each direction based on specified gaps or increments, the dimensions of the workpiece, and the outline of the cell selected for nesting.
   - **Number**—Specifies the number of instances in each direction of the pattern.
4. After you select the options you want, click **Done**. You are prompted to enter values according to the selected options.
5. Specify translation along the x- and y-axes.

After you create a nesting, use the **Mod Dim** option from the **MANUFACTURE** > **Modify** menu, similar to pattern dimensions, to modify all dimensions associated with it (increments or gaps, number of instances if applicable).

Increment vs. Outline Gap



Group Operations and Workcells

You can include operation and workcell features when you create a manufacturing UDF. When you place such a group:

The default names for the operation and workcell you create in the new model are `OP_CPY###` or `MACH_CPY##`, respectively (for example, `OP_CPY010` and `MACH_CPY01`).

All the parameters and tools associated with the workcell are copied. If tools with the same IDs already exist in the new model, a message appears and the workcell tools are not copied.

The NC sequences that reference this operation and workcell in the original model automatically retain these references.

To Specify Comments for an Operation or NC Sequence

1. Select the **Oper** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Comment**. A system window appears.
3. Type the desired comment.
4. Click **OK**.

Sheet Metal Manufacturing Populate

About Sheet Metal Manufacturing Populate

The **SMM POPULATE** dialog box contains the following tabs that enable you to perform three main functions.

- **Populate**—Populates the NC sequences from the master part to all of its instances. This allows you to cut all the parts on a worksheet after creating NC sequences on the master parts.
- **Subroutine**—Subroutines the NC sequences by reducing the CL data file size by defining and dynamically selecting subroutines.

- **Order**—Orders the NC sequence output by defining an optimum flow of tools for the workpiece.

To Populate NC Sequences

The **Populate** tab, located in the **SMM POPULATE** dialog box, enables you to populate NC sequences from a single master part to all of its instances on the workpiece. You can include or exclude a part from population.

Pro/ENGINEER must be informed that an NC sequence references a model before population of an NC sequence. The populated NC sequence mimics the pattern of the reference model instances on the workpiece. This relationship is established automatically for automatically created NC sequence types. All references are used to create a reference part for manually created NC sequences.

The following functions are present in the **Populate** tabbed page:

Use	To
	Set selected NC sequences to be populated and include items to the list.
	Set selected NC sequences to be unpopulated and exclude items from the list.
	Change reference part for selected NC sequence.
	Select item from the list.
	Select item from the screen.
	Select all items in the list.

The **NC Seq** > **POPULATE** menu allows you to select one or more of the following options.

Note: A populate feature exists if a blue mark, located next to the **POPULATE** button, is present.

- **Create**—Creates a new populate feature.
- **Delete**—Deletes a populate feature.
- **Redefine**—Redefines a populate feature.
- **Suppress**—Suppresses a populate feature.

- **Resume**—Resumes a previously suppressed populate feature.
- **Info**—Obtains information about the populate feature.

The **Status** column of the list located in the **Populate** tab indicates the populate status of an NC sequence . The status of an NC sequence can be as follows:

- —To populate the NC sequence.
- —To unpopulate the NC sequence.
- —You must indicate the reference part for the populate feature to determine how to populate the sequence from one location to the entire worksheet.

If the workcell parameter `AUTO_POPULATABLE` is set to `Yes`, all newly created NC sequences are automatically included in the populated feature. The default is No. If the workcell parameter `AUTO_POPULATABLE` is set to `Yes`, the system prompts you at the time you define the NC sequence to select a reference part to use to create the tool path for all duplicate parts (if `AUTO_POPULATABLE` is `No`, then the system will issue this prompt when you include such an NC sequence in the populate feature).

There can be only one populate feature in a manufacturing process. It is automatically reordered to come after all the NC sequence features in the workpiece. When you create new NC sequences, the populate feature automatically updates to include them provided `AUTO_POPULATABLE` is set to `Yes`.

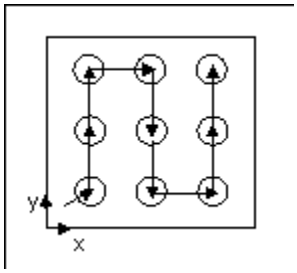
To Order NC Sequences

The **Order** tab, located in the **SMM POPULATE** dialog box, allows you to create an efficient tool-path flow over the workpiece. This reduces the overall cycle time of the operation. **Order** does not break up an NC sequence. It outputs complete sequences and subroutines in an efficient manner.

Order contains the following functions.

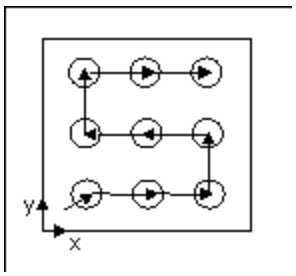
- **Origin Order**—Accept the original system supplied order.
- **Part by Par**t—Adjust the NC sequence output order.
- **Scan Order**—Allow for scan type ordering.
- **Manual Order**—Manually set the part order from a list of parts.
- **Default Order**—Accept the parts system supplied feature number order.
- **Reduce X Moves**—Incrementing the X coordinate and going back and forth in the X direction.

Reducing X Moves



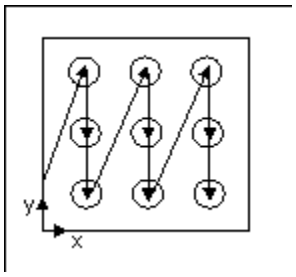
- **Reduce Y Moves**—Incrementing the Y coordinate and going back and forth in the Y direction.

Reducing Y Moves

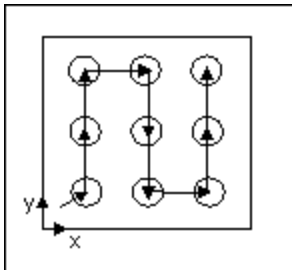


- **Tolerance**—Maximum allowable distance between scan part outline points in the reduced direction (during scan only).
- **Scan Move Direction**—Scan in one or both directions.

Scanning in One Direction



Scanning in Both Directions



- **Scan Part Outline Point**—Select the left, center, or right location on the part that is identified by the tolerance.
- **Scan Start Corner**—Set the corner of the workpiece to start the NC sequence.

To Subroutine NC Sequences

Subroutine programming allows you to create NC sequences, place them as macros at the beginning of the CL file, and then call them from the main body of the CL file as many times as needed. You can automatically create subroutines for all specified parts in the nest. This enhancement reduces the size of CL files, making them easier for the controller to handle and for the programmer to read and edit. Once you set up a process environment, individual NC sequences can be set for subroutine output.

Click the options located in the **Subroutine** tabbed page to perform the following functions.

Use	To
	Create single sequence subroutines.
	Create single sequence subroutines.
	Automatically create sequence subroutines.
	Automatically create all group subroutines.
	Exclude selected sequences and delete selected subroutines.
	Delete all subroutines.
	Expand all tree nodes.
	Collapse all tree nodes.

Also, click the following options located in the **Subroutine** tabbed page to reorder the subroutines:

- **Definition Placement**—To place the subroutine at **Top**, **Default**, or **Bottom** of the selected subroutine.
- **Enumerate**—To select all subroutines.
- **Rotation Step**—To specify the value for step rotation of subroutine.

Manufacturing Information

About Manufacturing Information

You can view manufacturing information about your model using the **SMT Manufacturing Info** dialog box. To open the dialog box, click **Info > Manufacturing**.

The **SMT Manufacturing Info** dialog box can display the following information:

- Manufacturing
- Workcell
- Operation
- NC Sequence

You can choose what information to view by selecting the desired filters and generating a report. The report, which is displayed on the screen, can be edited and saved. The file is saved in the current directory as `<manufacturename>.ppl`.

Bill of Materials

The **Bill of Materials** option in the **Info** menu generates a bill of materials (BOM) for the manufacturing model and lists all the tools added to the model. If a tool model is used, the model name, usage, and BOM for the model are output to the manufacturing BOM. The following illustration shows an example of a manufacturing BOM.

Sample Manufacturing BOM

```
Assembly MFG_SM contains:

   1      Part STOCK_2
   8      Part SMMOD_1

Summary of parts for assembly MFG_SM:

   1      Part STOCK_2
   8      Part SMMOD_1

Bill of materials for tools in manufacturing assembly:

ID    = SP1
TYPE  = STANDARD-PUNCH
-----------------------------------------
ID    = SP2
TYPE  = STANDARD-PUNCH
-----------------------------------------
ID    = WIRE1
TYPE  = CONTOURING
-----------------------------------------
```

To Generate Nesting Information

1. From the **SMT MFG MANUFACTURING** dialog box, click **Parts**.
2. Click **Automation**. The **SMT MFG AUTOMATION** dialog box opens.
3. Click **Info > Placing Info**. The **SMM AUTOMATION INFO** dialog box opens. The dialog box summarizes percent and area information.

To Create a Sheetmetal Manufacturing Report

1. Click **INFO > Manufacturing**. The **SMT Manufacturing Info** dialog box appears.
2. Select which model filters you want to include in your report.
3. Click **Apply**. The report is created.

Note: Depending upon the report, the system also generates most of the following additional information:

- **Manufacturing type (part or assembly)**—State (if any), the design model name, the workpiece name, and the date. Parts are labeled if they are either generic or an instance.
- **For each NC sequence**—The NC sequence number and the workpiece feature number associated with the NC sequence; the operation name, the workcell name and type, the number of axes, and the NC sequence type.

To Include Pre and Post Machining Files

You can use parameters to specify the pre and post files. Under the **Oper** tabbed page of the **SMT MFG MACHINING** dialog box, you can find these parameters. You can include user-defined macros, like setting the post-processor registers, at the very beginning and the very end of an NC sequence CL file, using the two parameters:

- `PRE_MACHINING_FILE`—Enter the name of the file you want to include at the very beginning of the CL file.
- `POST_MACHINING_FILE`—Enter the name of the file you want to include at the very end of the CL file.

Enter file names without the extension. If you specify the file in the `config.pro` file it should have the extension `.ncl`, or the default CL file extension.

The contents of these files are included in the CL file of the current NC sequence between `$$ –> BLOCK_START` and `$$ –> BLOCK_END`. When you use the Input option in the **CL DATA** menu to read a CL file, these contents are ignored.

Notes:

- Both of these files are included into a CL file before it goes to the post-processor.
- If these parameters are set at the operation level, pre- and post-machining files are added to each NC sequence included in the operation.

Turret Manager

About Turret Manager

The **Turret Manager** controls tools and stations of a sheet metal manufacturing workcell. To define a workcell turret, you must first create stations and then add tools to the stations.

You can access the **Turret Manager by** clicking **Wrkcell > Turret** in the **SMTMFG MACHINING** dialog box. Options in the **TURRET MANAGER** dialog box perform the following general functions.

Use	To
	Switch the turret manager dialog box between the tool and station manager environments.
	Create a new station or open the **Tool setup** dialog box.
	Create a new station or tool by copying the selected item.
	Delete a station or tool.
	Retrieve information on stations or tools.
	Customize the list of stations or tools in the dialog box.

Additional **Turret Manager** functions include the following:

- **Tool Setup**—Activate the **TOOL SETUP** dialog box for creating new tools.
- **Tool Section**—Display the cross-section of the currently selected tool.

About Station Manager Within Turret Manager

The **Station Manager**, which is part of the **Turret Manager**, is used to control stations of a workcell turret. Buttons associated with **Station Manager** perform the following general functions.

Use	To
	Switch the **Turret Setup** dialog box between tool and station setup environments.
	Load a tool in the selected station.
	Move a tool from one station to another.

	Unload a tool from a station.

The **Station Manager** consists of a menu bar and toolbar with the following options:

STATIONS

- **Create**—Create a new tool station.
- **Copy**—Copy a tool station.
- **Delete**—Delete a tool station.

UTILITIES

- **Customize List**—Customize the list of stations in the dialog box.
- **Load Tool**—Load a tool into a station.
- **Reload Tool**—Reload a tool into a station.
- **Unload Tool**—Unload a tool into a station.
- **Tool List**—Switch the **Turret Setup** dialog box between tool and station setup environments.

Additional **Station Manager** functions include the following:

- **Station**—Set the station number.
- **Indexability**—Set the station as indexable or fixed.
- **Orientation**—Set the angle of a nonindexable station.
- **Marked**—Mark the tool by adding a magenta dot by its name in the station manager. Tools automatically loaded by the system, such as Punch UDF, Form, and indexed tools are automatically marked.
- **Locked**—Protect the station from modification or deletion.
- **Tool**—Display information on the loaded tool if one exists in the station.
- **In Use**—Display if the tool is currently in use by an NC sequence.
- **Comment**—Display comments defined in the tool setup dialog.
- **Holder**—Display the current holder name and size in the station.

About Tool Manager Within Turret Manager

The **Tool Manager**, which is part of the **Turret Manager**, is used to control tools of a workcell turret. Options associated with **Tool Manager** perform the following general functions.

Use	To
	Switch the **Turret Setup** dialog box between Tool and Station setup environments.
	Access a tool catalog.
	Define a tool shape.

Define a tool shape using the following options:

- **Retrieve Shape**—Retrieves a tool parameter file.
- **Create Shape**—Creates a tool shape with Sketcher.
- **Modify Shape**—Modifies a tool shape.
- **Rename Shape**—Renames a tool shape.
- **Save Shape**—Saves a tool shape.
- **Shape Prompts—**Sets up user defined prompts for the shapes dimensions.

STATION

- **Create**—Creates a new tool.
- **Copy**—Copies the selected tool.
- **Delete**—Deletes the selected tool.

INFO

- **Where Used**—Shows a report indicating where the tool is used in the NC sequence.

UTILITIES

- **Customize List**—Customizes the information window.
- **Load Tool**—Loads a tool into a station.
- **Reload Tool**—Reloads a tool into a station.
- **Unload Tool**—Unloads a tool into a station.
- **Station List**—Opens the station setup environment.

SORT

- Sorts tool order in the list by holder, tool name, tool type, tool usage, area, station, or orientation.

TOOL CATALOG

- Searches and retrieves tools from the tool catalog.

About Tool Holder Within Turret Manager

The **Tool Holder**, which is part of the **Turret Manager**, contains information about tool holders. Click to open the **HOLDER SIZE LIST** dialog box. The following options are available.

Use	To
	UNDO last changes step by step.
	Remove the selected holder from the list.
	Edit the selected holder from the list.
	Add a new holder size to the list.
	Clear the holder size.

Tool Setup

About Tool Setup

During a typical manufacturing session, the operator selects specific tools to create the product. The tools are placed in a machine turret and are subsequently used during the NC sequence. In Pro/NC-SHEETMETAL, similarly, tools must be selected and loaded into a turret before an NC sequence is possible.

Each workcell has an associated turret, which can be configured with a variety of tools. Tools are created, edited, and selected using the **TURRET MANAGER** dialog box. The **Turret Manager** allows you to setup tools and tool stations. You can access the **TURRET MANAGER** dialog box by selecting **Turret** from the **Workcell** tabbed page. Click the **Tool Setup** check box to open the **TOOL SETUP** dialog box.

The **TOOL SETUP** dialog box contains different types of tools used in the NC sequence. Additional **Tool Setup** functions include the following:

TOOL

- **Retrieve**—Retrieve a tool parameter file.
- **Save**—Save a tool parameter file.
- **Create**—Create a tool.

CUSTOMIZE

- **Attachment**
- **Colapsible**

Tool Types

There are six types of tools available. Each tool type can be selected using the **Tool Type** menu located in the **TURRET MANAGER** dialog box. The following tools are available:

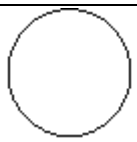
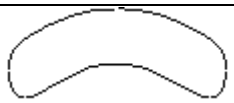
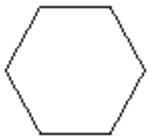
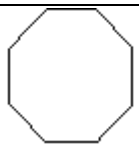
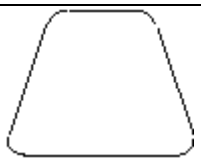
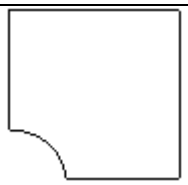
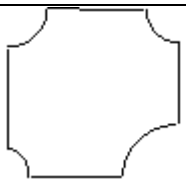
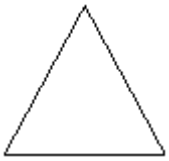
Tool Type	Description
	Use to contour laser or flame NC sequences
	Use for **Nibble Edge**, **Nibble Area**, **Shear**, and **Point Punch** NC sequences.
	Define tools automatically for Form NC sequences. At the time you define the form feature, you have an option to specify a tool name; this name then appears as the tool name in turret tables, manufacturing information. In order to supply the tool name, you must define a coordinate system, which is used as a punch location. You must create this coordinate system in the form reference part prior to the definition of the form feature.
	Use for Punch UDF NC sequences only. A Punch or Notch feature section automatically defines punch type tools.
	Use to design a Solid Punch tool in part mode for custom applications.
	Use for **Shear** NC sequences only. The **Shear** tool has a zero width and is a straight line whose length is defined by the Tool Length parameter. The dimension of the tool along the z-axis of the workpiece is specified by the Tool Height parameter.

Standard Punch Tool Shapes

The standard punch tool is located in the **TOOL SETUP** dialog box. To access the **TOOL SETUP** dialog box, click **Wrkcell > Turret** then click the **Tool Setup** check box. The standard punch tool is available in a variety of shapes. Modify tool shapes by typing dimensions in text boxes.

Click the **Shape** drop-down list box located in the **TOOL SETUP** dialog box to see the following available tool shapes:

Standard Punch Tool Shapes

Round		Rectangular	
Obround		Boxround	
Banana		Hexagon	
Octagon		Trapazoid	
Corner1r		Corner4r	
Triangle		Click the **Tool Section** check box to display the cross-section of the currently selected tool.	

To Create a New Tool

1. Select the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Turret**. The **TURRET MANAGER** dialog box opens.
3. Click  to open the tool manager environment.
4. Click **Tool > Create** or the **Tool Setup** check box. The **TOOL SETUP** dialog box opens.
5. Select the desired tool type and parameters.
6. Click **Done**.

Note: You must save the tool parameter file if you want to retrieve the tool later. A new tool can be stored in either the tools library, if it is set, or in your current working directory.

To Edit a Tool

1. Click the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Turret**. The **TURRET MANAGER** dialog box opens.
3. Click  to open the tool manager environment.
4. Click the **Tool Setup** check box. The **TOOL SETUP** dialog box opens.
5. Retrieve the desired tool if necessary.
6. Select the desired tool in the information window.
7. Edit the tool parameters in the **Tool Setup** dialog box..

Note: You must save the tool parameter file if you want to retrieve the tool later.

To Save Tools

1. Click the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Turret**. The **TURRET MANAGER** dialog box opens.
3. Click  to open the tool manager environment.
4. Click the **Tool Setup** check box. The **TOOL SETUP** dialog box opens.
5. Select a tool. Edit the tool parameters as desired.
6. Click **Tool > Save**.

   The tool is stored in the tools library, if it is set, or in your current working directory, in a file named `<tool_id>.tpm`. The tool can be used in any manufacturing model.

Note: Prior to Release 12.0, the default extension for the tool parameter files was .tprm. Old files with this extension are recognized by the system as tool parameter files, and they are retrievable. Whenever a tool parameter file is stored, however, it has the .tpm extension.

To Retrieve Tools

1. Click the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Turret**. The **TURRET MANAGER** dialog box opens.
3. Click  to open the tool manager environment.
4. Click the **Tool Setup** check box. The **TOOL SETUP** dialog box opens.
5. Click **Tool > Retrieve**. The **Open** dialog box appears.
6. Select the desired tool and click **Open**.

To Delete a Tool

1. Click the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Turret**. The **TURRET MANAGER** dialog box opens.
3. Click  to open the tool manager environment.
4. Click the **Tool Setup** check box. The **TOOL SETUP** dialog box opens.
5. Select the tool to be deleted from the information window.
6. Click  . The tool is deleted from the turret.

Note: Deleting a tool removes it from the turret. It does not, however, erase the tool file.

Tool Parameters

Each tool has a corresponding set of parameters, listed in the **Turret Setup** dialog box, which define its geometry. The tool schematic shows the form and dimensions of the tool. Click **Show** to display the tool. The following parameters are available:

Note: Some parameters are not available for specific tools.

- **Corner Radius**—The corner radius of the tool. The default tool section has a rectangular shape defined by the Tool Width and Tool Length parameters. A Corner Radius value that is less than both Tool Width/2 and Tool Length/2, will generate a tool that has a rectangular shape with rounded corners. A Corner Radius value that is equal to the Tool Width/2 with Tool Width lesser than the Tool Length, will generate an obround.
- **Tool Length**—The length of the tool along the x-axis, or along the edge for Nibble Edge NC sequences.
- **Tool Width**—The width of the tool along the y-axis, or normal to the edge for Nibble Edge NC sequences.
- **Tool Height**—The height of the tool along the z-axis of the workpiece coordinate system.
- **Units**—The units of the tool. Inch, foot, millimeter, and centimeter units are available.
- **Comment**—Optional comments. A text string stored with the tool parameters. This text is for information only.
- **Name**—The tool's name. The tool parameters are stored in the file `<tool_name>.tpm`, which is used throughout the manufacturing process to identify the tool uniquely. The name must be less than `32` alpha numeric characters long.
- **Type**—The type of tool. Standard Punch, UDF Punch, Form, Shear, Contour, and Solid Punch tools are available.
- **Station**—The tool location in the turret.

- **Indexability**—Specifies if the tool is indexable.
- **Holder Size**—Specifies the radius of the tool holder.

Tool Library

You can set up the tool library directory to store manufacturing tools. The configuration file option is:

```
pro_mf_tprm_dir <pathname>
```

Note: To avoid problems, use the complete pathname.

Using Library Parts

If you use a Solid Punch for a Point Punch NC sequence, specify it using a tool model (i.e., a Pro/ENGINEER part), with the following restrictions:

You must create the tool part as a single extruded protrusion. This should be the first feature. The section of this protrusion (2D) is copied into the manufacturing and used for tool simulation.

This section must have one coordinate system (a sketcher coordinate system, not a coordinate system feature). Use this coordinate system to place the tool at each datum point you select.

Tool Path

About Tool Path

Cutter Location (CL) data files are generated from the tool paths specified within Sheet Metal Manufacturing NC sequences. Each NC sequence generates a separate tool path.

To Display the Tool Path

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select an NC sequence.
3. Click **CL Output**. The **SMT MFG NCL PLAYER** appears.
4. Click **Play** to view the tool path.
5. To set the delay, click **OPTIONS > Set Delay**. The **PLAY SPEED** dialog box appears.
6. Move the slider to set the delay.

Notes:

- When displaying the tool path, the dotted line represents the geometrical center of the tool. In some cases, the display may not reflect the exact coordinates in the generated Pro/CLfile.
- The configuration option `cl_arrow_scale` enables you to control the size of the tool path arrow for Contouring NC sequences. The default is `1`. If you set it to `0`, the arrows do not display. Specifying any other positive number scales the arrow accordingly.
- Click **Stop** to abort the cutter path display at any time.

To Adjust the Tool Path

1. From the **NC Seq** tabbed page, select the desired NC sequence.
2. Click **OPERATE > Redefine**. The **SMT NC SEQUENCE** dialog box appears.
3. To change the order of the strokes/hits in the tool path, select the **Change Order** element and click **Define**. The **ORDER HITS** dialog box appears with the following options:
   a. Under **Start Point**, select one of the following:
   - **Select**—Select the first hit within the NC sequence tool path and specify the direction using the **TOOL PATH DIR** menu. Select any hit, including those in the middle of the tool path, as the start point.
   - **Delete**—Delete a defined starting point.

   b. Select the **Sort by Next Closest** to automatically select the next closest edge to be machined after completing the previous edge.

   c. Under **Hit stroke(s)**, select one of the following:
   - **Reorder**—Select strokes to indicate the order in which they should be output.
   - **Order by Default**—Revert to the default order of strokes
   - **Reverse**—Select strokes to reverse the order of hits within a stroke.
   - **Unreverse Last**—Undo last reversed order of hits within a stroke.
4. To remove hits, select the **Remove Hits** element and click **Define**. The **REMOVE HITS** dialog box appears with the following options:
   - **Single**—Removes individual hits.
   - **From To**—Removes a series of hits.
   - **Restore Last**—Restores the last deleted hit(s).
5. To insert CL commands at specific hits, select the **CL Command** element and click **Define**. Insert CL commands as desired.
6. To finish the tool path, click **Done** from the **SMT NC SEQUENCE** dialog box.

To Optimize the Tool Path Manually

Any portions of the tool path that are not automatically optimized can be optimized manually using the Adjust Path functionality.

1. From the **NC Seq** tabbed page, click **Optimize > Redefine**. The **SMM Optimize** dialog box opens.
2. Select a set to optimize from the **Opsets** list.
3. Click **Adjust Path**. The following options are available:
    - **Remove Duplicate Hits**—Remove duplicated hits from the tool path.
    - **Remove Duplicate Cut Lines** — Remove duplicate cut lines from the tool path.
    - **Remove hits**—Removes hits from the tool path using the **REMOVE HITS** dialog box.
    - **Reverse NC Seq**—Select NC sequence tool paths to reverse direction of the whole path.
    - **CL Command**—Opens the **SMM CL COMMAND** dialog box. This enables you to insert CL commands at selected hits.
4. From the **SMM Optimize** dialog box, click **OK**.

When the Tool Path Will and Will Not Be Recomputed

The tool path recomputes if one of the following situations occur:

- Manufacturing or tool parameters that affect the tool path (i.e., computation of the `GOTO` points on the tool path) are changed. For example, `OVERLAP_DIST` affects the tool path computation, while `CUT_FEED` does not.
- NC sequence setup references (**Edge**, **Shake Away**) are newly defined or changed.
- Dimensions associated with the NC sequence reference items (surfaces, edges) are changed.
- Cut motion dimensions are changed.
- A cut is defined for interactive cut motion.
- Nibble edge or cut motion ends are adjusted.
- Redo is used for corner conditions.
- Tool path for cut motions is recomputed if the parent NC sequence is changed.

Unless any of the changes listed have been made, the tool path is not recomputed if the tool path storage is enabled.

When the tool path is recomputed, the new data is appended to the tool path file, so the file grows with every tool path computation. When the model regenerates, the

system performs "purging" of the tool path file and removes old (invalid) tool paths, to reduce the size of the tool path file.

Adjusting the Tool Path

This functionality is applicable to Punch Press type NC sequences only.

The tool path for any Punch Press NC sequence consists of a series of hits. Hits indicate each location where the tool makes contact with the workpiece and removes material; they appear as small red circles. For Nibble Edge and Nibble Area NC sequences, a succession of hits composes a stroke, which is the other element of the tool path. A stoke is a succession of hits along an edge of a reference model, or in a line across an area. Strokes appear as a series of hits connected by a line; the first hit in the stroke displays as a small, filled red circle.

Machine Zones and Clamps

About Machine Zones and Clamps

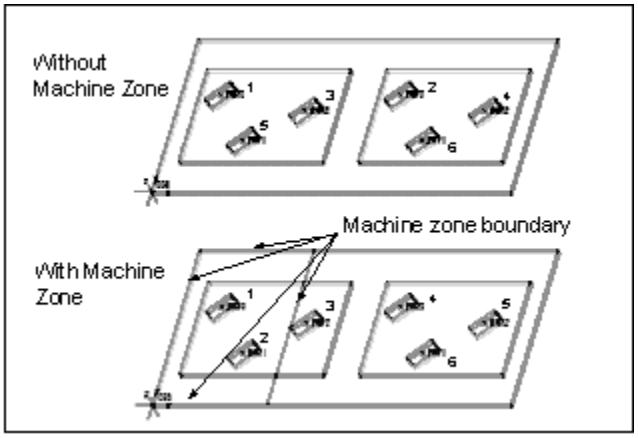
At setup time, you can define machine zones and locate clamps in a sheet metal manufacturing model. A machine zone is the area that can be machined without repositioning the workpiece. The workpiece can be repositioned in one direction only, along either the x-axis or the y-axis of the machine coordinate system. If you want to switch the direction of repositioning, delete the existing machine zone definition and create a new one. You can also redefine an existing machine zone.

CL Output

Machine zones and clamps affect the CL output. When you create an NC sequence with a machining zone defined and clamps positioned:

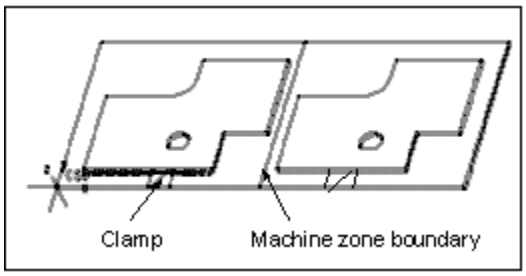
The order of CL data generation is changed. All geometry inside the zone is machined first; then the zone, together with the clamps, is translated along the appropriate direction and all geometry inside its new location is machined. The following illustration shows the order of punching with and without the machining zone. If an edge or area to be nibbled crosses the zone boundary, CL data is also generated only for the part inside the boundary; then, as the machine zone is translated, the rest of the edge/area is machined.

Order of CL Generation



Clamps are avoided in cutter location output. The minimum distance between the center of the tool and the clamp boundary `= clearance - distance + (1/2 tool width diameter)`.

Avoiding Clamps



To Define Machine Zones

1. Click **Wrkcell** in the **SMT MFG MACHINING** dialog box.
2. Select the workcell with which you want to associate the machine zone.
3. Click **Zones > Create**. The **MACHINE ZONE** dialog box opens. Select one or more of the following options:
    - **Zone Csys**—Enables you to define or change the machine coordinate system.
    - **Translate Direction**—Specify the direction in which the workpiece is repositioned.
    - **Workpiece Size**—Displays the size of the workpiece.
    - **Zone Size**—Enables you to customize the size of the zone by X and Y axis.
    - **Zone's Offsets**—Offsets the machine zone.
    - **Automatic Adjusting**—Adjusts the size of a machine zone if it does not fit on a workpiece.

- **Automatic Reposition**—Determines if the workpiece can be repositioned automatically to machine-omitted hits (under clamps).
- **Zone Intersection**—Click this check box to specify zone offsets.
- **Preview**—Enables you to preview the feature before it is actually created.

4. Select either **X Axis** or **Y Axis** from **Translate Direction** to specify the direction in which the workpiece is repositioned.
5. Type the zone size in the **Zone Size** box. This is the zone dimension along the selected axis. Along the other axis, the zone extends the entire size of the workpiece.

   When first created, all zones have the same size. Pro/ENGINEER creates as many zones as necessary to cover the whole workpiece along the selected axis.
6. To change the zone size, change the value in the customized **Zone Size** box.

   Whenever you modify the zone size, Pro/ENGINEER recalculates the number of zones necessary to cover the whole workpiece.
7. Click **Done**.

Creating Clamps

A clamp section may consist of only a single closed loop of sketched entities. Its location is defined as X and Y offsets from the machine zone origin. You can also specify a rotation angle.

There are two valid types of clamps: **Machine Clamp**, and **Reposition Pad**. Use machine clamps to calculate avoidance and reposition clamps for visualization purposes.

The **Zones** drop-down menu contains the following options:

- **Create**—Create the machine zones.
- **Modify**—Modify the selected clamp.
- **Delete**—Delete all the machine zones.

To Create a Clamp

1. Select the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Zones > Clamp > Create Clamp**. The **MACHINE CLAMP** dialog box opens.
3. Click **Sketch** to define a new clamp or **Retrieve** to retrieve an existing clamp. If clamps are already present in the manufacturing model, you can select them by clicking **Use Prev**.
4. If you want to modify the dimensions of the clamp, click **Mod Dim**. The system displays the clamp dimensions. Select the dimension you want to modify. Type the new value and click **Done Sel**.

5. You can position a clamp using the following options:

- **X Offset**—Enter a value for the x offset from Machine Csys. (Press <CR> after typing in the value.)
- **Y Offset**—Enter a value for the y offset. (Press <CR> after typing in the value.)
- **Angle**—Enter the angle of rotation of the clamp about the anchor point. (Press <CR> after typing in the value.)
- **Drag**—Place the clamp using the following mouse buttons:
  - **LEFT**—Finalizes the placement of the clamp.
  - **MIDDLE**—Pauses. The **LinearIncr** and **AngularIncr** options change drag increments. The effect is similar to the grid snap in Sketcher (e.g., you might want to enter a rotational increment of `90`). A drag increment of `0`, at any point, is equivalent to "grid snap off." Click **Done** to continue the placement process. To bring the clamp back to the **Machine Csys** location, click **Quit**.
  - **RIGHT**—Switches between dragging and rotating the clamp.
- **Clearance Distance**—Define the amount of clearance between a toolholder and a clamp (clearance distance + 1/2 length width diameter = distance of CL from clamp edge).

6. Click **Done**.

To Create a Reposition Pad

1. Select the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Zones > Pad > Create Pad**. The **REPOSITION PAD** dialog box opens.
3. Click **Sketch** to define a new pad or **Retrieve** to retrieve an existing pad.
4. If you want to modify the dimensions of the pad, click **Mod Dim**. The system displays the pad dimensions. Select the dimension you want to modify. Type the new value and click **Done Sel**.
5. You can position a pad using the following options:

- **X Offset**—Enter a value for the x offset from Machine Csys. (Press `<CR>` after typing in the value.)
- **Y Offset**—Enter a value for the y offset. (Press `<CR>` after typing in the value.)
- **Angle**—Enter the angle of rotation of the pad about the anchor point. (Press `<CR>` after typing in the value.)
- **Drag**—Place the pad using the mouse buttons as follows:
  - **LEFT**—Finalizes the placement of the pad.
  - **MIDDLE**—Pauses. The **LinearIncr** and **AngularIncr** options change drag increments. The effect is similar to the grid snap in Sketcher (e.g., you

might want to enter a rotational increment of 90). A drag increment of 0, at any point, is equivalent to "grid snap off." Click **Done** to continue the placement process. To bring the pad back to the **Machine Csys** location, click **Quit**.

- **RIGHT**—Switches between dragging and rotating the pad.

6. Click **Done**.

To Create a Cosmetic Section

1. Select the **Wrkcell** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Zones > Cosmetic > Create Cosmetic**. The **COSMETIC SECTION** dialog box opens.
3. Click **Sketch** to define a new cosmetic or **Retrieve** to retrieve an existing cosmetic.
4. You can position a cosmetic using the following options:

- **X Offset**—Enter a value for the x offset from Machine Csys. (Press <CR> after typing in the value.)
- **Y Offset**—Enter a value for the y offset. (Press <CR> after typing in the value.)
- **Angle**—Enter the angle of rotation of the pad about the anchor point. (Press <CR> after typing in the value.)
- **Drag**—Place the pad using the mouse buttons as follows:
  - **LEFT**—Finalizes the placement of the pad.
  - **MIDDLE**—Pauses. The **LinearIncr** and **AngularIncr** options change drag increments. The effect is similar to the grid snap in Sketcher (e.g., you might want to enter a rotational increment of 90). A drag increment of 0, at any point, is equivalent to "grid snap off." Click **Done** to continue the placement process. To bring the pad back to the **Machine Csys** location, click **Quit**.
  - **RIGHT**—Switches between dragging and rotating the pad.

5. Click **Done**.

Nesting

About Nesting Reference Models

Nesting functionality enables you to add, move, and pattern reference parts in a Sheet Metal manufacturing model.

Nesting is based on cells. A cell consists of one or more reference parts. A cyan rectangle encloses the reference part(s) to make it easily recognizable during nesting procedures. Only reference parts you add during the nesting process are in a cell.

When you select **Nesting** from the **Parts** tabbed page, the following commands are available:

- **Create**—Define a new cell for nesting reference parts.
- **Redefine**—Redefine an existing cell by adding, moving, or deleting reference parts.
- **Delete**—Delete an existing cell. You can remove all parts included in the cell from the manufacturing model. The system prompts you to confirm your selection if you choose this command.
- **Blank**—Remove a cell from the display of the manufacturing model.
- **Unblank**—Redisplay a blanked cell. If more than one cell is blanked, the CHOOSE menu appears. The CHOOSE menu includes the commands **Accept**, **Next**, **Previous**, and **Quit**. Use one of these commands to select the cell(s) that you want to unblank.
- **Multiply**—Create a pattern of cells.
- **Modify Dims**—Enable you to modify all dimensions, including drag distances and angles, as well as nesting gaps and increments.
- **Info**—Provide information on the total area of sheet metal, total area of nested parts, wasted area, and percentage of wasted sheet over total sheet surface. It also lists the names of the reference parts, and how many parts of this name are currently placed on the sheet.

To Create an MFG Model

1. Click **Parts** tabbed page from the **SMT MFG MACHINING** dialog box. Click **Automation**.
2. From the **SMT MFG AUTOMATION** dialog box, click **FILE > Load**. The **LOAD SETUP** dialog box opens. Select the attributes you want to load and click **OK**. The **Open** dialog box opens.
3. Locate the desired `.dat` file and click **Open**.
4. Click **Done**. The **Make SMT MFG** dialog box opens.
5. Select either **Single** or **Multiple** sheets, and **Save Remainder File**, if desired.
6. Click **OK**. The **MFG** model is created.

To Reference Parts from Existing MFG Models

1. From the **Parts** tabbed page on the **SMT MFG MACHINING** dialog box, click **Automation**. The **SMT MFG AUTOMATION** dialog box appears.
2. From the **Order** tabbed page, click **Load > Parts From MFG**. The **Open** dialog box opens.

3. Select the manufacturing file you want and click **Open**. The manufacturing model appears in a window.
4. Select the part you want from the part list. The chosen part is highlighted in the window.
5. Click **OK**. The manufacturing model and the chosen reference part appear in the **Order** tab page window.
6. If you are satisfied with the default values, click **Preview**. The system places the parts into the workpiece.

Note: The default offset and number of instances are defined in the **Defaults** tabbed page.

To Load and Set Parts

1. From the **SMT MFG AUTOMATION** dialog box, select the **Order** tabbed page.
2. Click **LOAD**. The following options are available:
   - **Parts from MFG**—Load parts from an existing manufacturing model.
   - **Cell of Parts**—Load previously nested parts.
   - **Part w/o NC seqs**—Load parts without defined NC sequences.
   - **DXF File**—Load a DXF file.
3. Select the desired file from the **Open** dialog box. You can view the part by clicking **Show**.
4. Click **Setup**. The **Part Setup** dialog box opens.
5. Modify the part setup as desired.

To Define a New Cell

1. Select the **Parts** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Click **Create** from the **NESTING** menu.
3. Click **Add Part** from the **NEST CELL** menu.
4. The **Open** dialog box appears. Select the part.
5. The **PART PLACE** menu appears with the following options:

- **DragOrigin**—Create or select a coordinate system on the reference part (the reference part is displayed in a sub-window). All Nesting data is with respect to this origin.
- **LinearIncr**—Enable you to enter a value for the linear drag increment.
- **AngularIncr**—Enable you to enter a value for the rotational drag increment.

Specifying the linear and angular drag increments is optional. The effect is similar to the grid snap in Sketcher (e.g., you might want to enter the rotational increment of `90`). Entering a drag increment of `0`, at any point, is equivalent to "grid snap off."

6. The highlighted contour of the reference part starts following the cursor as you move it. The tip of the cursor corresponds to the position of the coordinate system origin. Place the part using the mouse buttons:

- **LEFT**—Finalizes the placement of the part.
- **MIDDLE**—Pauses. The **LinearIncr** and **AngularIncr** options are available for you to change the drag increments. You cannot, however, change the drag origin at this point. Click **Done** to continue the placement process. To quit the whole process of adding a part, click **Quit**.
- **RIGHT**—Switches between dragging and rotating the part.

To Retrieve a Setup File

1. From the **Parts** tabbed page of the **SMT MFG MACHINING** dialog box, click **Automation**. The **SMT AUTOMATION** dialog box appears.
2. From the **SMT MFG AUTOMATION** dialog box, click **FILE** > **Load**. The **LOAD SETUP** dialog box opens.
3. Select the attributes you want to load and click **OK**. The **OPEN** dialog box opens.
4. Locate the desired `.dat` file and click **Open**.

To Set the Workpiece Size

1. From the **SMT MFG AUTOMATION** dialog box, select the **WP Size** tabbed page.
2. Set the **Workpiece Size**, **Border Size**, and **Grain Direction** options as desired.
3. Click **Done**.

To Modify Automation Options

In the **SMT MFG AUTOMATION** dialog box, click **Options**. Select one or more of the following options:

- **Display Options**—Opens the **SHOW SETUP** dialog box. Select the required display option.
- **Warning Selection**—Opens the **WARNING SELECTION** dialog box. This dialog box enables you to select specific error messages for checking incoming parts. Select one or more of the following messages:
    - **Thickness**—Part thickness is different from the workpiece thickness.
    - **NC Seq Type**—Sequences associated with the part are incompatible with the chosen cell type.

  - **Parts with Absent Tools**—The part cannot be manufactured with the selected tools.
  - **NC Seqs with Absent Tools**—The NC Sequence cannot be manufactured with the selected tools.
  - **Indexable Tools**—Tool not available in turret or indexable station.
  - **Tool Type**—Incompatible tool type selected.
  - **NC Seq with Ext Refs**—NC and data part references are not available.
  - **Part Size**—Selected part is larger than the workpiece.
- **Load/Save Items**—Opens the **SAVE/LOAD ITEMS** dialog box. Select the required items to load or save.
- **List Columns**—Enables you to customize field columns using the **CUSTOMIZE LIST** dialog box.

To Generate Nesting Information

1. From the **SMT MFG MANUFACTURING** dialog box, click **Parts**.
2. Click **Automation**. The **SMT MFG AUTOMATION** dialog box opens.
3. Click **Info > Placing Info**. The **SMM AUTOMATION INFO** dialog box opens. The dialog box summarizes percent and area information.

Autonesting

About Autonesting

Autonesting is an advanced system of sheet metal nesting. It provides dynamic packaging capabilities, which allow you to package sheet metal reference parts (along with existing NC sequences, if desired) and cells of parts from existing manufacturing models automatically.

You can easily change the number of the nested instances, customize the offsets between the models, and select nesting algorithms to create an ideal balance of performance and packaging optimization.

SMT MFG Automation Dialog Box

The **SMT MFG AUTOMATION** dialog box controls the entire autonesting process. It consists of a menu bar, 4 tabbed pages, and 3 buttons.

You can access the **SMT MFG** dialog box by clicking **Automation** on the **Parts** tabbed page.

Buttons

There are 3 buttons available in the dialog box:

- **Done**—Complete the autonesting process and create the new manufacturing model.
- **Cancel**—Quit the autonesting process.
- **Preview**—Preview the nest.

To Set Autonesting Defaults

1. From the **Parts** tabbed page on the **SMT MFG MACHINING** dialog box, click **Automation**. The **SMT MFG AUTOMATION** dialog box appears.
2. From the **SMT MFG AUTOMATION** dialog box, select the **Defaults** tabbed page. The following options are available:

- **Quantity**—Determine the quantity of parts.
- **Grain Alignment** (check-box)—Align the part to the grain.
- **Flip Allowance** (check-box)—Allow part flip.
- **Offsets**—Click one of the following options:
    - **Outer**—Set to the default profiling tool width.
    - **Inner**—Set to the inside profiling tool width.
- **Rotation Angle**—click one of the following options:
    - **Any**—All angles are valid.
    - **Step**—Rotate the model in 90 degree increments, or steps. The default step is 90 degrees.
- **Priority Weights**—Sets the default priority weights.
- **Clustering** (check-box)—Invokes True Shape nesting
- **Max Length**—Maximum length of the cluster. The machine zone size is the default value.
- **Max Width**—Maximum width of the cluster. The machine zone size is the default value.
- **Repeat Factor**—Specify the amount of time to optimize the cluster.
- **Use Holes** (check-box)—Invokes part-in-part nesting.
- **Minimal Hole Area**—The minimum hole size for part-in-part nesting.

3. Modify the defaults as desired.

Part Order Control

About Part Order Control

Part order control is exclusive to the autonesting function. You can prioritize the order of nesting for specific parts selected in the order list. Part order control is especially useful during high volume production when large part quantities are involved. Specified quantities of particular parts are prioritized to control the nesting of all parts. You can assign different priorities for the same part to minimize or avoid nesting.

Prioritizing the nesting of parts is determined by a numbering scale from `0` to `100`. The least priority is `0` and the most is `100`.

To Control Part Nesting Order

Perform the following steps to control the priority of nested parts.

1. After you nest parts onto your sheet metal workpiece using the Autonesting function, click the **Order** tabbed page from the **SMT MFG AUTOMATION** dialog box.
2. Type a priority number in the order list under the **Prty** column or double-click the part name to open the **Part Setup** dialog box. Type the desired priority number and count.

   Note: Right-click part name and click **Copy**. Two rows are added under the part name. You can specify multiple priorities and subquantities for the part.
3. Determine the priority weight.
4. Click **Done**. The **MAKE SMT MFG** dialog box opens.
5. Select single or multiple sheets. Click **OK**.

To Control the Priority Weight

Use **Priority Weight** to adjust the effect of the priority value to the overall production. The priority numbering scale for priority weight is in `.01` increments from `0` to `1`. Use the number `0` to turn the priority off and `1` to use priorities as specified in the order list. Values between `0` and `1` consider material utilization as well as the specified priority.

1. After you nest parts onto your sheet metal workpiece using the Autonesting function, click the **Defaults** tabbed page from the **SMT MFG AUTOMATION** dialog box.
2. Type the priority weight value in the **Priority Weight** text box or drag the rotation wheel to the desired value.
3. Click **Done**.

NC Sequencing

About NC Sequences

An NC sequence is a workpiece feature that represents a single tool path. You must set up a process environment before creating an NC sequence. The type of workcell defines the types of NC sequences available, as shown in the following table.

NC Sequence	Punch	Laser	Laser-Punch	Flame	Flame-Punch
		X	X	X	X
		X	X	X	X
	X		X		X
	X		X		X
	X		X		X
	X		X		X
	X		X		X
	X		X		X
	X		X		X
	X		X		X

When you create an NC sequence, a dialog box corresponding to the NC sequence type is displayed. Each of these dialog boxes has the following options:

- **Parameter**—Open the parameter tree.
- **Comment**—Type comments regarding NC sequences.
- **Define**—Specify the tool, parameters, and geometric references. You can also apply some low-level control depending on the NC sequence type.
- **Info**—Display parameter and NC sequence information.
- **Preview**—Display the tool path for the NC sequence prior to completion of the NC sequence. Available after all elements have been defined.
- **Done**—Completes creation of the current NC sequence.
- **Cancel**—Terminates the creation of the current NC sequence after confirmation.
- **Next**—Completes the current NC sequence and starts creating another Nibble Edge NC sequence with the same tool and parameters.

To Create a Reference Manufacturing Pattern

1. From the **SMT MFG MACHINING** dialog box, click **UTILITIES > Feature > Pattern>Create**. The **SELECT FEAT** menu appears.
2. Click one of the following:

- **Select**—Select a feature from the Pro/ENGINEER graphics window.
- **Operation**—The **SEL MENU** menu appears. Select the required operation.
- **NC Sequence**—The **NC SEQ LIST** menu appears. Select the required NC Sequence. The **PRO PAT TYPE** menu appears. Select one of the following:
    - **Csys Pattern**—The **PAT TYPE** menu appears. Specify values for **Translate** and **Rotate**.
    - **Dim Pattern**—Select a feature from the Pro/ENGINEER graphics window.

The reference manufacturing pattern is automatically generated.

About Auto Tool NC Sequences

The **Auto** menu in the **NC Seq** tabbed page enables you to create punch, form, and contouring NC sequences automatically. The following options are available:

- **Punch UDF**—Automatically creates Punch NC sequences to machine all the punch/notch features present in the reference part(s).
- **Forming**—Automatically creates Form NC sequences to machine all the form features present in the reference part(s).
- **Tool Shape**—Automatically creates Punch NC sequences by matching the tool shape of the Standard Punch tools in the Turret with reference part geometry (holes and slots). Partial matching on the outside contour of the reference part

can also be performed; for example, if the tool size matches the radius of a semi-circular notch, this notch is included in the Auto Tool sequence.

- **Nibbling**— Automatically selects all edges for nibbling sequencing form NC Sequences. You can setup multi-tool nibbling and modify individual nibbling sequences created from the sequence group.
- **Contouring**—Automatically creates Form NC sequences to machine all the Contour features present in the reference part(s). The contouring toolpath is based on part profiles and workcell parameters.
- **Approach Punch**—Automatically creates Form NC sequences to machine all the Approach Punch features present in the reference part(s).

If there are no Punch/Notch or Form features present, specific Tool options will be unavailable.

Unlike a regular Tool Shape NC sequence, which matches the tool shape to all part geometry, the Auto Tool Shape punches holes and slots, but does not machine contours created as a Punch or Notch feature. This ensures that there is no overlapping between the automatically created Punch NC sequences.

About Manufacturing User-Defined Features (Groups)

This functionality enables you to create the NC sequences necessary to manufacture a group of features (UDF) only once. You can then group NC sequences and place them in any other models that contain the same reference features' group.

If you do not reference a design UDF, the information about the reference part is stored with the group. If you then place the group in another manufacturing model that contain the same reference part, you have an option to automatically resolve the placement references.

About Auxiliary NC Sequences

Auxiliary NC sequences produce a point-to-point tool path. They can be used to specify the connecting tool motions and change the tool axis orientation, if needed, between two machining NC sequences. They also allow you to access the on-machine probe functionality, and are available for any workcell.

Note: You do not have to specify a tool for an Auxiliary NC sequence. You can create Tool Motions even though no tool is specified.

To Create an Auxiliary NC Sequence

1. Click the **NC Seq** tabbed page from the **SMT MFG MACHINING** dialog box.
2. Select the NC sequence with which you want to associate the auxiliary NC sequence.
3. Click **New > Auxiliary**. Click **Contouring** or **Punching**.
4. The **SMT NC SEQUENCE** dialog box opens.

5. Click **Tool** and **Define**. From the **TURRET SETUP** dialog box, define the tool and click **Done**.
6. Click **Define Path** and **Define**. The **AUXILIARY SETUP** dialog box opens.
   a. Under **Create**, you can select one of the following options:
   - **Sketch**—To sketch the defined path.
   - **Go Home**—To define the go home point as the defined path.
   - **Goto Pnt**—To define the go to point.
   - **Go Delta**—To define the x and y offset for the defined path.

   b. If desired, you can check the **Insert Before** check box to insert the new defined path before the last defined path.

   c. Under **Modify**, you can select one of the following options:
   - **Dims**—To modify the dimensions of the previously defined paths.
   - **Delete**—To delete a defined path.
7. To use the any of the optional elements, click the optional element you want, then click **Define**.
8. The following options are also available:
   - **Name**—Name the NC sequence.
   - **Parameters**—Edit the NC sequence parameters.
   - **Comment**—Add a comment.
   - **Info**—Displays NC sequence and parameter information.
   - **Next**—Go to the next auxiliary NC sequence.
   - **Preview**—View the tool motion using the **SMT MFG NCL PLAYER**.
9. Click **Done**. The auxiliary NC sequence is created.

Modifying NC Sequences

About Modifying NC Sequences

NC sequences and material removal features, like any workpiece or assembly feature, can be deleted, suppressed, and resumed etc.

To Redefine NC Sequences

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select the NC sequence to redefine.
3. Choose **OPERATE > Redefine**. The **SMT NC SEQUENCE** dialog box appears.

4. Select the item you want to redefine. Click **Define**. Modify as desired.
5. Click **Done**.

Note: An NC sequence cannot be redefined if it has been patterned or has children (e.g., material removal). Both must be must be deleted first.

To Reorder NC Sequences

1. From the **SMT MFG MACHINING** dialog box, click **NC Seq**.
2. Select the NC sequence to reorder.
3. Click **OPERATE > Reorder**.
4. Use  or  to move the NC sequence up or down, respectively.
5. Click  .

Note: The **Populate** feature is automatically reordered to come after the last NC sequence.

To Pattern an NC Sequence

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select the NC sequence to be patterned.
3. Click **OPERATE > Pattern > Create**. The **PAT TYPE** menu appears.
4. Click **Rotate** or **Translate** to select patterning type in first direction.

   If you click **Translate**, click **Enter** from the **ENTER VAL** menu and enter displacement in the X-direction. If you do not want a displacement in this direction, choose `0.000000` from the **ENTER VAL** menu. Repeat for the Y-direction.

   If you click **Rotate**, click **Enter** from the **ENTER VAL** menu and enter rotation angle.
5. Click **Enter** from the **ENTER VAL** menu and enter total number of instances in this direction.
6. Select the patterning type in second direction, or if you want a unidirectional pattern, click **None**. If you want a bidirectional pattern, repeat actions in steps `4` and `5` for the second pattern direction.
7. Click **Quit** from the **PAT TYPE** menu to abort the patternization of an NC sequence about a coordinate system.

Notes:

- Patterning in a second direction uses the same coordinate system as specified for the first direction.

- If you select an NC sequence for patterning that already has a pattern defined, an error message appears. Delete the existing pattern and then define a new pattern.
- Coordinate patterns can reference either the NC Sequence or the Machine coordinate system. They can be unidirectional or bidirectional. For each direction you can specify either displacement along the `X- and Y-axes`, or rotation about Z-axis.

To Reorder Sequences

You can reorder output sequences and group them by NC sequence name, tool, or reference part.

1. From the NC Seq tabbed page, click **Optimize > Redefine**. The **SMM Optimize** dialog box opens.
2. Select a set to optimize from the **Optsets** window.
3. Click **Order**. The following options are available:
   - **Manual Order**—Specify the order of performing the NC sequences manually in the **SMT NC SEQ ORDER** dialog box.
   - **Part Order**—Specify the order of reference parts in the **SMM PART ORDER** dialog box.
   - **Type Priority**—Specify the order based on types in the **SMM TYPE ORDER** dialog box.
   - **Tool Change**—Specify the order of tools in the **SMM TOOL ORDER** dialog box.
   - **Tool Travel**—Specify the order of travels in the **SMM TRAVEL ORDER** dialog box.
4. Depending on the option selected, NC sequence names, tool names, or reference part names (including all the nested instances) are listed in an editor window of the dialog box. Use the following options to reorder them as desired:
   -  —Move the name selected in the editor window up by one position.
   -  —Move the name selected in the editor window down by one position.
   -  —Specify the new position for the name selected in the editor window . If you move to a lower position on the list, the selected name is placed *after* the new position picked; if you move to a position higher on the list, the selected name is placed *before* the position picked.
   -  —Specify a new position for the name selected in the editor window by entering a new position number.
5. Click **OK**.

6. From the **SMM Optimize** dialog box, click **OK**. The sequence is reordered.

To Reorder Sets

1. From the **NC Seq** tabbed page, click **Optimize > Redefine**. The **SMM OPTIMIZE** dialog box opens.
2. From the **Optsets** window, select the set to move. Move the set to the desired position using the and arrows.
3. Click **OK**.

Operations

About Operations

An operation is a series of NC sequences that use a particular machine tool (workcell). It is also a workpiece feature that contains the following information:

- Name
- Workcell to be used
- Coordinate system for CL output
- Comments (optional)
- A set of manufacturing parameters that are used by NC sequences created within this operation (optional)

When you create the NC sequences and material removal features, they contain a reference to the current operation name. Operation setting is modal, i.e., once an operation is created, it stays current until another operation is created or activated.

Workcells

A workcell is a workpiece feature that specifies a machine tool by:

- Name
- Type
- A set of parameters
- Turret set-up
- Machine zones
- Coordinate system

The workcell type you use determines the types of NC sequences that you can create (e.g., Punch Press enables you to perform Nibbling, Punch, and Form NC sequences).

To Activate an Operation

1. Select the **Oper** tabbed page from the **SMT MFG MACHINING** dialog box.

2. Choose the desired operation from the list. The operation is activated in the Pro/ENGINEER graphics window.

Note: Once you activate an operation, it stays current until you create or activate another operation. All newly created NC sequences are included in this operation.

NC Post-Processing

About NC Post Processing

Pro/NCPOST is a post-processor generation system designed for end users having no software programming expertise. It can accommodate various flame contouring, laser contouring, wire edm, punch press, lathe and mill machine types that have up to five axes of continuous path control.

Use the QUEST facility to create a full representation of a numerically controlled (NC) machine and its controller. QUEST stores this information in a database that can be used by GENER.

GENER uses the information about the NC machine to convert the points and vectors of a cutter location (CL) file into the joint locations and machine codes of an NC control tape file.

The principal input for GENER is the CL file generated by Sheet Metal Manufacturing. GENER can post-process a CL file for any NC machine in the your database. The MACHIN can be used to select the post-processor, or a post-processor explicitly requested. In either case, GENER gets all necessary information about the NC machine from the database.

GENER outputs an NC control tape file that is ready for either punching to paper tape, or transmission to the NC machine via DNC network. Other software controls the actual transmission process. GENER also outputs a verification list that the NC programmer can use to check for problems, and the NC machine operator can use to follow the progress of the program.

Execution of the Post-Processor

From within Sheet Metal Manufacturing, you can:

- Select which post-processor to use.
- Execute post-processors with option to run CL file.
- Execute post-processors with options directly upon output of tool path.

Post Processor Verbose Window Display

The **Verbose** option activates a windows display. You have the ability to see CL input, tape output, macro trace messages, console I/O, and general statistics during GENER processing. A designated window contains each of the above items. The ncpost.pro file contains definitions that control which windows are displayed.

Interaction with the verbose windows is available either through use of the mouse or the keyboard. The following operations are supported:

- **d**
  Toggle tape detail—Press the d key to toggle how often stepping occurs in the tape output window. The two modes are every register or every block. The default at start-up is every block. Stepping at every register tells you which order the software outputs registers (i.e. linears before feed rate) so that macros can be written (for example with PPFUN/11 to force block purging) with an understanding of the register output order.
- <**right mouse button**>
  Step mode toggle—Press the right mouse button, or the escape key, to toggle GENER between single step mode and normal execution mode. When step mode is on, post-processing can be single stepped to provide a better understanding of the processing flow. When step mode is off, processing continues uninterrupted.
- <**left mouse button**>
  Single step—Press the left mouse button in the desired window when step mode is on to cause GENER to stop after the next line of output to the selected window.
- <**middle mouse button**>
  Window toggle—Press the middle mouse button in a selected window to turn output to the selected window on and off.

NC Post-Processors of Sheet Metal Manufacturing

In order to create and run post-processors for Sheet Metal manufacturing, you have to have a licence for Pro/NCPOST-SHEETMETAL or Pro/NCPOST-ADVANCED. Each of these includes a standard set of NC post-processors, and also enables you to create your own posts for punch press and 2-axis contouring.

Post-Processor Error Messages

GENER detects errors caused by invalid post-processor commands, inappropriate control of the NC machine and incompatibilities due to invalid QUEST responses. In all cases, processing of the CL file continues until the FINI record is read.

Error messages are textual. They identify a problem (or event), the severity of the problem, and any corrective action taken. Also listed are statistical and positional information for the error. For example:

Warning: CUTCOM Option LENGTH is not supported by this machine and cannot be simulated. Option ignored
SEVERITY(04) ISN(0035) CLREC(0041) ERRNUM(01524001)

Error messages contain three items: the type, text and status line. If the error occurs from within a macro, there may also be a message that indicates the position of the error within the macro. An additional line may also be output that indicates that the macro was terminated due to a serious error.

There are four types of error messages: Message, Warning, Error and Fatal. The message and warning type errors are generated when an unexpected event occurs for which the post-processor can correct. Error and fatal type messages are

generated when the post-processor detects an error that it cannot correct or make an assumption about.

The text portion of the error message begins with the post-processor word that generated the error message. The text body contains the error description, that includes any corrective action taken. For example, suppose the user coded the command LOADTL/1,OSETNO,3. During post-processing, the following message was output:

Warning: CUTCOM [Called by LOADTL] Option LENGTH is not supported by this machine and cannot be simulated. Option ignored.
SEVERITY(04) ISN(0035) CLREC(0041) ERRNUM(01524001)

The OSETNO option on the LOADTL command invoked a call to the CUTCOM command that handles cutter length compensation. However, the CUTCOM logic detected that cutter length compensation was not supported and it output the appropriate error message. This particular example is one of an error message generated on an indirect call. GENER tracks all indirect calls and places them at the beginning of the error message text.

The error status line is output after the text portion of the error message. Four items display on this line. They are the error severity (SEVERITY), internal sequence number (ISN), the CL record number (CLREC) and the GENER error number (ERRNUM). The error severity indicates the severity level of the message and may have any value between 0 and 99. A value between 0 and 3 signifies a "message." A value between 4 and 7 signifies a warning message. A value between 8 and 15 signifies an error message. Finally, a value between 16 and 99 signifies fatal message. The ISN points to the original APT source statement that caused the error. The CLREC is the Section III record number generated by the APT CL-Editor on which the error was found. The error number refers to the GENER system error number.

If the error occurs while processing a macro line, the macro is identified and the macro line causing the error is output. If an error occurs while evaluating a macro expression or executing a special macro command and the severity is greater or equal to 8, a TERMAC statement is generated to exit the macro and the following message is output:

*** TERMAC generated due to error(s).

Error File Utility

The error file utility permits you to customize error messages output by GENER. The file ncp160.err contains a list of all error messages used by GENER. Each error message consists of three fields; number, severity and text.

Do not modify the error number field because this is the number that GENER uses to find a particular error. You may modify all other information. The severity field specifies the severity of the error message. A severity from 0 through 3 signifies a message; warnings are from severity 4 through 7; errors from 8 through 15; and fatal errors from 16 and up. A negative severity disables the output of an error message. The message field specifies the text associated with an error number. The message text does not have a length limitation. A typical message appears as follows:

```
\ 01003004 4 MULTAX/ON with no Y-axis on this lathe.
Tool orientation may not be satisfied.
```

Any line that starts with an exclamation mark (!) is a comment. Lines that start with a back slash (\) define an error message. Error message definitions can continue from one line to the next and ignore any extra spacing characters.

To Generate an MCD File

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select the desired NC sequence.
3. Click **CL Output**. The **SMT MFG NCL PLAYER** appears.
4. Click **FILE > Post Process**. The **PP OPTIONS** menu appears. Select one of the following options:
   - **Verbose**—Invoke the verbose display of post-processing.
   - **Trace**—Trace all macros and CL records in the listing file.
   - **MACHIN**—Use the post-processor file for the machine specified in the MACHIN statement of the CL file. If this option is not checked off, the system prompts you to select a post-processor from the namelist menu of all available post-processors.
5. Click **Done**. The post-processor is invoked with the specified options. The post-processed file is named `<fname>.tap`, where `<fname>` is the name of the CL file being processed.

Post-Processing Deliverables

Output NC Program Data

The output NC program data is in 8-bit noparity ASCII format. You control the character count by the block_size definition file symbol. The block size is written as a formatted number in the range `00` through `99`, followed by a space, followed by the contents of the actual NC tape block. The three byte size header is not included in the block size count.

Records may or may not contain end-of-block characters, rewind-stop characters, leader, trailer and man-readable data. You control these, along with many other format details, from within the Output Format section of QUEST. Other format options configurable from the definitions file include the `tape_format` (variable, fixed or packed), `tape_recl` (tape record length), `tape_seq` (tape sequence numbering), `tape_eor` (tape end-of-record character), `dnc_format` (variable, fixed or packed), `dnc_recl` (dnc record length), `dnc_seq` (dnc sequence numbering), and `dnc_eor` (dnc end-of-record character).

By default, the system creates the tape file in the current directory with the same name as the CL file, and an extension of tap. The system may create two tape files

for 4-axis lathes, in which case the second file name is the same as the first, with a file extension of tp2. There is no mechanism to override the second tape file name.

If you define the configuration option `pro_mf_cl_dir`, the system creates the deliverable in the directory specified by this option.

Output NC Program Listing

The post-processor listing file contains the complete program listing, including timing data, diagnostics, tape listing, operator messages, etc. You control these, along with many other format details, from within the Display Format section of QUEST.

Two output formats are supported: FCC and list. You can control the format by the output_format definition file symbol. The FCC format, which stands for Fortran Carriage Control, reserves the first character of each line for page formatting purposes. GENER outputs a 1 in this position to advance to a new page, a 0 to double space an output line, and a blank for regular single spacing. You can print this file on any printer or driver recognizing FCC output. The list format does not reserve a character for carriage control. Instead, a form-feed character is output at the start of a record to advance to a new page. All other records are output one per line.

The page length is configurable from the definitions file with the `lines_per_page` variable.

By default, the system creates the listing file in the current directory with the same name as the CL file, and an extension of lst.

Identification Page

When GENER is run, an unnumbered identification page is output for control purposes. This page identifies the GENER software version and modification level, the name of the input CL file, the creation date of the QUEST generated post-processor, the database from which the post-processor was read, the software version of the database, the names of macro libraries being used if applicable and, optionally, the NC control tape program identification string (commonly called a PID).

For example:

```
ICAM Technologies Corporation
(c) Copyright 1995
Pro/NCPOST GENER Release 18.0-9550
Source: 3axtest.ncl.1
Quest: MILL501.104;4
Created 01-Jul-1996 12:00:30.00
Using /usr/francois/work/ncpost.dbf
Release 18.0
PID: 9001
```

This header page shows that Release 18.0 modification level of GENER is being run. You must reference this number (whatever appears in the listing file) when you communicate problems with the product to Parametric Technology Corporation support personnel.

The input CL file is named TEST.NCL.

The MILL501 post-processor was either explicitly requested by the user, or it was determined by scanning through the 3axtest.ncl.1 file for a MACHIN CL record. This post-processor was created just after noon on July 1, 1996. The post-processor database is located in the directory `/usr/francois/work/ncpost.dbf` and was created using Release 18.0 of Pro/NCPOST.

A PID string was specified when GENER was executed, or a default PID string was defined by the post-processor creator (using QUEST). In either case, the value written to the NC control file is 9001.

GENER cannot use a database that is created using a different version of Pro/NCPOST. Attempting to do so results in an error message and processing aborts. However, no harm is done to the old database. The QUEST utility can be used to update the database to the new version level if necessary.

From time to time, new GENER and QUEST software may be released at the same version level but at a higher modification level. When a post-processor created at a lower level of QUEST is run on a higher level of GENER, warning messages may be displayed indicating that problems exist. Specific instructions to alleviate this problem are always provided with the new software release. One sure way of eliminating this problem is to run the new software release of QUEST and regenerate the post-processor.

The NC Control Tape Listing

The GENER verification file always contains an NC control tape listing. This is provided as a reference for the NC machine operator. The tape listing is an exact copy of what appears on the tape file except that man readable are omitted.

The verification file may also contain status information listed in column format beside the NC control tape.

You can control the width and length of pages. Use Pro/NCPOST to control the width of a page. Use the `lines_per_page` symbol in the definition file to set the length of the page.

Title Block

The title block at the top of each page displays the page number, run date, post-processor name, and PARTNO string. The post-processor title appears on the first page only.

Occasionally instead of a PARTNO string, the text Undefined PARTNO appears. This happens when a PARTNO command is not found within the first 20 records of the CL file.

Display Data

The NC control tape data (as it would appear in the tape file) is located at the left hand side of each page. A single block is output at a time. The dollar sign ($) character replaces non-printing characters so as to not interfere with the printer, however the tape file contains the true characters.

The right hand side of each page may contain status information in column format. Control the appearance of this information by responses in the **General Description—Display Format** section of Pro/NCPOST QUEST. A header appears above each status column. Descriptions of the various status types follow:

nnAXIS:

Displays the position of the named axis, in normalized machine coordinates, at the end of the NC command block.

Linear axes are output in machine units using the right hand rule (i.e. reversed axes are output with a normal sign convention). Primary linear axes are identified in the header by X, Y and Z; secondary linears by U, V and W; and the extending axis by E.

Rotary axes values are output in degrees regardless of the actual units used by the machine. Sign conventions follow the right hand rule. Rotary head axes are identified by A, B and C; rotary tables by A', B' and C'; and the nutating axes by N and N2. Rotary axis positions shown in the display are actual positions and may not match the tape output where values of 360 degrees or greater occur.

If there is not enough space to display all axes in unique status columns, GENER uses the same column for similar axes (i.e., Z, W and E) outputting 2 or 3 lines of status information for each NC command block. The first line of linears lists primary axes; the second line lists secondary axes, the third lists the extending axis under the Z-axis column. The first line of rotaries lists the head axes; the second line lists the table axes; the nutating axes always appears in their own column.

TIP VEL:

Displays the calculated velocity of the tip of the cutting tool with respect to the part coordinate system. Normally this value matches the requested feedrate in the CL file. A value of zero is output on blocks that do not contain motion instructions. This column is not available for punch presses.

RPM:

Displays the calculated revolutions per minute of the spindle at the end of the NC command. A value of zero is output when the spindle is stopped. Negative values can indicate CCLW rotation if the creator of the post-processor specifically requests this feature. This column is not available for punch presses.

TIME:

Displays the elapsed machining time in hh:mm:ss (absolute) format or the delta block time in mmm:ss.ss (incremental) format.

nnnn ACT:

Displays the actual machining time in absolute or incremental format for the MAIN and SIDE heads of a 4-axis merging lathe. Actual machining time does not include the time spent waiting for a synchronization event.

CL:

Displays the number of the CL record that result in the NC instruction block being output. This provides a cross reference between CL records and NC commands. A value of zero is used during start-up processing.

ISN:

Displays the Internal Sequence Number of the APT statement (or CAD/CAM instruction) that create the CL records that result in the NC instruction block being output. This provides a cross reference between the original APT program statements and NC commands. A value of zero is used during start-up processing and also when information concerning ISN numbers is not provided in the CL file (which is common with some CAD/CAM systems).

HITS:

Displays the number of hits per tape block. This column is available for punch press machines only.

Summary Data

The last few lines on each page optionally contain an accumulated tape length and machining time summary. Tape length is output in feet, meters or kilobytes, depending on the post-processor units and output format. The reel number is included where applicable. Machining time is in hh:mm:ss format.

You can request a second type of summary line in QUEST to report the total machining time for the tool when a tool change occurs. The time is broken down to show times for positioning, cutting, miscellaneous and, for merging lathes, idle time. The summary line precedes the tape blocks that perform the tool change. Note that this type of summary is not available for punch press machines.

Program Summary

An NC program summary can be found at the end of the verification listing. An interim summary is also provided for each reel in a multi-reel program. These include tooling, machine travel, machining time and diagnostic summaries.

Tooling Summary

The tooling summary is optional. It supports up to five independent automatic tool changers (ATC) on a single milling or turning machine. For each ATC the summary lists all tools that are loaded. Information for each tool includes identity number, pocket number, diameter, length, gripper size, diameter and length compensation switch numbers, minimum and maximum feeds, and feed, rapid, miscellaneous and total times.

Punch presses have a nibble and punching summary for each tool along with the machining times.

The less than 132 column format of the tool summary displays one half of the information for all tools, followed by the second half of the information for the same set of tools. The 132 (or greater) column format lists all the information for each tool in one line. The tooling summary appears only if the post-processor creator specifically requests it.

Tool Travel Summary

The tool travel summary always appears. It lists the minimum, maximum and total travel for all axes. The tool travel is with respect to the machine coordinate system in effect at the end of the program (or reel). Use the ORIGIN, TRANS and POSTN post-processor commands to control the machine coordinate system.

For example, the travel summary for a mill might be:

TOOL TRAVEL SUMMARY:

AXIS	MINIMUM	MAXIMUM	TOTAL
X	-2.50000	2.50000	5.00000
Y	-2.50000	2.50000	5.00000
Z	-1.50000	2.00000	3.50000
A’	0.00000	0.00000	0.00000
B’	0.00000	0.00000	0.00000

Note: Linear axis travels are in machine units. Rotary axis values are in degrees.

Machining Time

The machining time summary is optional. It lists, in hh:mm:ss format, the time spent positioning, cutting, on miscellaneous operations, and for merging lathes the time spent at idle time. Also for merging lathes, the times for both heads are listed independently.

In the following example, the total time spent at feed was 42 seconds. This accounted for over 97% of the total machining time.

MACHINING TIME SUMMARY:

Type	HH:MM:SS	Percent
Rapid	00:00:01	2.135
Feed	00:00:42	97.865
Misc	00:00:00	0.000
Total	00:00:43	Note that for punch presses, the RAPID and FEED lines are replaced by POSITIONING and PUNCHING.

Diagnostic Summary

The diagnostic summary is optional. There are two types of diagnostic summary, either or both of which can appear, controlled by answers in QUEST.

The basic summary lists the number of diagnostics that occurred in each message class. If no diagnostics occurred, the summary states that fact. There are four classes of diagnostics:

- Messages are informational only.
- Warnings indicate problems which have been corrected automatically.
- Errors are uncorrectable problems that probably result in an unusable NC control tape.
- Fatal errors indicate severe problems.

In the following example, 3 warnings and 2 errors occurred. There were no informational messages nor were there fatal errors.

DIAGNOSTIC SUMMARY:

Type	Total
Message	0
Warning	3
Error	2
Fatal	0
Total	5

The detailed summary lists by message category, a compilation of all diagnostics that have occurred in the program. For each diagnostic type, a count of the number of occurrences and a page reference of the first occurrence is given.

External Post Processors

About External Post-Processors

The following post-processors are certified for use with Sheet Metal Manufacturing CL Data files:

- Other post-processor reading APT should also, with little additional code, be able to process Sheet Metal Manufacturing CL files.
- If you have CAM-POST installed, you can directly set-up the post-processors and generate MCD files from within Sheet Metal Manufacturing.
- If you have IntelliPost installed, you can configure post-processors, generate MCD files, and look at the output files from within Sheet Metal Manufacturing.

IntelliPost

IntelliPost Post-Processor Set Up

After you install IntelliPost, the following options appear in the **MFG SETUP** menu:

- **Configure**—Enable you to configure the post. When finished configuring, if you wish to test the post, select the exit command that writes the file but does not actually exit (see the IntelliPost manual). You can then click **Post Process** (described in the next bullet) and a new window appears to run the post. When you wish to exit the **Configure** step, reactivate the **CONFIG** window and do the full exit.
- **Post Process**—Enable you to post-process a CL file using a given data file. A pop-up window appears requesting a selection of **MILL** or **LATHE**. After this selection, the post-processor requests a CL file name and a post data (.dat) file. You can leave the current **Post Process** window and select the **List Files** (described in the next bullet) to get a list of CL files, and then return to this window to enter the name. After processing is complete, the window disappears.
- **List Files**—Enable you to view the MCD output files (*.PPR and *.PCH) and to list the CL files. A sub-menu appears from which you can select the file type to view. If you select the **CL file** type, a list of CL files appears. If you select one of the MCD file types, you can select a specific file to view.

Generating IntelliPost MCD Files

There are three places in the system where MCD files can be generated:

- When setting-up the post-processor the generation function is provided to give you a place to configure the post, process it, and fine tune the post-processor data file.
- When outputting the tool path, use **CL Data**, **Output**, **File**, by then select the MCD File option from the **Output Type** menu. The generation function is provided there to enable you to create and store an MCD file at the same time you store the CL file. You do not exit the menu or enter other menus.
- From inside the CL PLAYER, it is possible to perform a FILE / OPEN and then select a previously saved NCL File with the browser. Then you can launch the Post Processing:
    - **Post Process**—Enables you to process an existing CL file. A window requests the CL file name and the post data (.dat) file.

Direct External Post-Processing with IntelliPost

IntelliPost is integrated with Sheet Metal Manufacturing to enable you to configure and run the post-processor without leaving Sheet Metal Manufacturing. This is accomplished by means of the Pro/DEVELOP package that enables access to functions and data within the Sheet Metal Manufacturing system. The integration to IntelliPost is supported by Software Magic.

Install IntelliPost separately from Pro/ENGINEER and connect by means of written instructions and script files that are specific to the platform. you must obtain the licensing and authorizations from Software Magic prior to using the interface.

The integrated IntelliPost enables you to:

- Create customized post-processors.
- Select which custom post-processor to run.
- List the resultant MCD and PPR files.
- Test post-processors by configuring and executing the post-processor interactively.
- Execute the post-processor when outputting the tool path.

CAM-POST

CAM-POST Post-Processor Set Up

When you click **SETUP > PProcessor**, the system invokes **CAM-POST QUEST** to let you set up post-processors.

Generating CAM-POST MCD files

To generate an MCD file from a CL file:

1. From the **SMT MFG NCL PLAYER**, open the desired CL file.
2. Click **FILE > Post Process**. The **PP OPTIONS** menu appears. The following options are available:

- **Verbose**—Invoke the verbose display of post-processing. If you check off this option, the post-processor is invoked with the `-v` option.
- **Trace**—Trace all macros. If you check off this option, the post-processor is invoked with the **-tra=cm** option.
- **MACHIN**—Use the post-processor file for the machine specified in the MACHIN statement of the CL file. If you check off this option, the post-processor is invoked with the `-po=1` option. If you do not check off this option, the system prompts you to select a post-processor from the namelist menu of all available post-processors. The post-processor is then invoked with the `po=<ppname>` option, where `<ppname>` is the name of the post-processor selected.

- **PID**—Use the process identification. If you check off this option, the system prompts you for the process identification string. The post-processor is then invoked with the `pid=<pidstring>` option, where `<pidstring>` is the string entered at the prompt. The string should not contain any spaces.

3. Click **Done**. The post-processor is invoked with the specified options. The post-processed file is named `<fname>.tap`, where`< fname>` is the name of the CL file being processed.

Direct External Post-Processing with CAM-POST

CAM-POST is integrated in Sheet Metal Manufacturing to enable you to directly set-up the post-processors and generate MCD files.

In general, Pro/ENGINEER and CAM-POST are installed separately. CAM-POST must be installed and authorized prior to using the Sheet Metal Manufacturing interface.

To specify that the post-processor is present, set the following configuration option:

```
campost_dir <postpathname>
```

where `<postpathname>` is the CAM-POST load point directory.

From within Sheet Metal Manufacturing, you can:

- Create post-processors.
- Select which post-processor to use.
- Execute post-processors with option to run CL file.
- Execute post-processors with options directly upon output of tool path.

Pro/NC-Check

About the NC Check Process

You can perform NC Check:

- At the time you create an NC sequence, to check the current tool path. This option is only available for Laser and Flame NC sequences.
- After you create the NC sequence (any type) or operation, by selecting NC Check from the CL DATA menu. The system prompts you for a CL file name. At this point, you can either select an existing file, or create a new one.
- When you edit the CL data.

NC Check is implemented for all Punch Press NC sequences except Form, and for Contouring NC sequences. If the type of tool you use in an NC sequence is STANDARD PUNCH, each tool position you specify with GOTO creates a punch in the workpiece.

If you use a tool model, it is replaced by the default tool of appropriate type, as defined by the tool parameter values. The material is removed according to the actual tool shape and motion, i.e., all scallops left by the tool passes are displayed.

Do not confuse the automatic material removal simulation with the NC Check process:

Material Removal	**NC Check**
Permanently removes material from the workpiece and creates a new feature.	Simulates material removal for display purposes only; does not create geometry
Is based on a set of assumptions and may produce simplified representation of material removal, depending on the NC sequence type	Always takes into account the actual tool path and shape

To Run NC Check When You Create an NC Sequence

When you create a Laser or Flame NC sequence, you can run it through the NC Check process to check your current tool path.

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select the desired NC sequence to check.
3. From the **SMT NC SEQUENCE** dialog box, click **Preview**. The **SMT MFG NCL PLAYER** appears.
4. Click **OPTIONS > NC Check**.
5. Click **Play**. The system starts running the current CL data through NC Check.

To Run NC Check After You Create an NC Sequence

At any time in your manufacturing process, you can run the CL data of an existing NC sequence through the NC Check process to check the tool path and resulting geometry. Running several NC sequences on top of each other without Refresh gives you a comprehensive picture of workpiece geometry after machining. CL data for all the NC sequences you are going to run must be output to CL files prior to starting the NC Check process.

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select the desired NC sequence to check.
3. Click **CL Output**. The **SMT MFG NCL PLAYER** dialog box opens.
4. Click **OPTIONS > NC Check**.
5. Click **Play**. The system starts running the specified CL file. Once the NC Check process is completed (or aborted), you can run another NC sequence file.

Press Punch NC Sequences

To Create a Form NC Sequence

1. From the **SMT MFG MACHINING** dialog box, click **NC Seq**.
2. Click **New > Forming**. The **SMT NC SEQUENCE** dialog box opens. Select one or more of the following options:
    - **Name**—Specify a name for the NC sequence. This name replaces the default NC sequence name.
    - **Parameters**—Specify manufacturing parameters.
    - **Comment**—Specify comments about the NC sequence.
    - **Info**—Displays a summary of the NC sequence and parameter information.
    - **Next**—Go to the next NC sequence.
    - **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.
3. Define form element.
4. Select a reference part to create a Form NC Sequence.
5. Select forms to create a From NC Sequence.
6. Click **Done Sel**.
7. If required, click **CL Command** and click **Define**. This inserts new CL commands at selected hits.
8. Click **Done**.

Note: All Form features included in an NC sequence must use the same tool.

To Specify Overhang Points

1. In the **SMT NC SEQUENCE** dialog box, click the **Overhang** element and click **Define**. The **Overhang Definition** dialog box opens.
2. Click **Add**.
3. Select a vertex within the selected chain of edges. You cannot select chain endpoints.
4. If you want the overhang to be the same for both directions, select the **Same for both directions** check-box.
5. Specify the **Overhang Distance**.
6. Click **OK**.

To Select Edges and Specify Check Edges

Selecting Edges

You can select noncontinuous edges, chains, and loops using the **SELECT EDGES** dialog box. Select the **NC Seq** tabbed page from the **SMT MFG MACHINING** dialog box. Click **New** > **Nibbling**. Select **Check Edges** and click **Define**. The **SELECT EDGES** dialog box appears.

The **SELECT EDGES** dialog box contains the following options:

- **Include**—Include edges.
- **Exclude**—Exclude previously selected edges, chains, or loops.
- **Edge**—Select individual edges of reference parts.
- **Chain**—Select a chain of edges. You are prompted to select the first edge. Then the whole loop is highlighted and you are prompted to select the second edge. One of the chains between the two edges is highlighted in red.
- **Loop**—Select a closed loop of edges (e.g., the whole perimeter of the reference part) by selecting an edge that belongs to the loop. The system finds the whole loop based on the selected edge and highlights it. With this selection method, if you later modify geometry of the reference part but leave the original edge intact the system regenerates the nibbling NC sequence around the new contour.
- **Show Edges**—Display edges selected for the NC sequence (e.g., after Repaint).

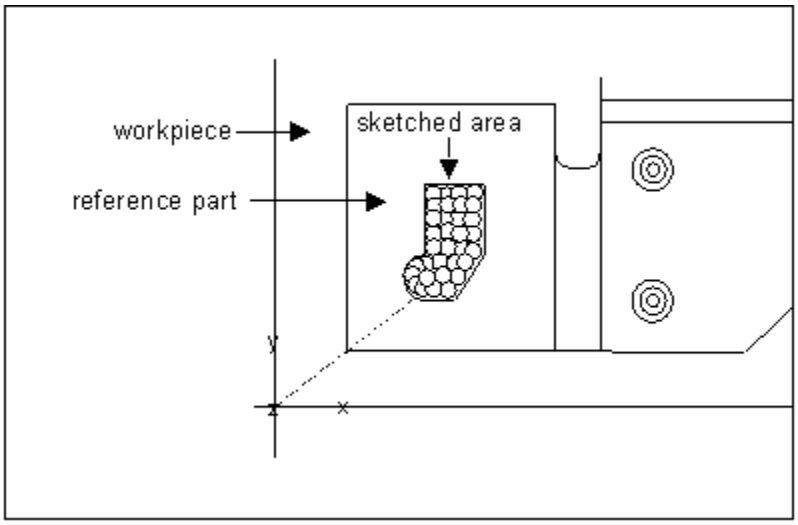
Nibbling NC Sequences

About Nibble Area NC Sequences

The Nibble Area NC sequence creates a point-to-point tool path across a sketched area. The tool path is broken down into smaller elements called strokes that are in turn broken down into hits, the smallest elements of the tool path. Hits indicate each location where the tool makes contact with the workpiece and removes material. A succession of hits in a line across an area composes a stroke. Strokes are displayed as a series of hits connected by a line.

Use the `OVERLAP_DIST` parameter to specify the overlap distance between the next and the previous hit. The `STEP_OVER` parameter defines the distance between the strokes. If `STEPOVER_ADJUST` is set to `YES`, by default, the step-over distance is adjusted to make both the beginning and the end of the cutter path close to the sketched boundaries.

Nibble Area NC Sequence



If you select a tool for the NC sequence that is too big to fit into a portion of the sketched area without gouging, this portion is not nibbled. This is reflected in CL data output. However, the material is shown as removed from the whole sketched area. Use CL data display or Pro/NC-CHECK to verify the NC sequence. You can either modify the original NC sequence to use a smaller tool, or split the area and create two separate nibbling NC sequences.

To Create a Nibble Area NC Sequence

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Click **New > Nibble Area**. The **SMT NC SEQUENCE** dialog box opens. The following options are available:
   - **Name**— Enter a name for the NC sequence. This name replaces the default NC sequence name.
   - **Parameters**—Specify manufacturing parameters.
   - **Comment**—Use the system editor to enter comments about the NC sequence.
   - **Info**—Displays a summary of NC sequence and parameter information.
   - **Next**—Go to the next NC sequence.
   - **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.
3. From the **TURRET MANAGER** dialog box, define the tool and click **Done**. The sketcher window appears.
4. Define the nibble area in the sketcher window and the reference in the **References** dialog box.
5. To use any of the optional elements, choose the one you want, then click **Define**. The following options are available:

- **Profile Tool**—Specify a second tool for profiling operations.
- **Scan Type**—Specify the nibble area tool path.
- **Remove Hits**—Select hits to be deleted from the tool path.
- **Change Order**—Change the order of the strokes and hits within the stroke.
- **CL Command**—Insert CL commands at selected hits.

6. Click **Done**.

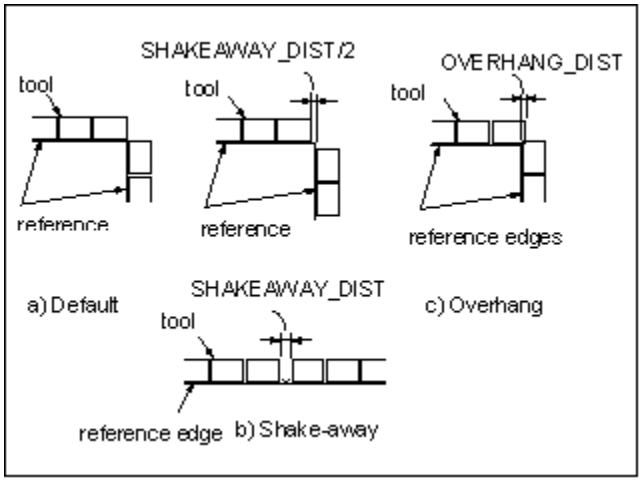
About Nibble Edge NC Sequences

The Nibble Edge NC sequence creates a point-to-point tool path along selected edges. The tool path is broken down into smaller elements, called strokes, that in turn are broken down into hits, the smallest elements of the tool path. Hits indicate each location where the tool makes contact with the workpiece and removes material; they display as small red shapes. A succession of hits comprises a stroke, that is the other element of the tool path. A stroke is a succession of hits along an edge of a reference model. When the tool path is generated, strokes display as a series of hits connected by a line; the first hit in the stroke displays as a small, filled red shape. These shapes indicate the position of the center of the tool; the distance between the hits is calculated based on the actual tool size. You can specify the distance by which each following hit overlaps the previous one using the `OVERLAP_DIST` parameter.

You can choose form a selection of shapes to perform the nibble edge NC sequence. A single sequence can contain multiple tools. Tools can be automatically or manually assigned to an edge, loop or chain as needed. Special tools can be used to punch corners.

By default, the tool path along each edge is equal to the edge length. You can specify shake-away points or vertices (Nibble Edge Tool Path:b) and overhang vertices (Nibble Edge Tool Path:c), where the tool path is trimmed or extended by a specified distance, respectively. You can specify different shakeaway and overhang distances for different points within the same NC sequence. For overhang vertices, you can also specify different overhang distances along the two edges of a vertex.

Nibble Edge Tool Path



If you select a tool for the NC sequence that is too big to nibble some of the edges without gouging, the tool path along these edges is not created. This is reflected in CL data output and in the material removal simulation. You can then either modify the original NC sequence to use a smaller tool, or create a separate nibbling NC sequence for these edges.

To Create a Nibble Edge NC Sequence

1. From the **SMT MFG MACHINING** dialog box, click **NC Seq**.
2. Click **New > Nibbling**. The **SMT NC SEQUENCE** dialog box and the **TURRET SETUP** dialog box open.
3. Select one or more of the following options from the **SMT NC SEQUENCE** dialog box:
   - **Name**— Specify a name for the NC sequence. This name replaces the default NC sequence name.
   - **Parameters**—Specify manufacturing parameters.
   - **Comment**—Specify comments about the NC sequence.
   - **Info**—Displays a summary of the NC sequence and parameter information.
   - **Next**—Go to the next NC sequence.
   - **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.
4. From the **TURRET SETUP** dialog box, define the tool, and click **Done**. The **SELECT EDGES** dialog box opens.
5. Select the required edge, chain, or loop and click **Done**.
6. To use any of the optional elements from the **SMT NC SEQUENCE** dialog box, select the one you want, and click **Define**. Select one or more of the following options:

- **Check Edges**—Excludes some of the edges from gouge checking.
- **Shake Away**—Specify the shake-away placements and sizes.
- **Overhang**—Specify the overhang placements and sizes.
- **Multiple Tool**—Specify the tools from the **MULTIPLE TOOLS** dialog box.
- **Remove Hits**—Select hits to be deleted from the tool path.
- **Change Order**—Change the order of the strokes and hits within the stroke.
- **CL Command**—Inserts new CL commands at selected hits.

7. Click **Done**.

About Multiple Tool Nibbling

You can simplify the creation of nibbling NC sequences necessary to punch out part profiles by using multiple tools. To access Multiple Tool, select **Multiple Tool** from the options in the **SMT NC SEQUENCE** dialog box, and click **Define**. The **MULTIPLE TOOLS** dialog box opens with the following options:

TOOLS

- **Nibbling**—Manually assigns tools to edges.

  GEOMETRY

  - **Change Tool for Edge**—Assigns a new tool for a single edge.
  - **Specify Edges for Tool**—Assigns edges for the selected tool.
- **Auto Select**—Automatically assigns tools to edges.
- **Recognize**—Manually assigns edges to tools.

  GEOMETRY

  - **Exclude Edge/Vertex**—Excludes an edge or vertex for a specified tool.
  - **Include Edge/Vertex**—Includes an edge or vertex for a specified tool.
- **Show Section**—Displays the tool section.
- **Add New**—Select a new tool from the **TOOL LIST** dialog box.
- **Remove**—Select a tool to remove it from the **TOOL LIST** dialog box.

PARAMETERS

Assign manufacturing parameters to individual tools.

Autonibbling

About Autonibbling

Autonibbling is an advanced system of sheet metal nibbling. Autonibbling allows you to automatically select all edges for nibbling sequencing. You can setup multi-tool nibbling and modify individual nibbling sequences created from the sequence group.

The **CONTOUR SELECTION** dialog box contains controls for the autonibbling functions. The following options are available:

SELECTION METHOD

- **Automatic**—Automatically contour your selection.
- **Manual**—Manually exclude contour selections.
- **Show**—Show selected contours.

USE CONTOURS

- **Outer**—Select to process outer contours.
- **Inner**—Select to process inner contours.

TOTAL and SELECTED

Displays total number of edges selected.

To Create an Autonibbling Sequence

1. Click the **NC Seq** tab from the **SMT MFG MACHINING** dialog box.
2. Click **Auto > Nibbling**. The **TURRET MANAGER** dialog box opens.
3. Load desired tools then click **Done**. The **CONTOUR SELECTION** dialog box opens with the following options:
    - **Selection Method**—Enter automatic or manual contouring method.
    - **Show**—Show selected contours.
    - **Use Contours**—Select to process inner or outer contours.
    - **Total**—Number of contours found.
    - **Selected**—Number of contours selected.
4. Click **Done**. The **SMT NC SEQUENCE** dialog box opens. Click **Preview** to preview the nibbling sequence using the **SMT MFG NCF PLAYER** or **Done** to complete.

Punch NC Sequences

About Punch NC Sequences

Tool path for Punch NC sequences consists of separate hits at specified locations.

There are three types of Punch NC sequences:

- **Punch Point**—Create punches by referencing datum points as the hit locations. This NC sequence can use either a Standard Punch, defined by a .tpm file, or a library part.
- **Tool Shape**—Automatically creates punches on holes or slots that match the shapes of the tools in the turret. This NC sequence can also use either a Standard Punch or a library part.
- **Punch UDF**—Create punches by referencing Punch or Notch features in the reference model. This NC sequence can only use the tool of type Punch defined by the Punch or Notch feature section.

To Create a Punch NC Sequence by Referencing Datum Points

1. In the **SMT MFG MACHINING** dialog box, click **NC Seq**.
2. Click **New > Punch Point**. The **SMT NC SEQUENCE** dialog box and the **TURRET SETUP** dialog box open.
3. Select one or more of the following options:
    - **Name**—Specify a name for the NC sequence. This name replaces the default NC sequence name.
    - **Parameters**—Specify manufacturing parameters.
    - **Comment**—Specify comments about the NC sequence.
    - **Info**—Displays a summary of the NC sequence and parameter information.
    - **Next**—Go to the next NC sequence.
    - **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.
4. In the **TURRET SETUP** dialog box, define the tool and click **Done**.
5. To specify datum points, click **PUNCH PNTS** > **Add**. The **SEL PNCH PNT** menu appears. Select one of the following options:
    - **Select**—Selects existing datum points.
    - **Create**—Creates datum points on the fly. Datum points created using this option belong to the workpiece.
6. Click **Remove** in the **PUNCH PNTS** menu to clear selected datum points.
7. Click **Show** in the **PUNCH PNTS** menu to highlight datum points that are currently included in the NC sequence.

8. If required, click **CL Command** and click **Define**. This inserts new CL commands at selected hits.
9. Click **Done**.

To Create a Punch NC Sequence by Referencing Tool Shape

For this type of NC sequence, the punch locations are determined by matching the reference part geometry (edge contours) with the shape of the tool. Define the tool shape either by the tool parameters or the tool part geometry. A good example of this NC sequence is using a round punch to punch all the existing holes of a certain diameter: once the tool shape is specified, all the holes that can be punched with it are selected automatically.

The system performs partial matching as well, i.e., matches a circular tool to all arcs with the same radius, including those on the outer contour of the reference part (i.e., Notches).

Note: This type of Punch NC sequence can also be created automatically using the Auto Tool option.

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Click **New** > **Tool shape**. The **SMT NC SEQUENCE** dialog box opens and the **SELECT PART** menu appears. The following options are available:
   - **Name**— Enter a name for the NC sequence. This name replaces the default NC sequence name.
   - **Parameters**—Specify manufacturing parameters.
   - **Comment**—Use the system editor to enter comments about the NC sequence.
   - **Info**—Displays a summary of NC sequence and parameter information.
   - **Next**—Go to the next NC sequence.
   - **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.
3. From the **TURRET SETUP** dialog box, define the tool and click **Done**.
4. Click **Select** from the **SELECT PART** menu. Select an appropriate reference part. The system finds all edge contours that match the tool shape and generates a default tool path by creating a hit at each matching location. If no matches are found an error message is displayed.
5. Click **Done**.
6. To use any of the optional elements, choose the one you want, then click **Define**. The following options are available:
   - **Remove Hits**—Select hits to be deleted from the tool path.
   - **Change Order**—Change the order of the strokes and hits within the stroke.
   - **CL Command**—Insert CL commands at selected hits.

7. Click **Done**.

To Create a Punch NC Sequence by Referencing Punch Features

Each Punch or Notch feature has a specific tool that defines its shape. The system prompts you to enter the tool name and symmetry flag when you define the group for a Punch feature creation. This tool information, as well as the coordinate system included in the section sketch, is used in creating a Punch NC sequence. All Punch or Notch features included in the NC sequence must have the same tool information.

1. From the **SMT MFG MACHINING** dialog box, click **NC Seq**.
2. Click **New** > **Punch UDF**. The **SMT NC SEQUENCE** dialog box opens and **SELECT PART** menu appears.
3. Select one or more of the following options from the **SMT NC SEQUENCE** dialog box:
    - **Name**—Specify a name for the NC sequence. This name replaces the default NC sequence name.
    - **Parameters**—Specify manufacturing parameters.
    - **Comment**—Specify comments about the NC sequence.
    - **Info**—Display a summary of the NC sequence and parameter information.
    - **Next**—Go to the next NC sequence.
    - **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.
4. Define punch element.
5. Select a reference part to create a Punch NC Sequence.
6. Select punches to create a Punch NC Sequence.
7. Click **Done Sel**.
8. If required, click **CL Command** and click **Define**. This inserts new CL commands at selected hits.
9. Click **Done**.

Contouring NC Sequences

About Contouring

Contouring is a manufacturing NC sequence that enables you to cut along a selected contour. Sheet Metal Manufacturing provides two types of Contouring NC sequences:

- **Laser**—Use laser/plasma cutting.
- **Flame**—Use flame cutting.

The user interface for these NC sequences is the same. The parameter files and tools for the two NC sequence types are interchangeable.

CL data output for the Contouring NC sequences differs in the following way:

For Laser contouring, `POWER / ON` is output at the beginning of the cut motion and `POWER / OFF` is output at the end.

For Flame contouring, `TORCH / ON` is output at the beginning of the cut motion and `TORCH / OFF` is output at the end.

To Create a Contouring NC Sequence

1. From the **SMT MFG MACHINING** dialog box, click **NC Seq**.
2. Click **NEW** > **Contouring** (you must be in a Contouring or Hybrid workcell). The **SMT NC SEQUENCE** dialog box and the **SMT CUT MOTION** dialog box open.
3. Select one or more of the following options from the **SMT NC SEQUENCE** dialog box:
   - **Name**—Specify a name for the NC sequence. This name replaces the default NC sequence name.
   - **Parameters**—Specify manufacturing parameters.
   - **Comment**—Specify comments about the NC sequence.
   - **Info**—Displays a summary of the NC sequence and parameter information.
   - **Next**—Go to the next NC sequence.
   - **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.
4. In the **SMT CUT MOTION** dialog box, select the desired edge, chain or loop and click **Done**.
5. Specify the **Contour Definition** as follows:
   - **Contour Type**—Select Outer or Inner contouring.
   - **Use Offset**—Select to use offset for contouring.
   - **Flip direction**—Click to flip the direction of contouring.
   - **Use CutCom**—Select to use cutter compensation.
   - **Start Edge**—Click **Select** and select the edge for starting the contouring.
   - Select a point on the X-Y axes to start contouring.
   - **Lead in**—Selects the point at which contouring starts.

     First Point—Contouring starts at the first point of the start edge.

     Middle Point—Contouring starts at the middle point of the first edge.

6. To use any of the optional elements from the **SMT NC SEQUENCE** dialog box, select the one you want, and click **Define**. You can select one or more of the following options:
   - **Approach/Exit**—Specify the approach and exit motions.
   - **Corner Conditions**—Add a corner condition for NC Sequence.
   - **Shakeaway Points**—Specify the shake-away points.
   - **CL Command**—Insert CL commands at selected hits.
7. Click **Done**.

To Add a Corner Condition for a Contouring NC Sequence

1. From the **NC Seq** tabbed page, select the desired NC sequence.
2. Click **OPERATE** > **Redefine**. The **SMT NC SEQUENCE** dialog box opens.
3. Select **Corner Conditions** and click **Define**. The **CORNER CONDITIONS** dialog box opens.
4. Click the type of **Concave Corners** and **Convex Corners** that you want to apply to the part and specify the required values for the corresponding corners.
5. If required, click and set the dwell time.
6. Click **Preview** to view the results.
7. Click **Corner Info**.
8. Use or to highlight a specific corner. Highlighted corners can be edited or deleted as required.
9. If required, highlight a specific corner and click **Delete** to delete the corner.
10. Click **Done**.

To Specify Shake-Away Points

1. From the **SMT NC SEQUENCE** dialog box, click **Shake Away** and click **Define**. The **SMT SHAKEAWAY** dialog box opens.
2. Specify the shakeaway **Width** and **Offset**.
3. If desired, select **Automatic Creation** and enter the shakeaway distance.
4. Alternatively, click **Add New**. Select a shake-away location (chain endpoints are not allowed) within the selected chain of edges. Enter the shakeaway distance.
5. If desired, click **Delete** and select one of the following:
   - **Delete Sel.**—Delete Shakeaway point by selection.
   - **Delete Last**—Delete latest created shakeaway point.

- **Delete All**—Delete all shakeaway points.

6. Click **Done**.

To Specify the Approach and Exit Motion

1. From the **NC Seq** tabbed page, select the required NC sequence.
2. Click **OPERATE** > **Redefine**. The **SMT NC SEQUENCE** dialog box opens.
3. Select **Approach/Exit** and click **Define**. The **APPROACH/EXIT MOTION** dialog box opens. Select one or more of the following options:

- Click the type of **Lead In** position that you want.
  - **First Point of Start Edge**—Starts the cut at the first point of the start edge.
  - **Middle Point of Start Edge**—Starts the cut in the middle of the start edge.
- Click the type of **Approach Motion** that you want, and specify the required values, if any, into the corresponding boxes.
  - **None**—Uses the default approach motion.
  - **Linear**—Sets the desired approach distance and angle.
  - **Tangent**—Sets the desired approach radius and angle.
- Click the type of **Exit Motion** that you want, and specify the required values, if any, into the corresponding boxes.
  - **None**—Uses the default approach motion.
  - **Linear**—Sets the desired approach distance and angle.
  - **Tangent**—Sets the desired approach radius and angle.

4. Click **Preview** to view the results.
5. Click **Done**.

Dimension Patterns

Dimension patterns of Contouring NC sequences can be created if the NC sequence contains a cut motion with dimensions that can be used to drive the pattern.

Reference Patterns

You can create a reference manufacturing pattern when the NC sequence to be patterned references a patterned feature. The functionality is similar to creating Reference patterns of features in regular Pro/ENGINEER.

NC sequences with a sketched cut section may be Reference patterned only if the section sketch for the NC sequence is created with **Use Edge** and references edges of a patterned feature.

After you pattern an NC sequence (using either pattern type), an Automatic material removal feature based on it can be reference patterned.

Autocontouring

About Autocontouring

Autocontouring is an advanced system of sheet metal contouring. Autocontouring allows you to automatically create laser or flame contouring NC sequences to cut all the features of single or nested parts. The workcell parameters in Autocontouring are used to automatically create NC sequences. These NC sequences include approach and exit motions, corner conditions, and shakeaways that can be manually manipulated.

Example: Approach punches can automatically be added at pierce locations for approaches or shakeaways.

To Create an Autocontouring Sequence

1. From the **SMT MFG MACHINING** dialog box, click **NC Seq**.
2. Click **Auto** > **Contouring**. You must be in a Contouring or Hybrid workcell. The **SMT NC SEQUENCE** dialog box and the **CONTOUR SELECTION** dialog box open.
3. Select one or more of the following options from the **CONTOUR SELECTION** dialog box:
    - **Selection Method**—Select **Automatic** or **Manual** contouring method.
    - **Show**—Show selected contours.
    - **Use Contours**—Select to process **Inner** or **Outer** contours.
    - **Use Offset**—Click this check box to use offset.
    - **Total**—Displays the number of contours found.
    - **Selected**—Displays the number of contours selected.
    - **Start Edge**—Click **Select** and select the required edge, chain, or loop, and click **Done**.
4. To use any of the optional elements from the **SMT NC SEQUENCE** dialog box, select the one you want, and click **Define**. Select one or more of the following options:
    - **Approach/Exit**—Specify the approach and exit motions.
    - **Corner Conditions**—Add a corner condition for NC Sequence.

- **Shakeaway Points**—Specify the shake-away points.
- **Cutter compensation**—Specify the cutter compensation.

5. Click **Done**.

To Specify Cutter Compensation

1. In the **SMT NC SEQUENCE** dialog box, click **Cutter compensation**, and click **Define**. The **CUTTER COMPENSATION** dialog box opens.
2. Click the **Use CutCom** check box.
3. Specify values for **Register**, **Tangent**, and **Normal**.
4. Click **Done**.

Shear NC Sequence

About Shear NC Sequences

The Shear NC sequence creates a point-to-point tool path along a sketched contour. The tool path is broken down into smaller elements called strokes that in turn are broken down into hits. Hits are the smallest elements of the tool path. Hits indicate each location where the tool makes contact with the workpiece and removes material. A succession of hits along an entity of the sketched contour composes a stroke. When the tool path is generated, strokes appear as a series of hits connected by a line. The distance between the hits is calculated based on the actual tool size.

Use the `OVERHANG_DIST` parameter to specify the distance by which each hit overlaps the previous one. By default, the tool path along each entity in the sketched contour is equal to the length of the entity. You can specify shake-away points or vertices and overhang vertices, where the tool path is, respectively, trimmed or extended by a specified distance, similar to Nibble Edge NC sequence. These distances are defined by NC sequence parameters `SHAKEAWAY_DIST` and `OVERHANG_DIST`.

Shearing NC sequences can be performed using either a Standard Punch type tool, or a special Shear type tool, that has width equal to zero. Use the `CYCLE/SHEAR` command to output the tool path. Other manufacturing parameters for Shear NC sequences are similar to those for Nibble Edge.

To Create a Shear NC Sequence

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Click **New > Shear**. The **SMT NC SEQUENCE** dialog box opens. The following options are available:
   - **Name**—Enter a name for the NC sequence. This name replaces the default NC sequence name.
   - **Parameters**—Specify manufacturing parameters.

- **Comment**—Use the system editor to enter comments about the NC sequence.
- **Info**—Displays a summary of NC sequence and parameter information.
- **Next**—Go to the next NC sequence.
- **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.

3. From the **TURRET SETUP** dialog box, define the tool and click **Done**. The sketcher window appears.
4. Sketch the shear NC sequence location. The sketch must contain a single continuous non-intersecting chain of entities, open or closed. If you are using a Shear type tool, the sketched contour must consist of straight lines only. Select a reference in the **References** dialog box. Click **Done**.
5. To use any of the optional elements, choose the one you want, then click **Define**. The following options are available:
   - **Shake Away**—Specify the shake-away points.
   - **Overhang**—Specify the overhang points.
   - **CL Command**—Insert CL commands at selected hits.
6. Click **Done**.

To Specify a Shakeaway Definition

1. From the **SMT NC SEQUENCE** dialog box, click the **Shake Away** element and click **Define**. The **Shakeaway Definition** dialog box opens.
2. Click **Add**.
3. Select a vertex within the selected chain of edges. You cannot select chain endpoints. The **Length Ratio** of the vertex is displayed.
4. If required, you can change the length ratio of the selected vertex.
5. Type the **Shakeaway Distance**.
6. Click **OK**.

Slitting NC Sequences

About Slitting NC Sequences

The Sheet Slitting NC Sequence allows you to cut scrap sheetmetal material into manageable pieces for unloading. After you finish cutting and punching the required parts from a sheet, a big flimsy remnant of unused material remains on the machine. This unused material may contain sharp corners and edges and may be difficult and dangerous to handle.

Using the Laser Slitting Sequences, you can create straight line cuts on the unused material and break the material into smaller, manageable pieces.

To Create a Slitting NC Sequence

1. In the **SMT MFG MACHINING** dialog box, click **NC Seq**.
2. Click **New** > **Slitting**. The **SMT NC SEQUENCE** dialog box opens.
3. Select one or more of the following options from the **SMT NC SEQUENCE** dialog box:
   - **Name**— Specify a name for the NC sequence. This name replaces the default NC sequence name.
   - **Parameters**—Specify manufacturing parameters.
   - **Comment**—Specify comments about the NC sequence.
   - **UD Pprint**—Specify user defined PPrint statement
   - **Info**—Display a summary of the NC sequence and parameter information.
   - **Next**—Go to the next NC sequence and not Slitting NC Sequence.
   - **Preview**—View the NC sequence using the **SMT MFG NCL PLAYER**.
4. To define a tool for the Slitting NC Sequence, select **Tool** in the **SMT NC SEQUENCE** dialog box and click **Define**. The **TURRET MANAGER** and **TOOL SETUP** dialog boxes open.
5. Select the **Laser** type of tool and click **Done**.
6. Select **Cut Lines** in the **SMT NC SEQUENCE** dialog box and click **Define**. The **SMM SLIT DEFINE** dialog box opens.
7. Specify cut lines to slit the scrap material.

   Use the following to provide details about the cutting lines along which the laser slitting should occur:

   - **Cut Axis**—Defines the slitting lines orientation along the x- or y-axis.
   - **Stock Allow**—Specifies the permissible amount of stock distance. The slitting occurs at the specified distance away from the cut parts. This is to avoid any damage to the cut parts.
   - **Definition**—Defines the cut lines to create a slitting tool path. You can define the cut lines by using **by number**, **manually**, or **by offset**.

     The default is **by number**. In the **Number of slits** box, type a value to specify the number of cut lines. A Slitting NC Sequence is created with the specified number of equally spaced cut lines.

     To define the offset distance between cut lines, select **by offset** in the **Definition** box. Type a value in the **Offset between slits** box to specify the distance between the cut lines. A Slitting NC Sequence is created with

the specified offset distance between consecutive cut lines and from the first edge of the sheet. The number of cut lines are calculated based on the specified offset distance and the sheet size.

To specify cut lines at the desired locations on the sheet, select **manually** in the **Definition** box. A table appears at the bottom of the **SMM SLIT DEFINE** dialog box listing each cut line to be created. Specify details about the cut line and the offset distance between each of them in the table. Use the thumbwheel to adjust the value.

Use  or  to add or remove items, respectively, from the table. Click **OK**.

8. Select **Scan Type** in the **SMT NC SEQUENCE** dialog box and click **Define**. The **SMM SCAN TYPE** dialog box opens.
9. Define a direction for the slitting tool path scan using the following options:
   - **Move Directions**—Specifies the scan direction while slitting.

     One—Scans in one direction only while slitting.

     Both—Scans in both directions wile slitting.
   - **Start Corner**—Select a corner for starting the scan for slitting.
10. If required, click **CL Command** and click **Define**. This inserts new CL commands at the selected points.
11. Click **Done**.

Manufacturing Parameters

Setting Parameters

You can access manufacturing parameters, by clicking **MFG PARAMS** > **Options**. You can select these parameters when you create, modify or redefine an NC sequence.

You can either set all the parameters for an NC sequence manually, one-by-one, automatically, or initialize the parameters file from the database.

To Initialize the NC Sequence Parameters

There are several ways to initialize the NC sequence parameters:

- If a site, corresponding to the type of the NC sequence, has been activated (whether by associating it to the workcell or by using the **Activate** option), it is automatically used to initialize the parameters. You can use the **Site** option in the **Mfg Params** menu to retrieve parameters of another site appropriate for the current NC sequence. Note that retrieving site parameters does not activate the site for subsequent NC sequences.

- If parameters have been set at the operation level, they are carried over to the NC sequence level automatically.
- Retrieve an existing NC sequence-specific parameters file. Click **Retrieve** from the **Mfg Params** menu and select a file name from the **Data Files** menu. This menu contains all the appropriate type files in the current directory, or in the library directory, if set. If the file resides in another directory, select Names and enter path and name for the file to be retrieved. The search will be started in the `pro_mf_param_dir` directory, if set; otherwise, in your current working directory.

  Note: If the retrieved file contains some parameters inapplicable to the current NC sequence, these parameters are listed in the INFO window.
- Retrieve the set of parameters used for the previous NC sequence. The **Use Prev** option is available only after you have created at least one NC sequence.

  Note: You can use the **Set** option at any time to modify the parameter values.

Parameter Inheritance

All levels of manufacturing features automatically inherit their parameters from an upper-level feature, unless you explicitly modify (customize) a parameter at the current level. That is, Cut Motions and Tool Motions inherit their parameters from the parent NC sequence, while NC sequences may (under certain circumstances) inherit their parameters from sites.

The following the rules apply to the NC sequence parameter inheritance:

- If, at the time of creating an NC sequence, a site of appropriate type is associated with the current workcell, the NC sequence will inherit the parameters of this site. This is called implicit inheritance. If you later modify the parameters of the site, the NC sequence parameter values will be updated accordingly. Also, if you later replace the associated site with another one, or change the workcell associated with the operation, the NC sequence parameters will update to the values in the new site associated with the workcell.
- If you initialize the NC sequence parameters using a site other than the one associated with the workcell, then the NC sequence will inherit the parameters of this site. This is called explicit inheritance. If you later modify the parameters of the site, the NC sequence parameter values will be updated accordingly. However, changing the site associated with the workcell, or the workcell itself, will not affect the parameters of this NC sequence.
- If you initialize the NC sequence parameters using the **Retrieve** option in the **SMM PARAMETERS** dialog box, these parameters will be marked as customized.
- Click the **Parameters** option from the **SMT NC SEQUENCE** dialog box. The **SMM PARAMETERS** dialog box appears. If you modify parameter values using this dialog box, parameters will be marked as customized.

Parameters marked as inherited will updated automatically all the way down if you change a parameter value at the upper level. However, if you have explicitly modified a parameter value at a certain level, this parameter will be marked as customized and will not change when you modify parameters at an upper level.

Customized parameters can also be inherited by the lower-level features. In other words, if you modify `CUT_FEED` at the NC sequence level, it will no longer change if you later modify `CUT_FEED` in the parent site file, but the Cut Motion `CUT_FEED` will update to the new NC sequence value (unless you explicitly customize it at the Cut Motion level, too).

When you look at the parameters of an NC sequence, Cut Motion, or Tool Motion, the inherited parameter values appear in parentheses. However, if you save a parameters file to disk, this file will list all the parameters without parentheses (because when you retrieve this file into another NC sequence or Cut Motion, these parameters will be marked as customized).

Modifying the NC Sequence Parameters

The **Parameters** option from the **SMT NC SEQUENCE** dialog box enables you to set or modify parameters for the NC sequence. If you have initialized the parameters, all or some of the parameter values are filled. You can then modify them to suit this specific NC sequence.

To Include a Parameter in a Relation

1. From the **SMT MFG MACHINING** dialog box, click **Setup > Relations**. The **Relations** menu appears.
2. Under **Look in**, select the required element.
3. Select the reference part or the workpiece or both to display dimensions in symbolic form.
4. Click **Parameter** > **Add Parameter** and specify the relation using the required parameters, dimensions, or expressions.
5. Click **OK**.

To Set or Modify NC Sequence Parameters

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select the desired NC sequence.
3. Click **OPERATE > Redefine**. The **SMT NC SEQUENCE** dialog box opens.
4. Click **Parameters**. The **SMM PARAMETERS** dialog box opens with a complete set of parameters corresponding to the NC sequence type. The parameters are listed under branches that can be collapsed or expanded.
5. Select the desired parameter. The value appears in the input panel at the top of the dialog box. Depending on the parameter type, do the following:

- If the parameter has a numeric value, enter a new value and press `<CR>`.

  You can enter a mathematical expression, which may contain other parameters in the same NC sequence. For example, if you enter:

```
OVERLAP_DIST LENGTH * 0.2
```

the overlapping distance is based on the size of the current tool (if the tool is later changed, OVERLAP_DIST is also updated).

You can also include relations, model dimension symbols (in assembly format, e.g., *d12:0*) and user-defined parameters that are already defined in relations for the model.

- When modifying a parameter that has a string value, such as SCAN_TYPE, select the arrow to the right of the input panel. A drop-down list appears with all possible values for this parameter. Scroll to the value you want (to display it in the input panel), then press <CR>.

6. To save changes, click **FILE > Save**.
7. Click **FILE > Done** to finish.

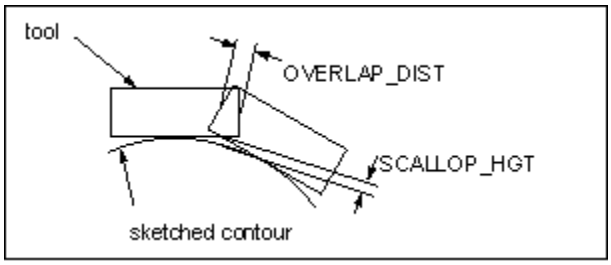
Shear Specific Parameters

Cut Parameters

SCALLOP_HGT

Together with OVERLAP_DIST, can be used to control the next tool position with respect to the previous one (see SCALLOP_HGT and OVERLAP_DIST). The default SCALLOP_HGT is 0, in which case the respective tool positions are defined by the OVERLAP_DIST parameter value. If you specify a non-zero value for SCALLOP_HGT, the system computes the overlap distance using this value, then compares it with the OVERLAP_DIST value you specified, and uses the lesser of the two.

SCALLOP_HGT and OVERLAP_DIST



OVERLAP_DIST

Defines the overlapping of successive tool hits. Specify OVERLAP_DIST as an absolute value in the units of the workpiece. The default is 0.

MAX_OVERLAP_DIST

Enables you to specify the maximum overlapping of successive tool hits. Specify as an absolute value in the units of the workpiece. The default is a dash (-),which means to ignore.

SHAKEAWAY_DIST

Defines the default distance to be left on both sides of a shake-away point or vertex. Specify `SHAKEAWAY_DIST` as an absolute value in the units of the workpiece.

OVERHANG_DIST

Defines the default distance to go past the edge on each side of an overhang vertex.

Punch Specific Parameters

Cut Options

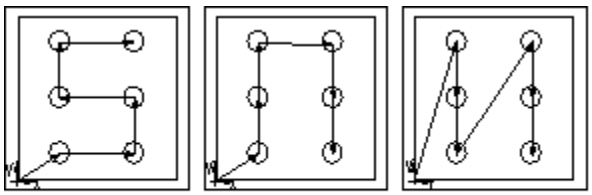
SCAN_TYPE

Defines how the system scans the punch instances:

- `TYPE_1` (the default)—Increment the y coordinate and goes back and forth in the x direction.
- `TYPE_SPIRAL`—Clockwise, starts from the instance nearest to the coordinate system.
- `TYPE_ONE_DIR`—Increment the x coordinate and decrements the y.

The scan algorithm applies to the whole workpiece. It does not matter if the punch instances belong to the same reference part or to different ones.

SCAN_TYPE



TYPE_1 TYPE_SPIRAL TYPE_ONE_DIR

Machine

TOOL_ORIENT

Defines the angular orientation of the punch tool when creating a Punch NC sequence referencing datum points. The default value is `0`.

Contouring Specific Parameters

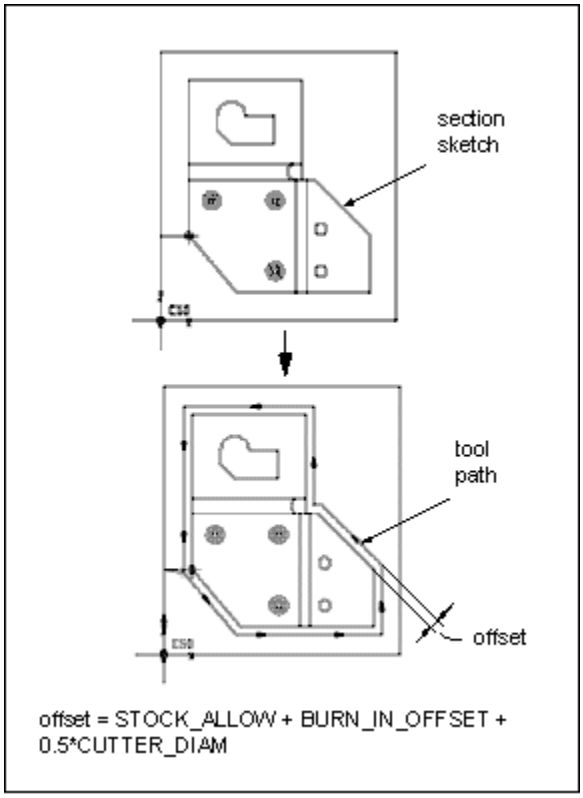
Feeds

- CUT_FEED—The rate at which the tool moves into the workpiece. The default CUT_FEED is -1.
- CUT_UNITS—IPM (inches per minute-default), FPM (feet per minute), MMPM (millimeters per minute), FPR (feet per revolution), IPR (inches per revolution), MMPR (millimeters per revolution).
- ARC_FEED—Enable you to control the cut feed around arcs. If set to a dash (-) (the default), the CUT_FEED value is used. If set to 0, the RAPID statement is output before the CIRCLE statement.
- FREE_FEED—The feed rate used for rapid traverse (RETRACT_UNITS are used for rapid feed rate units). If FREE_FEED is set to 0, the RAPID command is output to the CL file. The default FREE_FEED is a dash (-), in which case the CUT_FEED is used.

Cut Params

- STOCK_ALLOW—The amount of stock left after the NC sequence. The default is -1.
- SHAKEAWAY_DIST—Define the distance to be left on each side of a shake-away point or vertex. SHAKEAWAY_DIST is specified as an absolute value in the units of the workpiece.
- BURN_IN_OFFSET—Together with the tool parameter CUTTER_DIAM, defines the trajectory of the center of the tool with respect to desired geometry. The default is 0.

Offsetting the Tool Path



- CORNER_LENGTH—Define the size (the side length) of the parallelograms added to the tool path when corner condition is specified. The default is 0.
- CORNER_ANGLE—For a sharp corner, the angle of the parallelogram equals the angle between the adjourning entities, and the CORNER_ANGLE value is ignored. However, if you add a corner condition between two tangent entities, the angle of the parallelogram is defined by the CORNER_ANGLE parameter. The default is 0.
- CONCAVE_RADIUS—The radius for a round corner condition at a concave corner.
- CONVEX_RADIUS—The radius for a round corner condition at a convex corner.
- CHAMFER_DIM—The size of a chamfer for a chamfer corner condition.

Machine

- COOLANT_OPTION—ON, OFF (default), FLOOD, MIST, TAP, or THRU.
- COOLANT_PRESSURE—NONE (default), LOW, MEDIUM, HIGH.

CIRC_INTERPOLATION

Specifies the format in which the tool path along a circular edge is output to the CL file. The options are:

- POINTS_ONLY (default)—Use this format for machines that have no interpolation. The tool just moves along the edge, according to the tolerance value.

- ARC_ONLY—Use this format for machines that have full circular interpolation. Only the CIRCLE statement and the minimum number of points necessary for post-processing are output to the CL file.
- POINTS_&_ARC—Use this format for machines that have incremental circular interpolation. The CIRCLE statement and the maximum number of points according to the tolerance value are output to the CL file.
- APT_FORMAT—Use it if your post-processor expects the CIRCLE statement to be in the APT format. The default format for the CIRCLE statements in Pro/MANUFACTURING is different from the APT format (see the following illustration).

CIRCLE Statement Formats

Default format

```
CIRCLE / 50.0000000000. 40.0000000000.$
-0.0000000000,-0.0000000000,-1.0000000000,5.0000000000
GOTO / 46.0956559528, 36.8765247622, 0.0000000000,$
-0.0000000000,-0.0000000000,-1.0000000000
45.4973889514. 37.8259499213. 000.0000000000.$
-0.0000000000, -0.0000000000, -1.0000000000
       45.1259331538, 38.8848890735, 0.0000000000,$
-0.0000000000. -0.0000000000. -
       45.0000000000, 40.0000000000, 0.0000000000,$
-0.0000000000, -0.0000000000, -
```

APT format

```
INDIRV / -0.598267. 0.949425.
TLO
GOFWD/(CIRCLE/50.000000,40.000000,0.000000,5.000000),$
  ON, (LINE /50.000000,40.000000, 0.000000, $
            45.000000, 40.000000, 0.000000)
```

- NUMBER_OF_ARC_PTS—Specify the number of points to be output to the CL file if CIRC_INTERPOLATION is set to ARC_ONLY. The default is 3.
- CUTCOM—Control tool compensation. Does not reverse material side if you reverse direction. The options are:
  - ON—turns on the tool compensation in the CL file. The compensation is Right or Left, depending on CUT_TYPE and SPINDLE_SENSE.
  - OFF (the default)—no tool compensation provided. CUTCOM statements are not output for cut motions.
  - Entry/Exit
- APPROACH_DISTANCE—Specifythe length of approach motions for Contouring NC sequences. The default is 0.
- EXIT_DIST—Specify the length of exit motions for Contouring NC sequences. The default is 0.

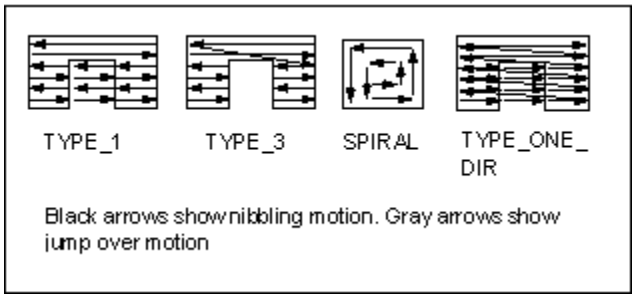
- ENTRY_ANGLE—Specifythe entry angle for a linear lead in, and the arc angle for a radial lead in.
- EXIT_ANGLE—Specifythe exit angle for a linear lead out, and the arc angle for a radial lead out.
- ENTRY_RADIUS—Specify the radius for a radial lead in.
- EXIT_RADIUS—Specify the radius for a radial lead out.

Nibble Area Specific Parameters

Cut Options

- SCAN_TYPE—Refer to the way the tool scans the sketched section and avoids islands:
  - TYPE_1—The tool continuously moves across the section, jumping over encountered islands.
  - TYPE_3—The tool scans continuous zones defined by the island geometry in turn, moving around the islands. Upon completing one zone, the tool may jump over to nibble the remaining zones.
  - TYPE_SPIRAL—Generate a spiral cutter path.
  - TYPE_ONE_DIR—The tool moves in one direction only. At the end of each cutting pass it retracts and jumps over to the opposite side of the section, to start the next pass in the same direction. Avoiding islands is the same as in TYPE_1.

SCAN_TYPE



- ROUGH_OPTION—Control whether a profiling motion occurs during a Nibble Area NC sequence. The options are:
  - ROUGH_ONLY (the default)—Create an NC sequence where the tool scans the whole area with no profiling.
  - ROUGH_&_PROF—Create an NC sequence that scans the whole area, then profiles the contour.
  - PROF_ONLY—Only profiling is done. In this case, the STEP_OVER and STEP_OVER_ADJUST parameters are not used; the whole profile is machined

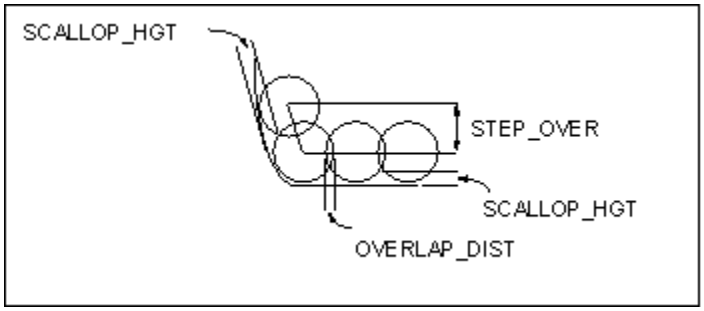
using the SCALLOP_HGT, OVERLAP_DIST, and MAX_OVERLAP_DIST parameters.

- STEPOVER_ADJUST—If set to YES (the default), adjusts the step-over distance during the NC sequence to make both the beginning and the end of the cutter path close to the sketched boundaries. The adjusted step-over distance does not exceed the STEP_OVER value. If set to NO, no adjustment is made.

Cut Params

- STEP_OVER—Control the distance between the strokes (see SCALLOP_HGT and OVERLAP_DIST). The default STEP_OVER is -1.0.
- PROF_OVERLAP_DIST—Define the overlapping of successive tool hits for profile nibbling. PROF_OVERLAP_DIST is specified as an absolute value in the units of the workpiece. The default is 0.
- ROUGH_OVERLAP_DIST—Define the overlapping of successive tool hits for rough nibbling. ROUGH_OVERLAP_DIST is specified as an absolute value in the units of the workpiece. The default is 0.
- PROF_SCALLOP_HGT—Can be used to control both the distance between the strokes and between the hits within a stroke for profile nibbling. The default PROF_SCALLOP_HGT is 0, in which case the distance between the strokes is controlled by the STEP_OVER parameter, and the distance between the hits is controlled by the PROF_OVERLAP_DIST parameter. If a non-zero value is specified for PROF_SCALLOP_HGT, the system computes the step-over distance and the overlap distance using this value, then compares, respectively, with STEP_OVER and PROF_OVERLAP_DIST you specified, and uses the lesser value for each case.
- ROUGH_SCALLOP_HGT—Can be used to control both the distance between the strokes and between the hits within a stroke. The default ROUGH_SCALLOP_HGT is 0, in which case the distance between the strokes is controlled by the STEP_OVER parameter, and the distance between the hits is controlled by the ROUGH_OVERLAP_DIST parameter. If a non-zero value is specified for ROUGH_SCALLOP_HGT, the system computes the step-over distance and the overlap distance using this value, then compares, respectively, with STEP_OVER and ROUGH_OVERLAP_DIST you specified, and uses the lesser value for each case.

SCALLOP_HGT, STEP_OVER, and OVERLAP_DIST



Nibble Edge Specific Parameters

Cut Options

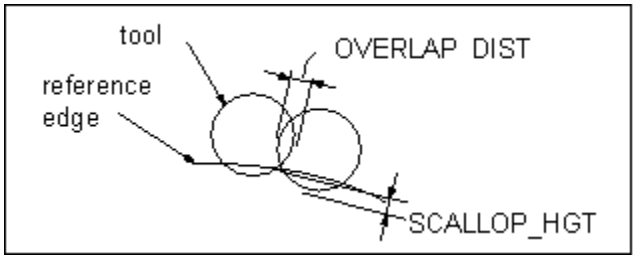
- `OUTPUT_TYPE`—Affects the CL Data output. If the value is `POINT_BY_POINT` (the default), the cutter path is output as a sequence of `GOTO` statements. If the value is `CYCLE_NIBBLE`, the tool path for linear entities is output using the `CYCLE/NIBBLE` commands. For example, the cutter path for a single straight edge is output as follows:
  - `CYCLE / NIBBLE, n—(where n - number of hits, i.e., number of punches to be made from the previous hit position to the position specified by the subsequent GOTO statement)`
  - `GOTO / x1, y1, z1—(to the beginning of the entity)`
  - `GOTO / x2, y2, z2—(to the end of the entity)`
  - `CYCLE / NIBBLE, OFF`

  Once a `CYCLE/NIBBLE` command is encountered, the same number of hits are used for each subsequent `GOTO` statement until another `CYCLE/NIBBLE` command or `CYCLE/NIBBLE`, `OFF` is reached.

Cut Params

- `NE_SCALLOP_HGT`—Together with `OVERLAP_DIST`, `SCALLOP_HGT` can be used to control the next tool position with respect to the previous one (see `SCALLOP_HGT` and `OVERLAP_DIST`). The default `SCALLOP_HGT` is 0, in which case the respective tool positions are defined by the `OVERLAP_DIST` parameter value. If a non-zero value is specified for `SCALLOP_HGT`, the system computes the overlap distance using this value, then compares it with the `OVERLAP_DIST` value you specified, and uses the lesser of the two.
- `NE_OVERLAP_DIST`—Define the overlapping of successive tool hits. `OVERLAP_DIST` is specified as an absolute value in the units of the workpiece. The default is 0.

SCALLOP_HGT and OVERLAP_DIST

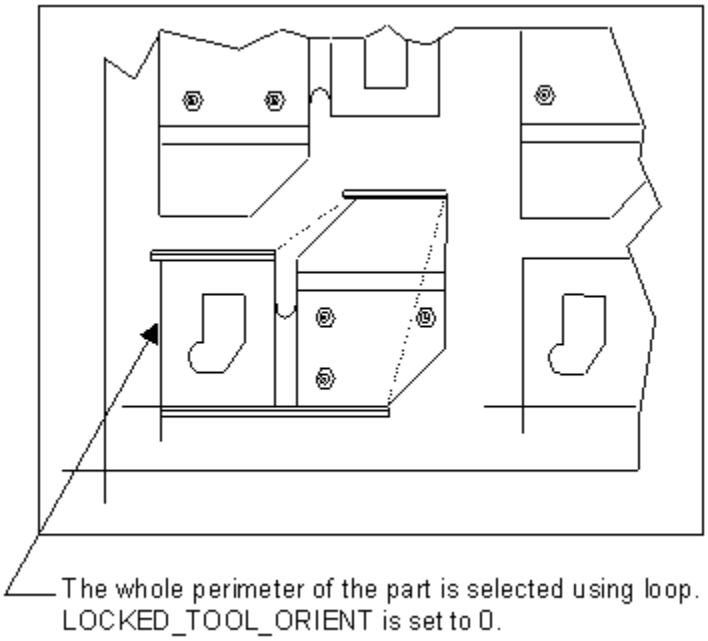


- MAX_OVERLAP_DIST—Enable you to specify the maximum overlapping of successive tool hits. It is specified as an absolute value in the units of the workpiece. The default is a dash (-), which means to ignore.
- NE_SHAKEAWAY_DIST—Define the default distance to be left symmetrically on *both* sides of a shake away point or vertex. This value can be modified individually for each shake-away point or vertex. SHAKEAWAY_DIST is specified as an absolute value in the units of the workpiece.
- NE_OVERHANG_DIST—Define the default distance to go past the edge on *each* side of an overhang vertex. This value can be modified individually for each overhang vertex. OVERHANG_DIST is specified as an absolute value in the units of the workpiece.

Machine

- NE_LOCKED_TOOL_ORIENT—If the parameter is set to any value other than a dash (-) (the default), only linear entities that require tool orientation equal to this value are machined. The parameter value represents the angle to the X-axis of the Machine Csys. For example, to machine all edges parallel to the X axis of the Machine Csys, set LOCKED_TOOL_ORIENT to 0.

Machining Edges with Specified Orientation

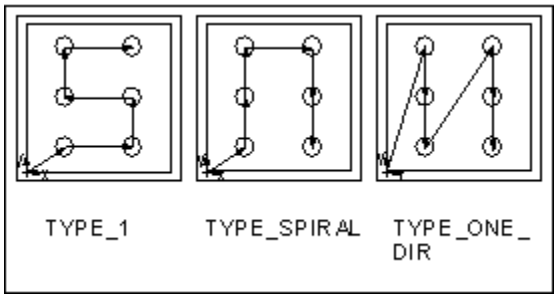


Form Specific Parameters

Cut Options

- SCAN_TYPE—Define how the system scans the form instances:
  - `TYPE_1` (the default)—Increments the y coordinate and goes back and forth in the x direction.
  - `TYPE_SPIRAL`—Clockwise, starts from the instance nearest to the coordinate system.
  - `TYPE_ONE_DIR`—Increments the x coordinate and decrements the y.
  - The scan algorithm applies to the whole workpiece. It does not matter if the form instances belong to the same reference part or to different ones.

SCAN_TYPE



Using Parameters in Relations

You can drive NC sequence and tool parameters by other parameters or part dimensions using relations.

The NC sequence parameter syntax is:

Parameter syntax	Where	Example
PARAM_NAME:FID_#	PARAM_NAME—the parameter name #—the internal feature ID of the NC sequence.	STEP_OVER:FID_22 = d6:1 * 0.1
PARAM_NAME:TID_ID	PARAM_NAME—the parameter name, ID—the TOOL_ID.	STEP_OVER:FID_22 = CUTTER_RADIUS:TID_NIB 1 * 1.1

If a parameter is driven by a relation, the system puts a note next to it in the parameters file. If you modify a relation-driven parameter, the parameter value is reset upon regeneration.

Options **Tool Info** and **NC Seq Info** in the Relations menu, accessible in Manufacture mode only, facilitate using tool and NC sequence parameters in relations. They display parameters (including ID) for a selected tool or NC sequence, respectively.

Parameters Library

Your NC sequence parameters can be stored in a parameter library. They are available to all users for retrieval into their manufacturing processes, to modify, or set up new NC sequences. The configuration file option to use is:

```
pro_mf_param_dir <pathname>
```

Always enter the complete pathname to the parameter library in the configuration file to avoid problems when you work in different directories with Pro/ENGINEER. For example:

```
pro_mf_param_dir /usr/users/params
```

Workcell Parameters

Workcell parameters can be set, saved, and retrieved using the same procedures as for NC sequence parameters. The default extension for a workcell parameter file is .cel. The workcell parameters are the same for all workcell types. When you choose **Retrieve** from the **Mfg Params** menu, only files with .cel extension appear in the list.

The workcell parameters (except AUTO_POPULATABLE and TOOL_CHG_COMMAND) are for information purposes only; they are not used for setting the NC sequence parameters or CL data output.

The following parameters can be specified for a workcell:

- MACHINE_ID—The machine ID. The default is a dash (-).
- LOCATION—The machine location. The default is a dash (-).
- CONTROLLER—The controller name. The default is a dash (-).
- RAPID_FEED_RATE—The feed rate used for rapid traverse. The default is a dash (-).
- RAPID_FEED_UNIT -- IPM (default), FPM, MMPM, FPR, IPR, MMPR.
- TOOL_CHANGE_TIME—Time needed for changing a tool. The default is a dash (-).
- CELL_X_MIN —Minimum size of a workcell along its x-axis.
- CELL_X_MAX—Maximum size of a workcell along its x-axis.
- CELL_Y_MIN—Minimum size of a workcell along its y-axis.
- CELL_Y_MAX—Maximum size of a workcell along its y-axis.
- CELL_MAX_FEED—Maximum allowable feedrate for the workcell.
- MAX_TURRET_SIZE—Maximum allowable turret size for the workcell.
- AUTO_POPULATABLE—If set to YES, all newly created NC sequences are automatically included in the Populate feature. The default is NO.
- TOOL_CHG_TYPE—Defines the tool change type for the workcell: TURRET (default) or CHANGER.
- TOOL_CHG_COMMAND—Specifies if tool rotation is to be considered a tool change command when minimizing tool changes. Note that this parameter only affects the default setting in the menu; you can make the appropriate selection directly at the time of tool path optimization, independent of the setting of this parameter. The values are LOAD (default) and LOAD_&_ROTATE.

  Values for CELL_X_MIN, CELL_X_MAX, etc., should be the actual dimensions that indicate the extent of the workcell relative to the machine coordinate system. For instance, if a workcell is 60 inches wide, and the origin of the machine coordinate system is located halfway between the ends, the value of CELL_X_MIN should be -30 and the value of CELL_X_MAX should be 30.

  If you display or otherwise output the CL data for an NC sequence that exceeds the limitations of the workcell in which it is defined, the Information Window appears. The window contains a list of the values of the limits that have been exceeded and their corresponding actual values in the manufacturing model. When you dismiss the Information Window, the CL data displays in its current state.

- FROM_OUTPUT—Specifies how the FROM statement will be output to an operation CL data file:
  - NO_FROM_S (default)—No FROM statements are output. If a From point is specified, its location is output as FREE_FEED, GOTO.

- FIRST_FROM—A FROM statement is output at the beginning of the file. It corresponds to the location of the From point, if specified, or to the first location on the tool path for the first NC sequence. All other NC sequences are added to the operation without a FROM statement. For a workcell with two heads, there will be two FROM statements: one for Head1 and one for Head2.
- ALL_SEQ_FROM_S—FROM statements are output at the beginning of each NC sequence. For the first NC sequence, this FROM statement corresponds to the location of the operation From point, if specified, or to the first location on the tool path for this NC sequence.

Common Parameters

About Common Parameters

Those that are NC sequence-specific are listed in the appropriate NC sequence sections. They are listed under a heading that corresponds to the menu options you use to set up the parameters.

Notes:

- You must supply a value for all parameters that have a default of -1.
- Some parameters may have a value of dash (-) which means that the parameter is not used. Usually a system default or another parameter is used instead. The dash (-) value is only acceptable for those parameters that have it as a default.
- Length units for NC sequence parameters (where applicable) are the same as the units of the workpiece. If the workpiece units are changed using the Same Size option (so that the dimension values are changed), all the appropriate parameters of currently existing NC sequences are also scaled.

Name

- MACH_NAME—The machine name as required in post-processing.
- MACH_ID—The machine ID as required in post-processing.
- NCL_FILE—The prefix given to the CL file generated by the NC sequence.
- PRE_MACHINING_FILE—Enter the name of the file you want to include at the very beginning of the CL file. The file must be located in your current working directory and have the extension .ncl. The default is a dash (-), i.e., none.
- POST_MACHINING_FILE—Enter the name of the file you want to include at the very end of the CL file. The file must be located in your current working directory and have the extension .ncl. The default is a dash (-), i.e., none.

Cut Options

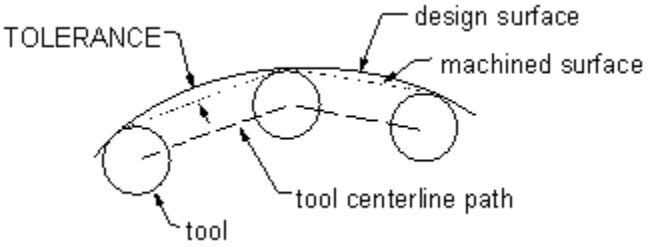
- APPROXIMATE_SPLINES—Indicate whether or not you want to approximate spline edges with arcs when creating CL data output for edge nibbling or contouring NC

sequences. The default value is NO; if you specify a value of YES, the spline edges are approximated.

Cut Params

- TOLERANCE—Machine tools move in small straight line increments to approximate curved geometry. The maximum distance that the straight line path deviates from the curved geometry is set by TOLERANCE. The default TOLERANCE is -1.

Cut Param Tolerance



Machine

- REQ_MACHINE_POWER—Specifies the amount of power necessary for an NC sequence, and can be later used in relations. The default is 0.

Note: The REQ_MACHINE_POWER parameter is inapplicable for Contouring NC sequences.

- TRAP_STATUS—Controls the output of the UNLOAD command to the NC sequence CL file. If TRAP_STATUS is set to ON, the UNLOADcommand is output at the end of the NC sequence. The default TRAP_STATUS is OFF.

Note: The TRAP_STATUS parameter is inapplicable for Contouring NC sequences.

- END_STOP_CONDITION—Specifies the stop command is issued at the end of the CL data output for an NC sequence:
- NONE (default)—No command is issued.
- OPSTOP—The OPSTOPcommand is issued.
- PROGRAM_STOP—The STOP command is issued.

CL Output and Menus

CL Commands

About CL Commands

The **SMM CL Command** dialog box enables you to include specific post-processor commands for NC output. These commands are output to the NC sequence CL file. If you add a tool motion command, the corresponding motion is shown in the tool path display on the screen.

When entering CL commands, you can enter parameters, preceded by an ampersand sign (&). If there is a corresponding parameter defined in relations, its value is used in the CL command. If the parameter is not found, the system prompts you to enter the type and value of the parameter, and this parameter is added to the relations. This way, the CL command can be changed at the top level (through Relations).

To Find a CL Command

1. Click the **NC Seq** tabbed page from the **SMT MFG MACHINING** dialog box. Select an NC sequence.
2. Select **CL Command** and **Define**. The **SMT CL COMMAND** dialog box appears.
3. From the **Command List** window, select the desired command ID.
4. Click **EDIT** > **Find**. The **FIND LINE** dialog box opens.
5. Enter a search string and click **Find**.

   Note: If a block of commands has been added, only the first command appears in the search list, followed by the number of lines in the block. For example, if you choose `FEDRAT/1.000000, IPR` (2 lines) from the **namelist** menu, locations of all two-line blocks that start with the `FEDRAT/1.000000, IPR` command are shown, even if the other commands in the block are different.

To Insert a New CL Command

1. From the **SMT NC SEQUENCE** dialog box, select the **CL Command** element and click **Define**. The **SMM CL COMMAND** dialog box opens.
2. Type the command name in the **Command ID** window.
3. Using the **Command Wizard**, select the desired commands. You can edit commands in the **Command Edit Area** window.
4. Place the CL command using the placement buttons. The following options are available:
   -  —Click the tool path to select the command placement. You can pick anywhere on the tool path as it is defined so far; note that you cannot click a cut motion that is not yet incorporated in the tool path.

- ◀—The CL command is output before the first cutter position of the NC sequence (or the first cutter position of the operation if you are optimizing an operation tool path).
- ▶—The CL command is output after the last cutter position of the NC sequence (or the last cutter position of the operation if you are optimizing an operation tool path).

5. Click **FILE** > **Save** to save the CL command.
6. Click **FILE** > **Quit** to finish.

Supported CL Data Commands

If you add multiple command lines at a certain point, they are treated as a "block" i.e., you are able to move, copy, or delete only the whole block of command lines.

You can specify only one CL command at the beginning or end of the tool path (it can have multiple lines). Therefore, if you repeatedly try to select the **Beginning** or **End** option, a message informs you that a command already exists at this location, and you can modify it. Use the **Command** option in the **MODIFY CMD** menu to add the appropriate lines to the existing CL command, as described below.

To Delete a CL Command

1. Click the **NC Seq** tabbed page from the **SMT MFG MACHINING** dialog box. Select an NC sequence.
2. Select **CL Command** and **Define**. The **SMT CL COMMAND** dialog box appears.
3. From the **Command List** window, select the CL command you want to delete.
4. Click **EDIT > Delete**. The CL command is deleted.

   Note: If the selected location contains a block of commands, the whole block is deleted.

To Modify a User-Defined CL Command

1. Select the **NC Seq** tabbed page form the **SMT MFG MACHINING** diaolg box. Select the required NC sequence.
2. From the **SMT NC SEQUENCE** dialog box, select the **CL Command** element and click **Define**. The **SMM CL COMMAND** dialog box opens.
3. Locations of the previously added user-defined CL commands are highlighted in cyan. Select the command to modify from the **Command List** window.
4. Edit the CL command as desired in the **Command Edit Area** window.
5. Click **FILE > Save** to save the CL command.
6. Click **FILE > Quit** to finish.

CL Data Files

About CL Data Files

Cutter Location (CL) data files are generated from the cutter paths specified within Sheet Metal Manufacturing NC sequences. Each NC sequence generates a separate CL file; you can also output a single file for a whole operation. These CL data files can then be passed to machine-specific or generic post-processors for NC tape generation or DNC communications.

Generating Machine Control Data (MCD) files is discussed in About Post Processing.

CL Data Library

You can automatically store your NC sequence CL data files in a CL data library. This serves as a central collection point for the CL files produced. You can retrieve the CL files from the CL data library post-process or re-display. The configuration file option to use is:

```
pro_mf_cl_dir <pathname>
```

Always specify the complete path name to the CL data file library to avoid problems when using Pro/ENGINEER in different directories.

CL Data Files

To access the CL data files, click CL Output from the **SMT MFG MACHINING** dialog box.

NC Aliases

The **NC Aliases** command in the **Setup** menu in the **SMT MFG MACHINING** dialog box enables you to establish aliases for CL commands. NC aliases are useful if the post-processor that you use does not support the default CL commands output by Pro/ENGINEER. The NC aliases substitute a command you provide into the CL data. The NC alias may also include additional CL data associated with the command, as well as dimensions, user-defined parameters, and text. NC aliases are saved in a file with the extension ".ncd." NC alias files are stored automatically whenever the manufacturing model is saved.

Four CL commands are supported for aliasing: `MFGNO`, `PARTNO`, `LOADTL`, `PIERCE` and `CLAMP`. You cannot enter additional CL commands into this list to assign them aliases. NC aliases employ functionality similar to that of drawing notes: you can specify dimensions, parameters, or other fields to be output in the CL command.

For example:

CL Command (system default)	**User Command (NC alias)**
LOADTL	MACRO_LOAD : &nc_full_arg with &d1 and

	&param1 and &param2

where:

- `LOADTL`— default CL command output by Pro/ENGINEER. For this example, the complete output statement is: `LOADTL / 3, LENGTH, 5.0.`
- `MACRO_LOAD`—CL command being substituted for `LOADTL`.
- `&nc_full_arg`— output that follows the / in the output. In this example, it is 3, `LENGTH`, 5.0 (see above). The individual fields in this string can be output using &nc_arg[arg_num] (with arg_num beginning at 1). For instance, &nc_arg2 would output `LENGTH`.
- `d1`— dimension value.
- `param1 / param2`—user-defined parameters; & indicates that their values should be displayed (as in drawing notes). For this example, the value of `param1` is `BLUE` and the value of `param2` is `GREEN`.

The output in the CL data file reads: `MACRO_LOAD : 3, LENGTH, 5.0`, with `10.00` and `BLUE` and `GREEN`

Note: When you use Pro/TABLE to define an NC alias, it must be contained on a single line.

To Save CL Data to a File

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select an NC sequence.
3. Click **CL Output**. The **SMT MFG NCL PLAYER** appears.
4. Click **FILE > Save**. The CL data is saved.
5. Click **Close**.

   Note: A CL data file extension can be set to something other than `.ncl`. Use the following configuration file option: `ncl_file_extension <file extension>` (without '.')

   File extension length can be up to three characters. If you specify a longer file extension, it is truncated and a warning is issued.

To View CL Data Files

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select the desired NC sequence.
3. Click **CL Output**. The **SMT MFG NCL PLAYER** appears.
4. Select **NC Program Text**. The NC sequence CL data is displayed.

To Convert CL Files

The Pro/ENGINEER editor can handle lines up to 80 characters long. Therefore, the `CIRCLE` statements and `GOTO` statements with an axis (i, j, k vector) are broken in two lines, with the continuation character ($) output automatically at the end of the first line.

Before sending CL files to the post-processor, you might need to convert them to the format without the continuation character using the command `pro_ncl_cvt`. To convert CL files:

1. In the directory where your CL file resides, type:

   `pro_ncl_cvt`

2. Type the CL file name.
3. Type a different name for the output file.

To Use Search in a CL Data File

1. From the **SMT MFG NCL PLAYER**, click **Edit** > **Find**. The **FIND LINE** dialog box opens.
2. Type the search string and click **Find**.

To Input a CL Data File and Display the Tool Path

1. From the **SMT MFG MACHINING** dialog box, select the **NC Seq** tabbed page.
2. Select an NC sequence.
3. Click **CL Output**. The **SMT MFG NCL PLAYER** appears.
4. Click **FILE > Open**. The **OPEN** dialog box opens.
5. Select the desired CL data file.
6. Click **Open**.

To Output CL Data Using Pro/BATCH

1. In your system window, type `[pro_batch]`. The system displays the **Pro/BATCH** dialog window.
2. Typethe name of the batch file and the command to use with Pro/BATCH (that is, the command you use to start up Pro/ENGINEER), in the appropriate input fields at the bottom of the window.
3. Click **File > Browse**. The system displays the **File Browser** window.
4. Set the filter in the **File Browser** window to manufacture file (`*.mfg`). Select the manufacturing model(s) and click **Add Objects**.
5. Add as many models as you want, then click **Close Browser**.
6. Highlight the manufacturing model entries in the Pro/BATCH dialog window.

7. From the main menu, click **Options > Set Action > clfile**. The system updates the Action column for the selected objects to **clfile**.
8. Highlight a model entry and click **Options > Set Option**. The system displays the **Set Option - clfile** dialog box.
9. Select type of output:

- By operation.
- By NC sequence.

10. If outputting by operation, enter operation name.
11. If outputting by NC sequence, select how you want to specify it:

- If selecting by name, click **By Name** and type the NC sequence name.
- If selecting by number, click **By Number** and type the NC sequence number.

12. Enter the name of CL file to output to.
13. Click **OK**.
14. Set up the other manufacturing models by repeating steps 8 through 13.
15. Save the batch file (**File > Save**).
16. Click **Schedule > Start** the task and specify the number of hours before the system starts executing the file. Click **OK**.
17. Click **File > Exit** to exit from the **Pro/BATCH** dialog window.

To Output CL Data with Rotation or Translation

1. From the **SMT MFG NCL PLAYER**, click **OPTIONS > Transform**. The **TRANSLATE/ROTATE** dialog box opens.
2. Click **Rotate** or **Translate**.
3. Type the value for rotation or translation and click **Add**.

   Note: CL data rotations and translations are cumulative. You can repeat Steps 2 to 3 as many times as necessary to change the CL output location.

Miscellaneous CL Output Information

About PPRINT Setup

You must set up a PPRINT table to output sheet metal model information to CL files. The PPRINT table contains all the items that can be output through PPRINT. The status value determines if you want to output an item or not. Entering a Print value will output the item. You can add an optional comment to be output along with the item.

Whenever you output CL data to file, the system checks the PPRINT table. If any flag is set to *yes* and the appropriate information is available, the corresponding PPRINT command outputs to the CL file.

1. Click **Setup > PPRINT** in the **SMT MFG MACHINING** dialog box to open the **SMM PPRINT SETUP** dialog box.

Use	To
	Set the status value to Print.
	Set the status value to Skip.
	Add a comment to the item.

To Set Up a PPRINT Table

From the **SMT MFG MACHINING** dialog box, click **SETUP > PPRINT**. The **PPRINT** menu appears with the following options:

- **Retrieve**—Retrieve an existing PPRINT table from the current working directory.
- **Save**—Save the current PPRINT table for later use. The system prompts you for the file name. The file has an extension .ppr and is stored in the current working directory.

Whenever you output CL data to a file, the system checks the PPRINT table. If any flag is set to `YES` and the appropriate information is available, the corresponding PPRINT command outputs to the CL file.

The following items are output once per CL file:

- `PART_NAME`
- `DATE_TIME`
- `SCALE`
- `TRANSLATE`
- `ROTATE`

The following items are output once per operation:

- `OPERATION_NAME`
- `OPERATION_COMMENTS`
- `UDF_NAME`
- `TURRET`

The following items are output once per NC sequence:

- NC_SEQUENCE_NAME
- FEATURE_ID
- SEQUENCE_TYPE
- CUT_FEEDRATE_&_UNITS
- ARC_FEEDRATE_&_UNITS
- FREE_FEEDRATE_&_UNITS
- SCAN_TYPE

The following items are output once per LOADTL or TURRET statement:

- TOOL_NAME
- TOOL_COMMENTS
- TOOL_PARAMETERS

The TOOL_COMMENTS string is obtained depending on the type of the tool:

- In case of a Standard Punch, from the TOOL_COMMENT parameter in the parameters file.
- In case of a Library part, from the part parameter TOOL_COMMENT.
- In case of a Form or a Punch, from the TOOL_COMMENT parameter of the referenced feature (Form or Punch/Notch).

To Copy an Existing CL Command to a New Location

1. Click the **NC Seq** tabbed page from the **SMT MFG MACHINING** dialog box. Select an NC sequence.
2. Select **CL Command** and **Define**. The **SMT CL COMMAND** dialog box appears.
3. From the **Command List** window, select the CL command you want to copy.
4. Click **EDIT > Copy**. A copy of the command, named **Nameless**, appears in the **Command List** window.
5. Name, edit, and place the CL command as desired.
6. Click **FILE > Save** to save the CL command.
7. Click **FILE > Quit** to finish.

To Change the CL Display Status

1. From the **SMT MFG NCL PLAYER**, click **Options**. The following choices are available:

- **Display Tool**—Select operations and NC sequences to display tool only.
- **Display Path**—Select operations and NC sequences to display path only.

2. The display is updated.

To Include Pre and Post Machining Files

You can use parameters to specify the pre and post files. Under the **Oper** tabbed page of the **SMT MFG MACHINING** dialog box, you can find these parameters. You can include user-defined macros, like setting the post-processor registers, at the very beginning and the very end of an NC sequence CL file, using the two parameters:

- `PRE_MACHINING_FILE`—Enter the name of the file you want to include at the very beginning of the CL file.
- `POST_MACHINING_FILE`—Enter the name of the file you want to include at the very end of the CL file.

Enter file names without the extension. If you specify the file in the `config.pro` file it should have the extension `.ncl`, or the default CL file extension.

The contents of these files are included in the CL file of the current NC sequence between `$$ –> BLOCK_START` and `$$ –> BLOCK_END`. When you use the Input option in the **CL DATA** menu to read a CL file, these contents are ignored.

Notes:

- Both of these files are included into a CL file before it goes to the post-processor.
- If these parameters are set at the operation level, pre- and post-machining files are added to each NC sequence included in the operation.

Automation Data Files

About Automation Data Files

You can save sheet metal automation data files to be used for autonesting and populating NC sequences. These automation files contain superfluous information not required for implementation. Therefore, you must structure these files before implementing all or some of the data into the **SMT MFG Automation** dialog box. A standard ASCII file will allow you to import data from external MRP systems. The following ASCII automation data files can be generated:

- Workpiece Size
- Control and default values
- Cluster Control
- Order List

Automated Data File Headings and Variables

Category Heading

Variable Name

Value

Description

WORKPIECE SIZE

Unit

Inch/ Foot / mm/cm/ m

Length

#.###

WP dimensions

Width

#.###

Thickness

#.###

Left

#.###

Border offsets

Right

#.###

Top

#.###

Bottom

#.###

Direction

Unused / X axis / Y axis

CONTROL AND DEFAULT VALUES

Quantity

#

Default quantities of parts

Offset

#.###

Outer offset

HoleOffs

#.###

Internal hole offset

Rotation

Any / #.##

Flip

Yes / No

Allow part flip

Grain

Yes / No

Align to grain

View

Front / Top / ISO / Default

Part

Yes / No

Show part polygon

Rough

Yes / No

Show part rough polygon

Outline

Yes / No

Show cluster outline

CLUSTER CONTROL

Usage

Yes / No

Allow true shape nesting

Length

#.###

Nesting dims

Width

#.###

Factor

#

Quality factor

Use Hole

Yes / No

Allow part-in-part nesting

Hole Area

#.###

Minimum hole area for part-in-part

WORKCELLS AND OPERATIONS

Workcell Name

Name of workcell to be used

Operation Name

Name of manufacturing operation

ORDER LIST

Mfg Name

Part Name

File name

Can be dxf file name

Id

Ordered

#

No parts to nest

Priority

Offset

#.###

Part Outer offset

HoleOffs

#.###

Part internal hole offset

Rotation

#.##

Part rotation

Flip

Yes / No

Allow part flip

Grain

Yes / No

Align part grain

Nested

#

Example: Sheet Metal Automation Data Files

The following are examples of sheet metal automation data files.

WORKPIECE SIZE

WORKPIECE				**BORDER**				**GRAIN**
Unit	Length	Width	Thickness	Left	Right	Top	Bottom	Direction
Inch	120.000	60.000	0.125	0.000	0.000	0.000	0.000	Unused

CONTROL AND DEFAULT VALUES

DEFAULT						**DISPLAY**			
Quantity	Offset	HoleOffs	Rotation	Flip	Grain	View	Part	Rough	Outline
1	0.000	0.000	Any	No	No	Front	Yes	No	Yes

CLUSTER CONTROL

CLUSTERING						
Usage	Length	Width	Factor	Use Hole	Hole Area	Complete
Yes	-	-	3	No	0.000	No

ORDER LIST

MAIN			
Mfg Name	Part Name	Id	Ordered
-	DESIGN_PART_1	-	24
-	DESIGN_PART_2	-	15
-	DESIGN_PART_DFX	-	25

CONTROL							RESULT
Priority	Offset	HoleOffs	Angle	Rotation	Flip	Grain	Nested
-	-	-	0.000	-	No	No	24
-	-	-	0.000	-	No	No	15
-	-	-	0.000	-	No	No	25

Interactive Path Control

About Optimizing the Operation Tool Path

The **Optimize** option in the **NC Seq** tabbed page enables you to interactively optimize the tool path for a whole operation:

You can optimize different portions of the operation tool path using different rules.

- The rules for automatic optimization include:
  - Specifying the starting position for each set
  - Minimizing tool change and tool rotations
  - Minimizing tool travel
  - Removing duplicate hits and cut lines
- The operation tool path can be sorted by:
  - Order of creation of NC sequences
  - Tool order
  - Part order

Any portions of the tool path that are not automatically optimized can be optimized manually using the **Adjust Path** functionality.

If a Populate feature has been created, Optimize operates on the result produced by Populate. It is therefore automatically reordered to come after Populate in the workpiece feature sequence.

Note: The Rules and Order Seqs options are mutually exclusive. If you select Order after the tool path has been optimized using Rules, you are prompted it you want to delete the rules, and visa versa.

To Manipulate Optimization Sets

1. Click **NC Seq** in the **SMT MFG MANUFACTURING** dialog box.
2. Click **Optimize** > **Create** or **Optimize** > **Redefine**. The **SMM OPTIMIZE** dialog box opens.
3. Use one or more of the following options to create and modify optimization sets:
   - **Remove Duplicate Hits**—Removes duplicate punches in the tool path.
   - **Remove Duplicate Cut Lines**—Removes duplicate laser cuts in the tool path.
   - **Create**—Defines a new optimization set.
   - **Delete**—Deletes a previously defined optimization set.
   - **Modify**—Modifies an existing optimization set using the **OPTIMIZE Set Edit** dialog box.
   - **Reverse**—Reverses the order of a selected optimization set.
   - **Order**—Reorders a selected optimization set.
   - **Adjust Path**— Use this functionality to manually optimize any portions of the tool path that are not automatically optimized.

     Remove Hits—Select hits using the **REMOVE HITS** dialog box. The selected hits are removed from the tool path.

     Reverse NC Seq—Select a tool path to reverse the NC sequence.

     CL Command—Select a CL Command from **SMM CL COMMAND** dialog box to adjust the tool path.

To Create an Optimization Set

1. From the **NC Seq** tabbed page, click **Optimize > Create**. The **SMM OPTIMIZE** dialog box opens.
2. Click **Create**. Click **Punch Optset** or **Contour Optset**. The **OPTIMIZE SET EDIT** dialog box opens. The dialog box lists NC sequence information and contains the following options:

- —Specify hits to include in the optimization set.

- —Specify hits to exclude from the optimization set.
- —Select hits by using the mouse on the screen.
- —Select hits by using the mouse in the editor window.
- **By Tool**—Select all hits a certain tool performs (the tool names are listed in a selection menu).
- **By Part**—Select all hits to machine a certain reference part. Use the GET SELECT menu options to select the part.
- **By Type**—Select all hits that belong to a certain NC sequence type. The NCS TYPES menu lists all Punch Press NC sequence types. Select the type you want.
- —Select all hits.

3. Click **Done** to finish defining the optimization set.

To Specify Rules for Automatic Optimization of a Set

1. From the **NC Seq** tabbed page, click **Optimize** > **Redefine**. The **SMM Optimize** dialog box opens.
2. Under **Optsets**, select a set or sets to optimize from the list. If one does not exist, click **Create** to define a new set.
   - To reorder the highlighted set or sets, click or in the **SMM Optimize** dialog box.
   - To define the optimization rules for the highlighted set or sets, click **Order**. The **SMM Order** dialog box opens.
3. Adjust the order as necessary by selecting the appropriate box or boxes:
   - **Manual Order**—Set priorities based on manual selections.
   - **Part Priority**— Set priorities based on type using the movement arrows. An item at the end of the list has minimal priority.
   - **Tool Change**—The system minimizes tool changes and group index (ROTATE /) moves within a tool change based on whether **Rotate&Load** or **Load Only** is selected. The **TOOL_CHG_COMMAND** workcell parameter value defines default selection.
   - **Tool Travel**—The system uses the concept of **Next Closest** to minimize tool motion between hits. Depending on the option selected, the system automatically goes to the next closest sequence, stroke, or hit.

   Note: You can inverse the order of Tool Change and Tool Travel by clicking .
4. Click **Specify** to further define the optimization rule behavior.

5. Click **OK**.

To Create or Redefine an Operation

1. Click **Oper** in the **SMT MFG MACHINING** dialog box.
2. If some operations have already been set up, a namelist menu appears with the names of all existing operations. The current operation is highlighted. Click **Operate** > **Redefine** to redefine the operation.
3. Click **Wrkcell** in the **SMT MFG MACHINING** dialog box. Select one or more of the following options:
   - **Create**—Set up a new workcell.
   - **Retrieve**—Retrieve a previously defined workcell into the manufacturing process.
   - **Csys**—Specify a coordinate system for CL data output.
   - **Comments**—Add operation comments (see below).
   - **Parameters**—Initialize parameters at the operation level. The parameters that you can initialize at this point are: `NCL_FILE`, `PRE_MACHINING_FILE`, and `POST_MACHINING_FILE`.
   - **Zones**—Specify machine zone for the workcell.
4. Click the **Wrkcell** tab in the **SMT MFG MACHINING** dialog box to choose a workcell associated with an active operation.
5. Select the required workcell.
6. Click the **Oper** tab. The workcell name is displayed next to the operation.

CL Command Syntax Menus

About CL Command Syntax Menus

Sheet Metal Manufacturing provides you with an ability to manually add commands to CL data when customizing the tool path for an NC sequence.

There are three ways to input a CL command:

- **Menus**—Build the command by selecting APT keywords from the system-supplied or user-defined menus, and entering necessary values. As you select a command, you can follow it with the optional specifiers, or choose QUIT OPTIONS to complete the command and move to the next line.
- **Keyboard**—Enter it from the keyboard. If the entered line is invalid, you will get the error message.
- **From File**—Read in a file containing the CL command lines.

This appendix describes using the syntax menus.

Using the Syntax Menus

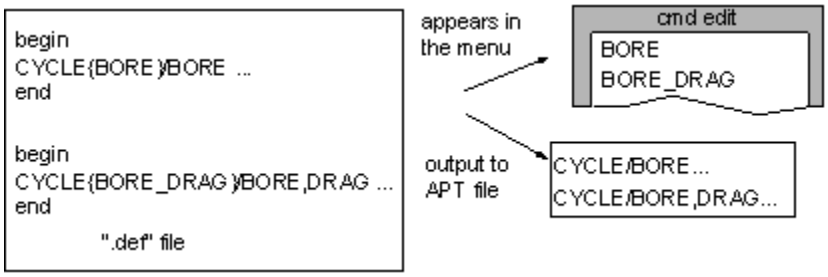
The keyword menus are built according to the APT syntax. The **Command Wizard** window of the **SMM CL COMMAND** dialog box represents a topical index. Selecting an option from this menu brings up another menu containing all major keywords for selected topic. Nothing is output to the APT file yet. If you change your mind, or cannot find the desired keyword in this topic, you can use , , or to return to the main menu, then select another topic. Selecting an option starts outputting an APT command. You cannot quit until the command is completed. If you are not satisfied, delete or change the erroneous line.

Each option a major APT keyword. If there are parameters required (such as point coordinates, etc.), you will be prompted accordingly. Enter values for these parameters. These are optional, so you can specify as many as you like. If the completed command contains an error, you will get an error message.

Creating Definition Files

The rules for creating a menu definition file are:

- Each entry (except the first) must contain a valid APT syntax string, starting with the word begin and terminated with the word end.
- The first entry, also contained between begin and end, must be the "topic name", which will appear in the topic index **cmd edit** menu.
- The syntax string must start with a major keyword (such as `CYCLE`, `GOTO`, etc.). If parameters (minor keywords or user input) are required, they must follow after the slash (the slash may be replaced by a blank space).
  Example: `CYCLE/BORE`
- If the parameter is a value that should be input at run time, it must be preceded with a pound sign (`#`).
  Example: `CLDIST/#CLDIST`
- Normally, the major keyword will appear as the second-level **cmd edit** menu option. However, you can specify a different word for the menu option, by inserting it in curly brackets `{}` between the major keyword and the slash. For example:



- All optional parameters should be included in square brackets [ ]. If an optional parameter has attributes, they must be included in the same brackets. For example: `CYCLE/BORE[,RAPTO,#RAPTO_r]`

- If some of the options are mutually exclusive, they must all be included in the same brackets, and separated by pipe "|". For example:
  `CYCLE/BORE[,IPM|,MMPM|,IPR|,MMPR,#FEDRAT_f]`
- To specify multiple input, use "...". Any value followed by "..." will be prompted for repeatedly, until you enter <CR> without supplying a value. For example:
  `CYCLE/AUTO,DEEPBK,#FEDTO_z,#FEDRAT_f[,INCR,#INCR_d...]`
- To change the order of precedence when parsing the parameters, use parentheses. The default order of precedence is from left to right. For example:
  `CYCLE/BORE[,IPM|,MMPM|,IPR|,(MMPR,#FEDRAT_f)]`
  results in specifying the FEDRAT_f value only for MMPR, whereas without the parentheses each of the mutually exclusive options will have the FEDRAT_f attribute.

Syntax Reference Table

The following is a quick syntax reference table. See the set of rules above for detailed usage explanations and examples.

Symbol	**Usage**
/	Separates the major command word from the parameters (minor keywords or user input). May be replaced by a blank space.
,	Separates the parameters.
[]	Encloses optional parameters.
|	Separates mutually exclusive parameters.
()	Changes the order of precedence when parsing the parameters.
{}	Used to specify a menu option name other than the major keyword.
# label	Precedes a parameter value that must be input at run time. Label is used in the prompt.
<> label	Precedes a string of text that must be input at run time. Label is used in the prompt.
...	Designates multiple input. The previous label will be prompted for until a blank line is entered.

Menu Syntax

When creating your own definition files, keep in mind that the words appearing in the **cmd edit** menus must comply with the Pro/ENGINEER menu syntax. Each item must be unique in the menu. For example, if you have two different commands with the

same major keyword, specify an alternate name in the curly brackets {} for at least one of them.

The maximum number of characters in a menu item allowed by the system is 19. You may want to use only 12 characters to fit in the standard Pro/ENGINEER menu, or modify your menu width.

Note: Because of proportional text width on SUN workstations, the number of uppercase characters that will fit in the standard Pro/ENGINEER menu may be less than 12.

Compiling Definition Files

When all the definition files are created, they must be compiled to transfer them to the `.syn` files. The command to compile a definition file is:

```
pro_mcs_comp < filename.def
```

options:

`-p`

print the results. The menu tree of the compiled file will be output on the screen. If there are any errors, the compiler will inform you, otherwise a message: `Finish (reading) NO ERRORS` will be output to the screen before the menu tree creation.

`-i`

specify the filename, `pro_mcs_comp -i filename.def`

The compiler creates the corresponding `.syn` file in your current directory.

Note: The `pro_mcs_comp` command is located in the obj subdirectory of the machine-specific directory under the Pro/ENGINEER loadpoint, for example `proe/sun4/obj/pro_mcs_comp`.

Index File

The index file `cmdsyn.ndx` must contain the names (including the extension) of all the `.syn` files created. This file basically defines the contents of the first, index, **cmd edit** menu. The first entries of the files included in the **cmdsyn.ndx** file will appear as options in the index **cmd edit** menu, and as you select an option, the appropriate second-level **cmd edit** menu will be displayed.

To Customize Your Syntax Menus

1. Create the menu definition files. They must have an extension `.def`.
2. Compile the menu definition files to transfer them to the `.syn` files.
3. Include all the `.syn` file names into the index file `cmdsyn.ndx`.

To Change Feed Rate at Some Point Along the Tool Path

1. Select an NC Sequence. The **SMT NC SEQUENCE** dialog box opens.

2. Click **CL Command** and click **Define**. The **SMM CL COMMAND** dialog box opens.
3. Click a command and use  to place the CL command at the select point of the tool path, or use  to place the CL command at the beginning of the tool path, or use  to place the CL command at the end of the tool path.
4. Select **Movement** in **Command Wizard** window.
5. Select **FEDRAT/**.
6. Specify the required value for feed ratio.
7. Click **Done**.

System-Supplied Syntax Menus

Sheet Metal Manufacturing provides two sets of system-supplied menu structures: ISO and AI. ISO is the default. If you want to use the AI menu structure, use the configuration option:

`pro_mfg_cmdsyn_dir proe/mfg_cmdsyn_ai`

where `proe` is the Pro/ENGINEER loadpoint on your system.

Defining Your Own Syntax Menus

You can replace the system-supplied syntax menu structure (used to add lines in editor) with your own.

The default directory for syntax menu files can be specified using the `config.pro` option `pro_mfg_cmdsyn_dir`. Sheet Metal Manufacturing will look for the syntax files in the following order:

1. in the current directory,
2. in the default directory, if it is set, and
3. in the system directory.

The system directory for the `.syn` files is `proe/mfg_cmdsyn`, where `proe` is the Pro/ENGINEER loadpoint.

Naming Convensions

Naming Conventions

The following naming conventions are used in this optional module:

File Extension	File Type
.asm	Assembly file

.aux	Auxiliary parameter data file
.cel	Machine parameter data file
.dat	Data files created for editing, such as relations data
.drw	Drawing file
.edm	Contouring parameter data file
.gph	User-defined feature file (including workcells)
.inf	Information data file
.memb	Assembly member information file
.mfg	Manufacturing process file
.mtn	Tool motion parameter file
.ncd	CL syntax alias file
.nck	NC Check image file
.ncl	CL data file (including pre- and post-machining files)
.plt	Plot file
.ppl	Route sheet data file
.ppr	PPRINT settings table file
.prt	Part file
.ptd	Part family table file
.sec	Section file
.shd	Shade display file
.sit	Site parameter data file
.smt	Parameter data file for all Punch Press NC sequences
.tph	Tool path storage file
.tpm	Tool parameter file

Index