NC Builder™ Turning User Guide and Menu Reference

CADDS® 5 Revision 8.0

DOC40179-001

Parametric Technology Corporation

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494-2714
8 January 2001

Table of Contents

Preface

NC Builder Turning Interface

Machine Tool Definitions

Discrete Tool Motion

Profiling Tasks

Turning and Facing Tasks

Grooving Tasks

Drilling Tasks

Threading Tasks

Tapping Tasks

Commands

Preface

NC Builder Turning User Guide and Menu Reference provides instructions on producing a new turning product using the NCBuilder Graphical User Interface.

Related Documents

The following documents may be helpful as you use *NC Builder Turning User Guide and Menu Reference*:

- *NC Builder User Guide and Menu Reference*
- *CVNC-T2 Command Reference*
- *CVNC-T2 User Guide*

Book Conventions

The following table illustrates and explains conventions used in writing about CADDS applications.

Convention	Example	Explanation
Menu selections and options	List Section option, Specify Layer field	Indicates a selection you must make from a menu or property sheet or a text field that you must fill in.
User-selected graphic location	X, d_1 or P1	Marks a location or entity selection in graphic examples.
User input in CADDS text fields and on any command line	`cvaec.hd.data.param` `tar -xvf /dev/rst0`	Enter the text in a CADDS text field or on any command line.
System output	`Binary transfer complete.`	Indicates system responses in the CADDS text window or on any command line.
Variable in user input	`tar -cvf /dev/rst0` filename	Replace the variable with an appropriate substitute; for example, replace filename with an actual file name.
Variable in text	tagname	Indicates a variable that requires an appropriate substitute when used in a real operation; for example, replace tagname with an actual tag name.
CADDS commands and modifiers	INSERT LINE TANTO	Shows CADDS commands and modifiers as they appear in the command line interface.
Text string	`"SRFGROUPA"` or `'SRFGROUPA'`	Shows text strings. You must enclose text string with single or double quotation marks.
Integer	n	Supply an integer for the *n*.
Real number	x	Supply a real number for the *x*.
#	`# mkdir /cdrom`	Indicates the root (superuser) prompt on command lines.
%	`% rlogin` remote_system_name `-l root`	Indicates the C shell prompt on command lines.
$	`$ rlogin` remote_system_name `-l root`	Indicates the Bourne shell prompt on command lines.

Window Managers and the User Interface

According to the window manager that you use, the look and feel of the user interface in CADDS can change. Refer to the following table:

Look and Feel of User Interface Elements

User Interface Element	Common Desktop Environment (CDE) on Solaris, HP, DEC, and IBM	Window Manager Other Than CDE on Solaris, HP, DEC, IBM, SGI, and NT
Option button	ON — Round, filled in the center OFF — Round, empty	ON — Diamond, filled OFF — Diamond, empty
Toggle key	ON — Square with a check mark OFF — Square, empty	ON — Square, filled OFF — Square, empty

Online User Documentation

Online documentation for each book is provided in HTML if the documentation CD-ROM is installed. You can view the online documentation in the following ways:

- From an HTML browser
- From the Information Access button on the CADDS desktop or the Local Data Manager (LDM)

Please note: The LDM is valid only for standalone CADDS.

You can also view the online documentation directly from the CD-ROM without installing it.

From an HTML Browser:

1. Navigate to the directory where the documents are installed. For example,

   `/usr/apl/cadds/data/html/htmldoc/` (UNIX)

   Drive:`\usr\apl\cadds\data\html\htmldoc\` (Windows NT)
2. Click `mainmenu.html`. A list of available CADDS documentation appears.
3. Click the book title you want to view.

From the Information Access Button on the CADDS Desktop or LDM:

1. Start CADDS.
2. Choose Information Access, the i button, in the top-left corner of the CADDS desktop or the LDM.
3. Choose DOCUMENTATION. A list of available CADDS documentation appears.
4. Click the book title you want to view.

From the Documentation CD-ROM:

1. Mount the documentation CD-ROM.
2. Point your browser to:

   CDROM_mount_point/htmldoc/mainmenu.html (UNIX)

   CDROM_Drive:\htmldoc\mainmenu.html (Windows NT)

Online Command Help

You can view the online command help directly from the CADDS desktop in the following ways:

- From the Information Access button on the CADDS desktop or the LDM
- From the command line

From the Information Access Button on the CADDS Desktop or LDM:

1. Start CADDS.
2. Choose Information Access, the i button, in the top-left corner of the CADDS desktop or the LDM.
3. Choose COMMAND HELP. The Command Help property sheet opens displaying a list of verb-noun combinations of commands.

From the Command Line: Type the exclamation mark (!) to display online documentation before typing the verb-noun combination as follows:

```
#01#!INSERT LINE
```

Printing Documentation

A PDF (Portable Document Format) file is included on the CD-ROM for each online book. See the first page of each online book for the document number referenced in the PDF file name. Check with your system administrator if you need more information.

You must have Acrobat Reader installed to view and print PDF files.

The default documentation directories are:

- /usr/apl/cadds/data/html/pdf/doc_number.pdf (UNIX)
- CDROM_Drive:\usr\apl\cadds\data\html\pdf\doc_number.pdf (Windows NT)

Resources and Services

For resources and services to help you with PTC (Parametric Technology Corporation) software products, see the *PTC Customer Service Guide*. It includes instructions for using the World Wide Web or fax transmissions for customer support.

Documentation Comments

PTC welcomes your suggestions and comments. You can send feedback in the following ways:

- Send comments electronically to `doc-webhelp@ptc.com.`
- Fill out and mail the PTC Documentation Survey located in the *PTC Customer Service Guide.*

Chapter 1

NC Builder Turning Interface

This chapter details the interface features of a new turning product using the NC Builder Graphical User Interface. The following sections describe these interface features.

- Accessing the NC Builder Turning Option
- Using the Setup Sheet
- Using the Job Sheet
- Using the Operation Sheet
- Using the Task Selector

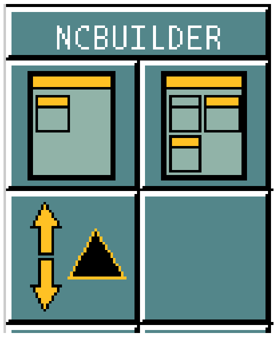
Accessing the NC Builder Turning Option

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.

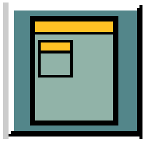
OR

You can access the turning functionality by entering the CADDS command PROGRAM NCTURN. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears. This task set contains the three icons to drive NC Builder.

Figure 1-1 NCBuilder task set

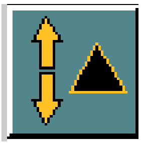


NC Builder Icons



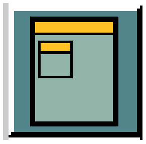
Launches the Setup sheet to create a Setup block.

Launches the Job sheet to create a Job block.



Launches the NCBuilder Positioner sheet to position between blocks, and from which to launch the Setup and Job sheets for editing blocks.

Using the Setup Sheet



Choose the Setup Block option from the NCBuilder task set to launch the Setup sheet.

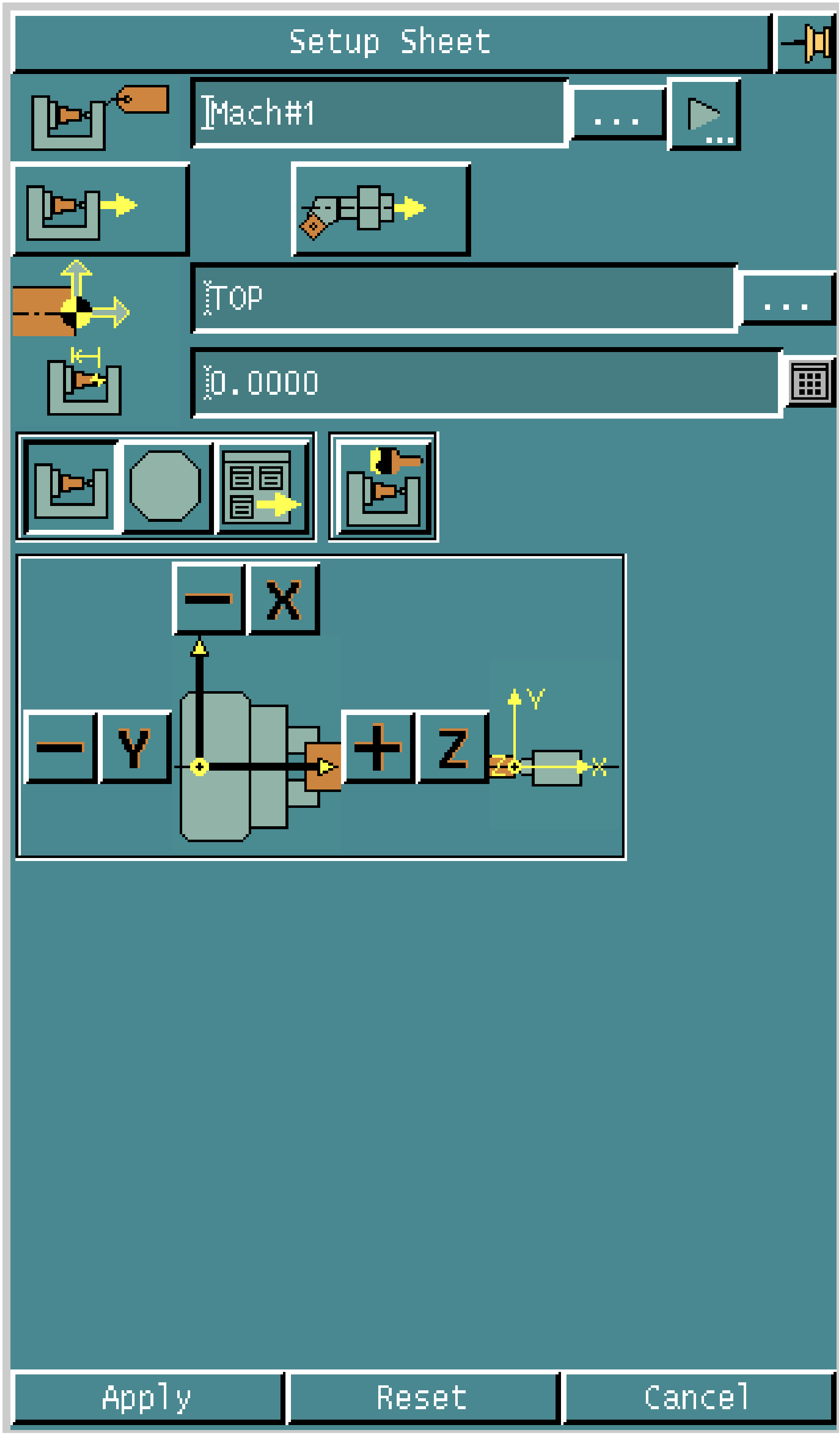
Use the Setup sheet to define the machine setup including the turrets, stations, and tooling definitions. You can define each distinctive area of the machine tool as well as general start up parameters including the datum cpl, programming cpl, and output requirements.

You can also use the Setup sheet to create tool assemblies from pre-defined lists of tooling inserts, holders, and boxes. Once defined and assigned to stations on the machine tool turret or turrets, you can save these tool assemblies to a text file for future use, or load such a previously defined file to define the tooling set to be loaded.

Similarly, you can save or reload complete machine setup text files, which is the designed and recommended technique for using the product. Thus you can define a file for each different machine tool and use these files when you want to program a job. In this way, a complex machine tool may be utilized and restored into the interface with just a few clicks of the mouse.

Please note: The various options in the Setup sheet are described in detail in Chapter 2, “Machine Tool Definitions”.

Figure 1-2 Setup Sheet

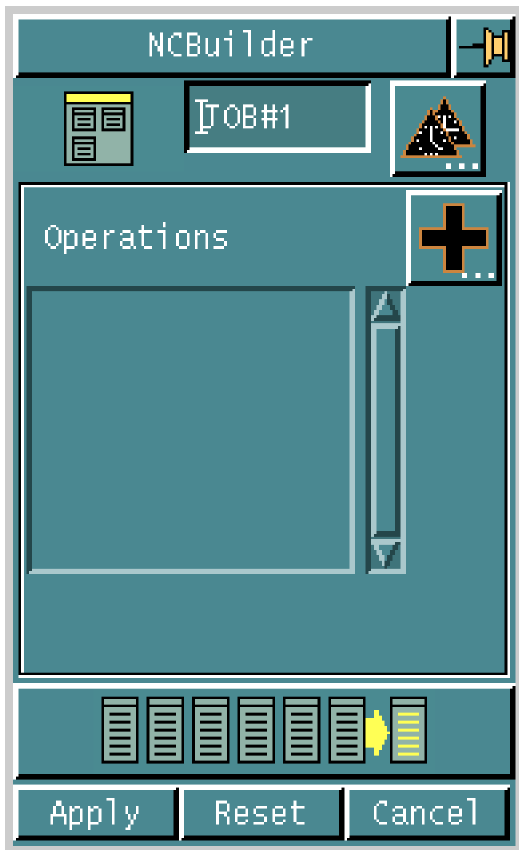


Using the Job Sheet

Choose the Job Block option from the NCBuilder task set to launch the NCBuilder sheet.

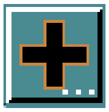
Use the NCBuilder sheet to create a Job block. This Job sheet is the same as that for the NC Builder interface for milling and has the same set of features.

Figure 1-3 Job Sheet



Please note: See the *NC Builder User Guide and Menu Reference* for further information.

Using the Operation Sheet



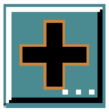
Choose the Add Operation icon from the NCBuilder sheet to launch the Operation sheet.

Use the Operation Sheet to add a new operation. This Operation Sheet is similar to that for the NC Builder interface for milling and has a similar set of features.

Figure 1-4 Operation Sheet

Please note: See the *NC Builder User Guide and Menu Reference* for further information.

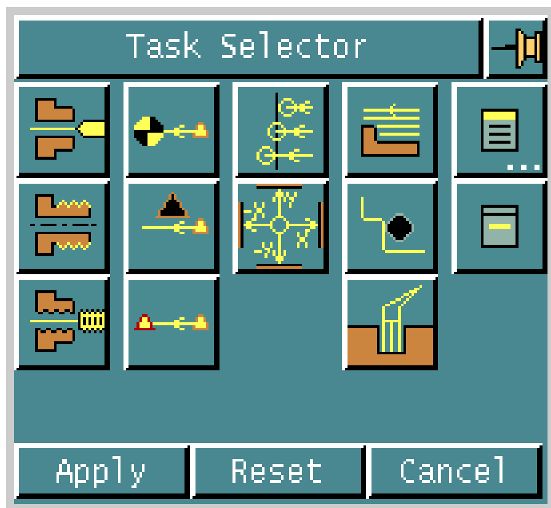
Using the Task Selector



Choose the Add icon on the Operation Sheet to launch the Task Selector.

Use the Task Selector to add a new task or method. This Task Selector is similar to the Task Selector for the NC Builder milling interface but contains a different set of tasks. These include tasks for turning, facing, profiling, grooving, discrete tool motion, threading, drilling, and tapping.

Figure 1-5 Task Selector



Please note: The various options in the Task Selector are described in detail in Chapter 3, “Discrete Tool Motion”, Chapter 4, “Profiling Tasks”, Chapter 5, “Turning and Facing Tasks”, Chapter 6, “Grooving Tasks”, Chapter 7, “Drilling Tasks”, Chapter 8, “Threading Tasks”, and Chapter 9, “Tapping Tasks”.

Chapter 2

Machine Tool Definitions

This chapter gives details of support for machine tools with front and/or rear turrets for forward or backward turning. These turrets may have multiple stations on either the edge or face of the turret, for internal, external, or driven tooling. This chapter also describes the sequence, and build up of reference points and offset values for each of the turrets and its stations.

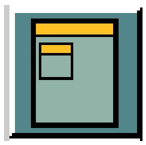
Information is also provided on how to assign tool assemblies to stations, along with discussions on inserts, holders, and tool boxes.

The following sections describe these processes in detail.

- Accessing the Setup Sheet
- Using Various Options on the Setup Sheet
- Specifying Turret, Station, and Tool Assembly Parameters
- Individual Turret Configuration
- Individual Station Configuration for a Turret
- Assignment of Tool Assemblies to Stations
- Specifying Tool Insert Parameters
- Specifying Tool Holder Parameters
- Specifying Tool Box Parameters
- Tool Gage Lengths
- Specifying Output Generation Parameters

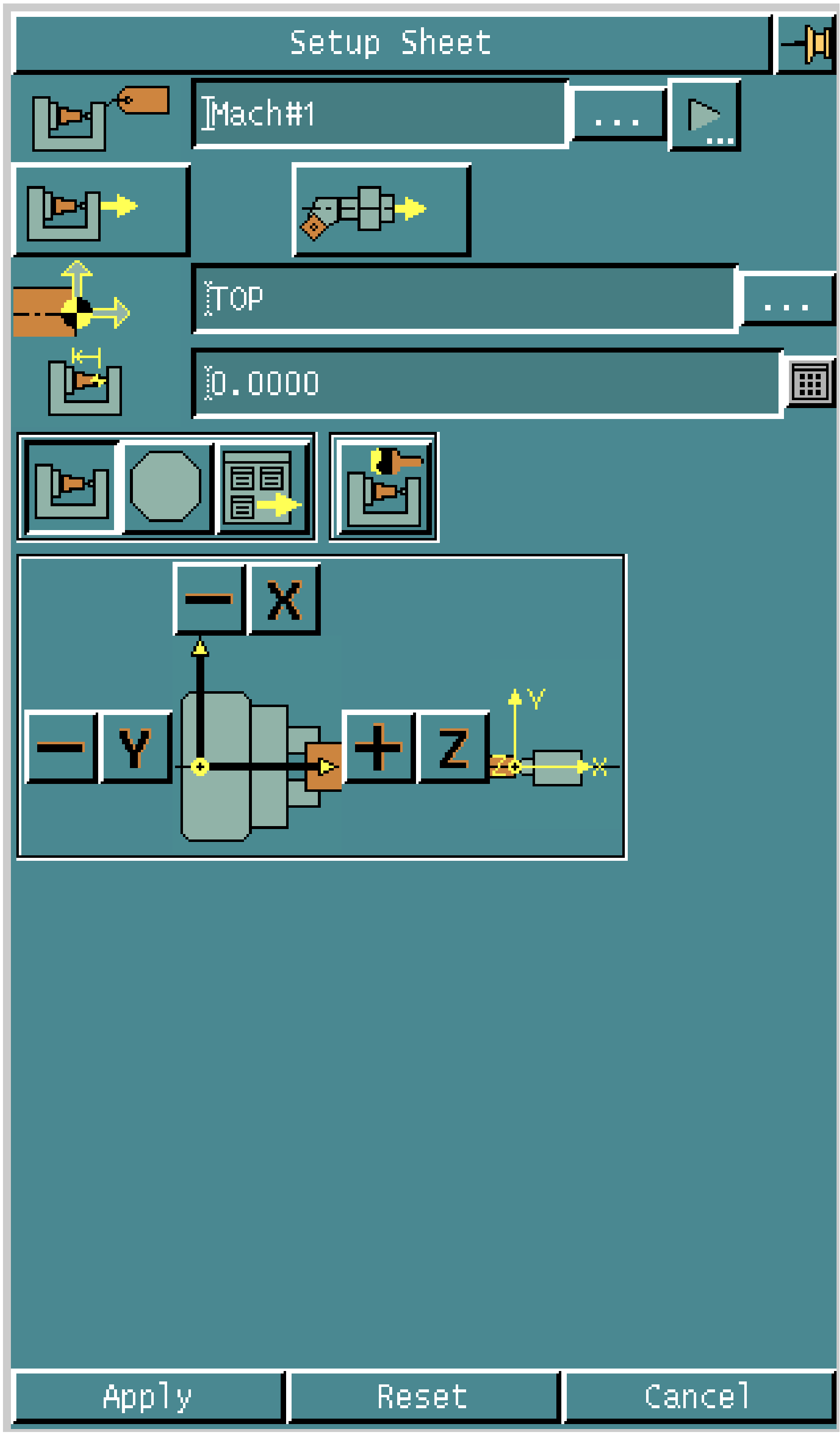
Accessing the Setup Sheet

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.



4. Choose the Setup Block option from the NCBuilder task set. The Setup sheet appears.

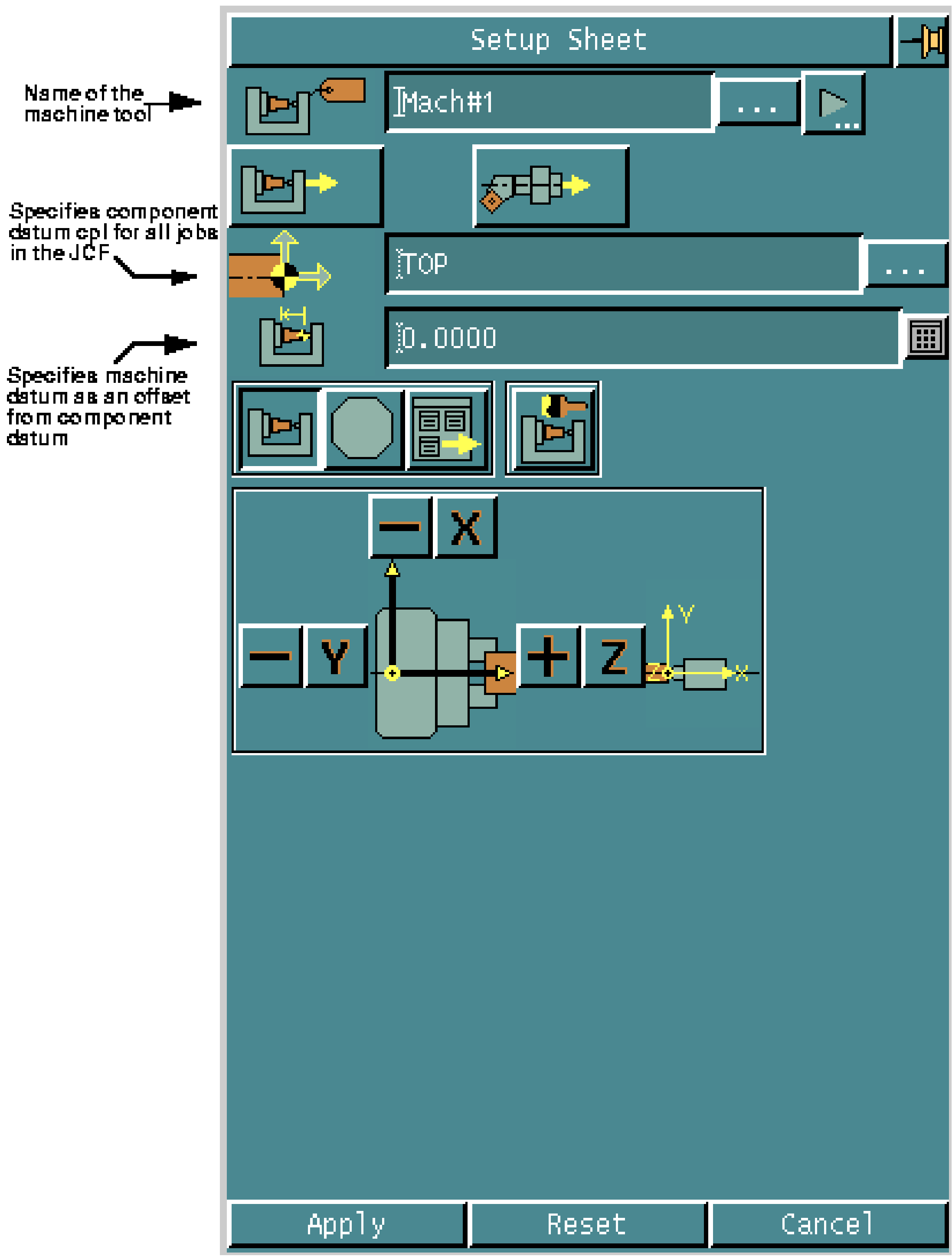
Figure 2-1 Setup Sheet



Using Various Options on the Setup Sheet

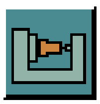
The following sections describe various options on the Setup Sheet.

Figure 2-2 Setup Sheet

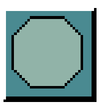


1. Specify the name of the machine tool in the Machine Name field.
2. Specify the name of the file into which to write the tool assemblies for the current machine, in case the tool assemblies are to be written to a file.
3. Specify the component datum CPL for all jobs in the JCF in the Datum field. See “Specifying the Component Datum” on page 2-5 for details.
4. Specify the machine datum as an offset from the component datum in the Component Offset field. See “Specifying the Machine Datum” on page 2-6 for details.

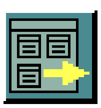
Various other options on the Setup sheet are:



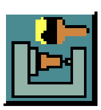
Click this option to specify machine tool parameters.



Click this option to specify turret, station, and tool assembly definition parameters. See “Specifying Turret, Station, and Tool Assembly Parameters” on page 2-8 for details.



Click this option to specify output generation parameters. See “Specifying Output Generation Parameters” on page 2-39 for details.

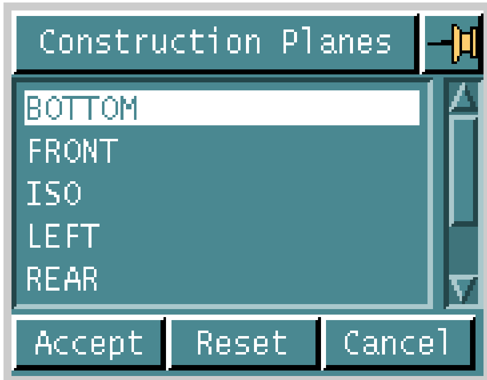


Click this option to specify that the machine tool graphics updating is turned on.

Specifying the Component Datum

In the Setup Sheet, you can specify a construction plane (CPL) which represents the component datum, or select a CPL from a runtime list of construction planes defined for the part, as shown in Figure 2-2. The CPL you specify becomes the active CPL.

Figure 2-3 Runtime list of Construction Planes



The CPL also specifies the origin for output generation, although the axes for the output generation may be translated to those of the machine datum, as described in the following section.

Specifying the Machine Datum

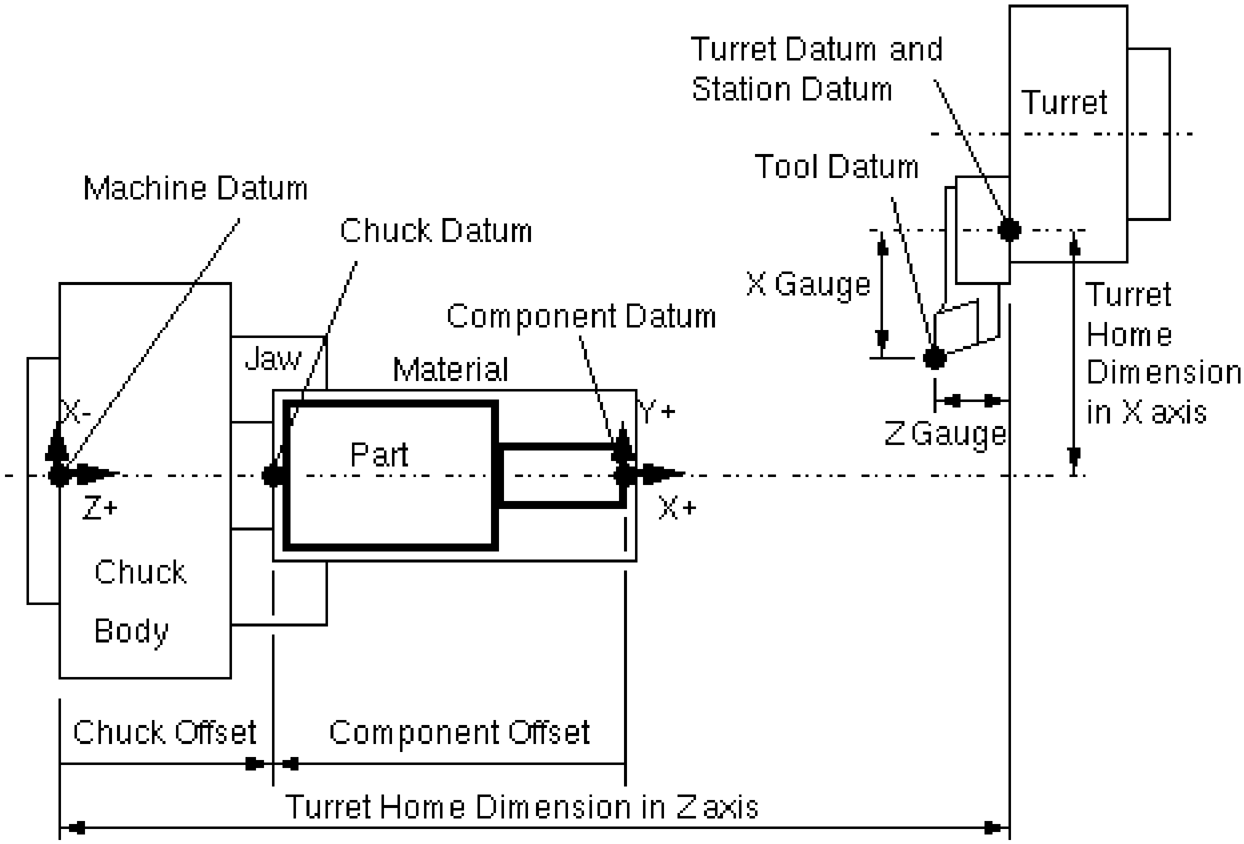
Machine datum is the point from which all other machine dimensions are taken, and is a fixed point often located at the nose of the spindle on the machine tool.

In the Setup Sheet, you can specify a value which indicates the axial offset of the machine datum from the component datum. This component offset is automatically reduced by the value of the axial offset of the chuck datum from the machine datum, leaving the axial offset of the chuck datum from the component datum.

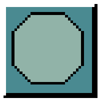
You can also specify how the machine datum axes are mapped relative to the XY axes of the component datum, by choosing the icon which displays a page of machine tool parameters, as shown in Figure 2-1.

For example, in the following figure, Component X is mapped to Machine Z, and Component Y is mapped to Machine X-.

Figure 2-4 Machine Datum

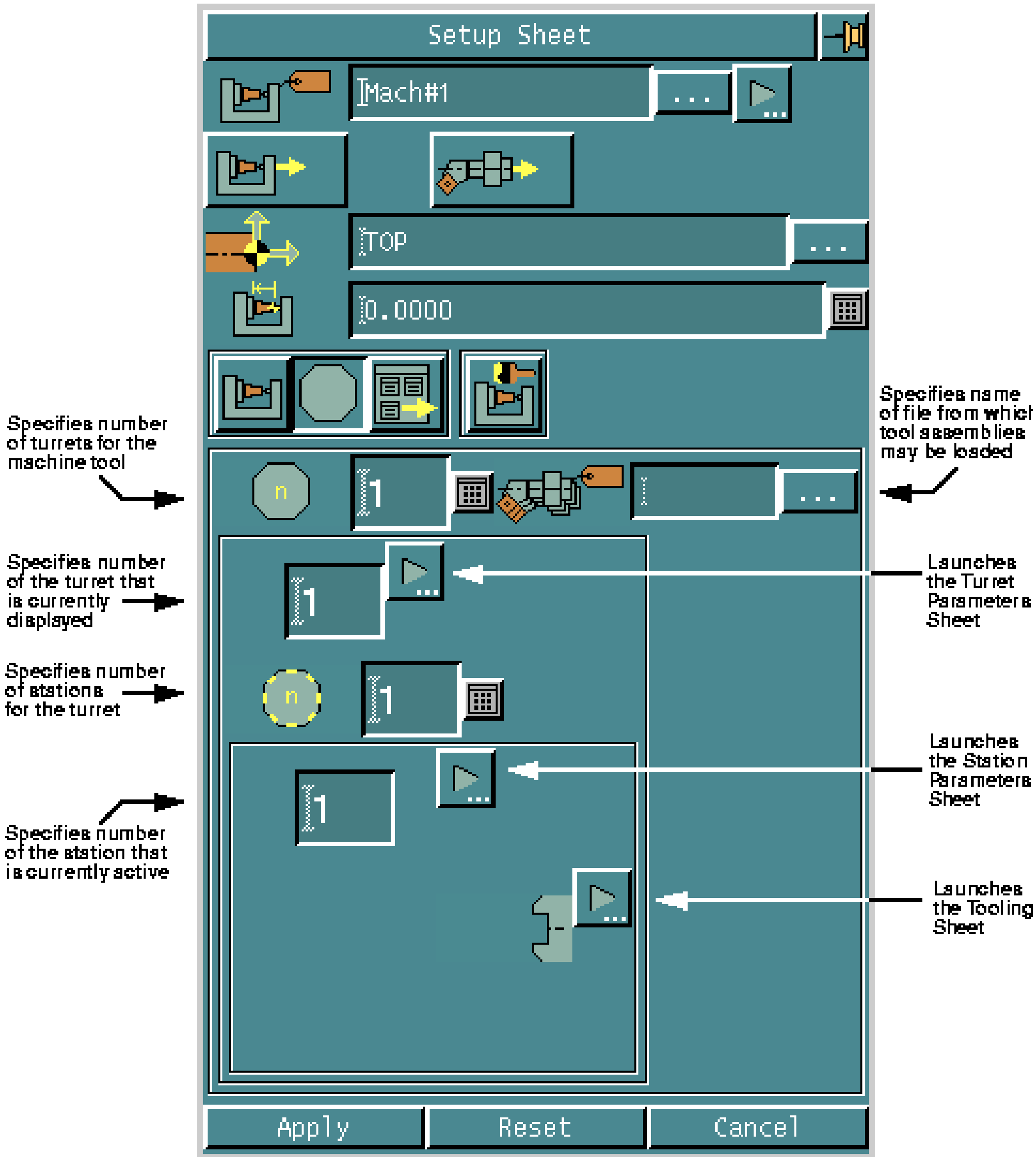


Specifying Turret, Station, and Tool Assembly Parameters



You can use the Setup Sheet to specify turret, station, and tool assembly parameters, as shown in the following figure.

Figure 2-5 Setup Sheet to specify Turret, Station, and Tool Assembly parameters



Individual Turret Configuration

Each turret that you define can be configured with the following options:

- Front or rear
- Forward or backward
- Rotation around machine X-, Y-, or Z-axis
- Home position of turret datum from machine datum
- FROM point
- Turret rotation centre from turret datum
- Angle of rotary axis
- Number of stations

Each of these options is described in detail in the following sections.

Figure 2-6 Turret Parameters Sheet

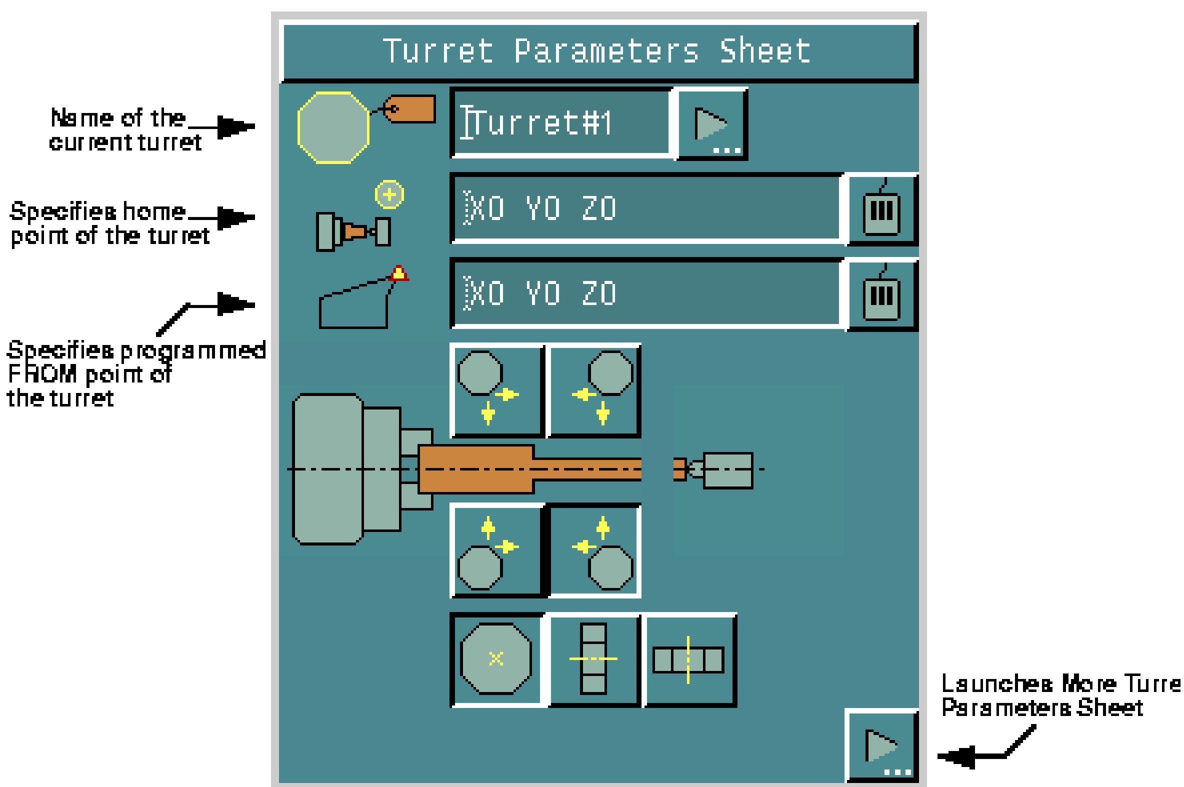
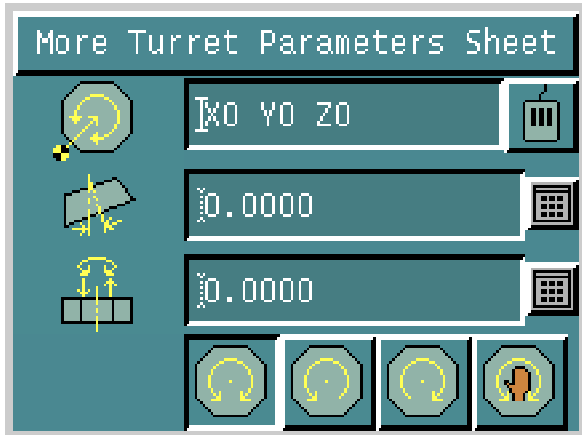


Figure 2-7 More Turret Parameters Sheet



Front, Rear, Forward, or Backward Turrets

Use the Turret Parameters Sheet to specify whether the turret you defined is a:

- Rear backward turret
- Rear forward turret
- Front backward turret
- Front forward turret

Front indicates that the turret is usually used for machining with the turret located below the center line with respect to the machine datum.

Rear indicates that the turret is usually used for machining with the turret above the center line with respect to the machine datum.

Both these cases are irrespective of the home point location which may be in the opposite quadrant. For example, a front turret may have a home point in the rear quadrant and vice versa.

Forward indicates that the turret stations face towards the chuck, while backward indicates that the turret stations face away from the chuck.

The information you provide is used to determine the position of the active station, and on which edges or faces of the turret the stations will be placed, as shown in the following 3 figures.

Rotation Around Machine X-, Y-, or Z-axis

Since the component datum CPL may not be the same as the machine datum, constant reference to the X-, Y-, and Z-axis can be confusing. Therefore, the following terms have been used when defining the turret's rotating axis:

Axial rotation: Rotation around the X-axis with respect to component datum.

Radial rotation: Rotation around the Y-axis with respect to component datum.

Vertical rotation: Rotation around the Z-axis with respect to component datum.

Please note: You can define each turret to rotate around any of the above axes, for the purpose of aligning the stations, even though the turret may not have rotational capability.

Figure 2-8 Radial Rotation showing Tool Assembly in Active Station

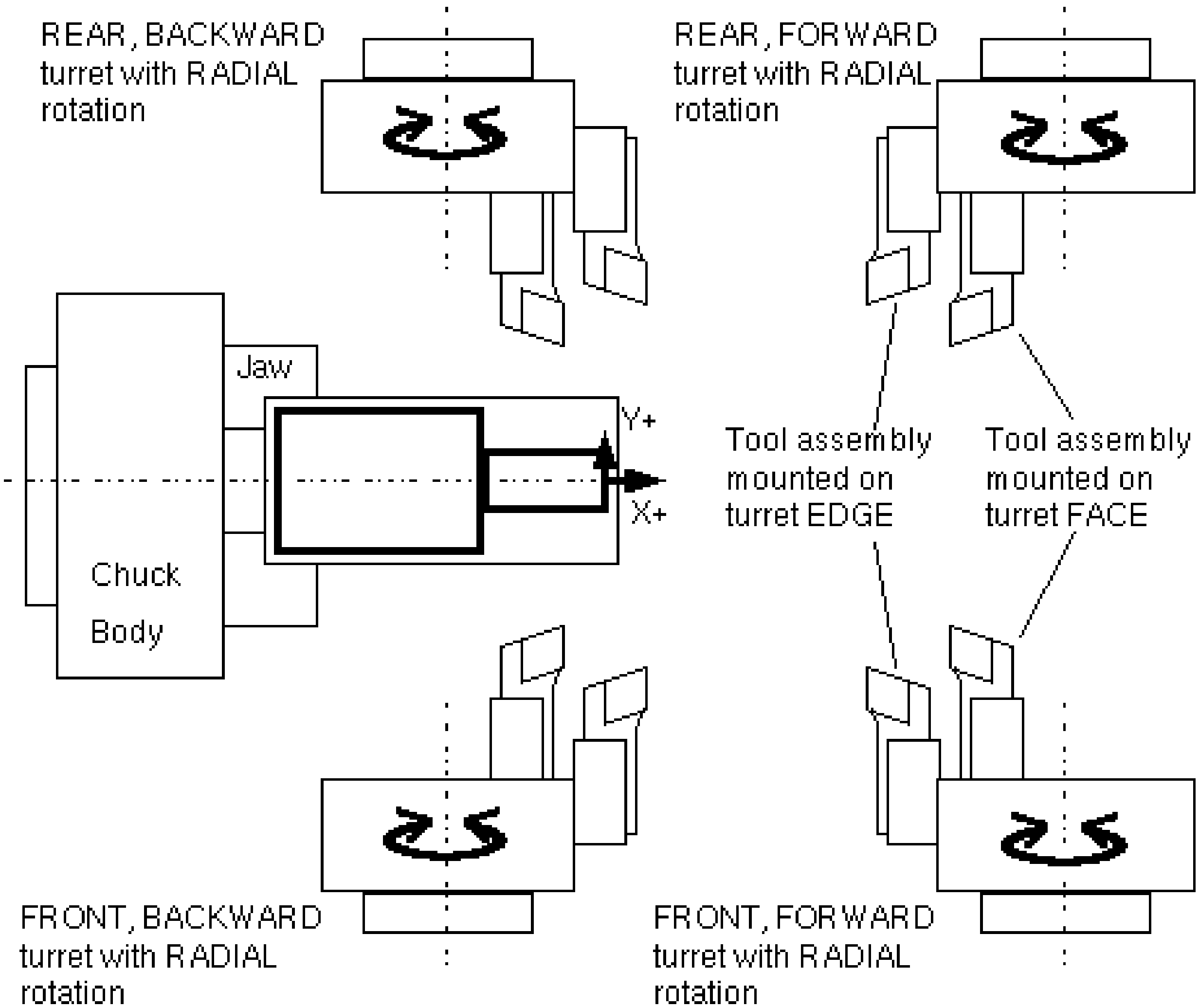


Figure 2-9 Vertical Rotation showing Tool Assembly in Active Station

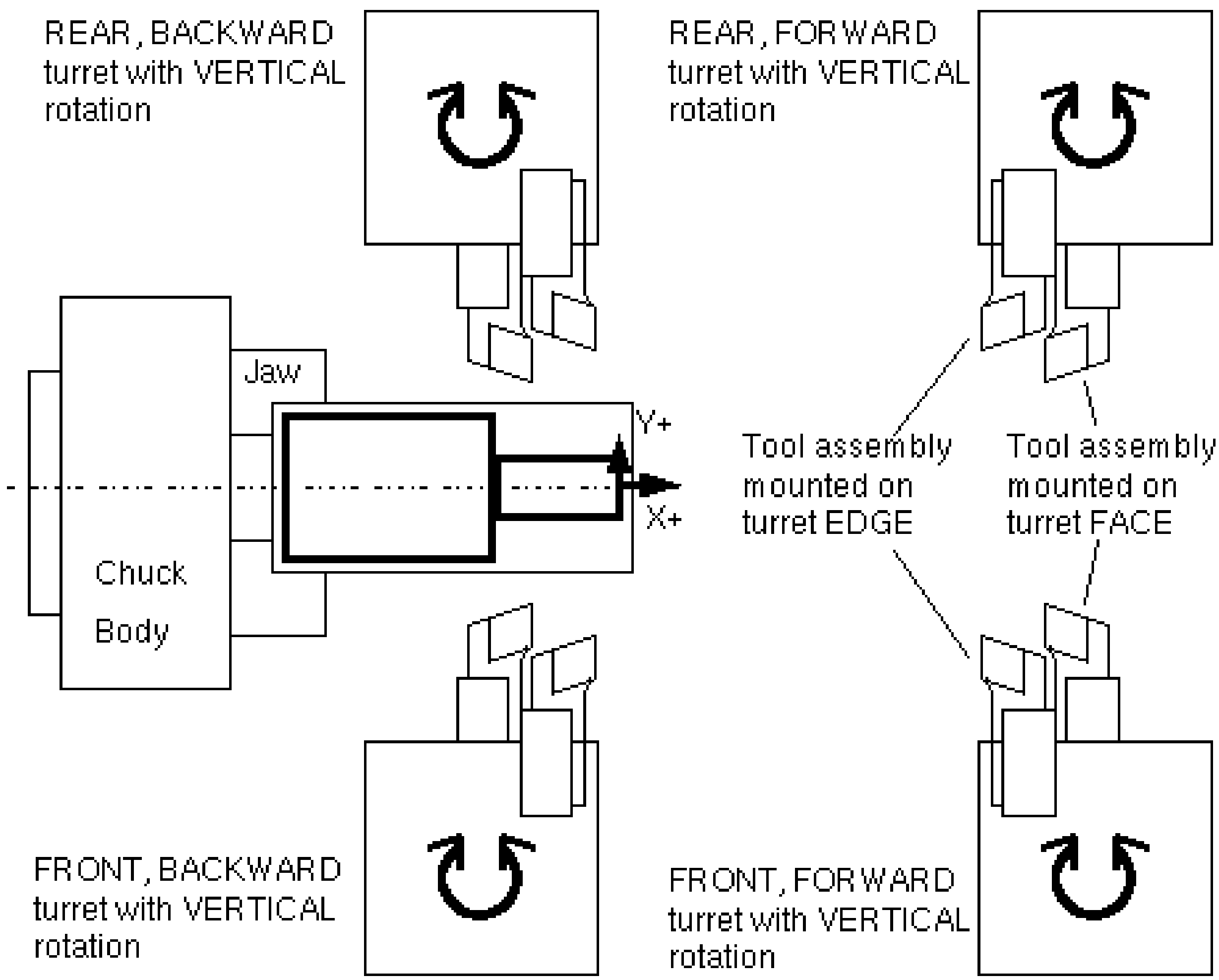
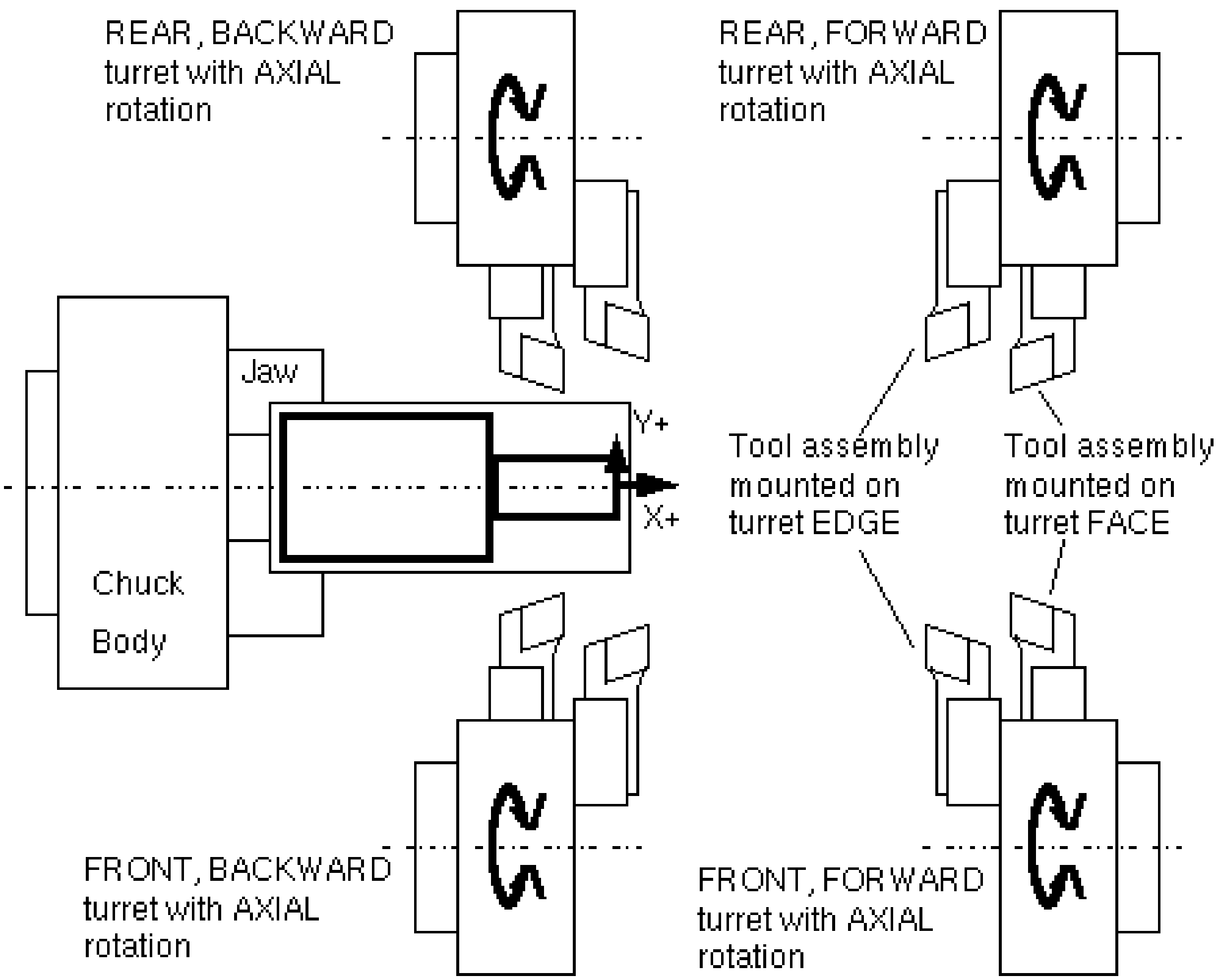


Figure 2-10 Axial Rotation showing Tool Assembly in Active Station



Home Position of Turret Datum from Machine Datum

You can specify a home position for each turret, in the Turret Parameters Sheet, as shown in Figure 2-6. This position is interpreted as the home position for the drive of the machine tool on which the turret is mounted, and represents the turret datum. The home position may, therefore, be different to the programmed FROM point. It is defined from the component datum CPL and is interpreted back to the machine datum given the component offset. See Figure 2-10.

FROM Point

You can specify a unique programmed FROM point for each turret, in the Turret Parameters Sheet, as shown in Figure 2-6. This point is not necessarily the same as the home position described above.

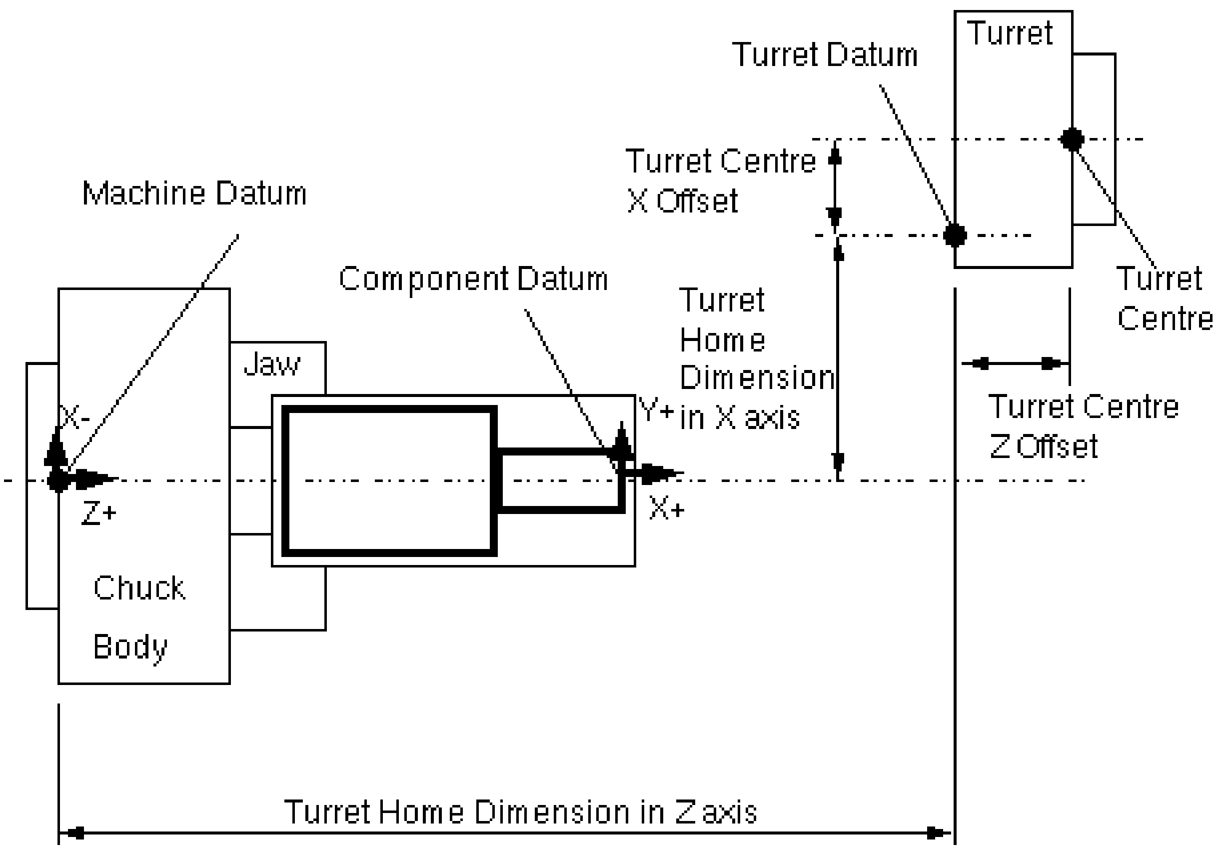
Turret Rotation Centre from Turret Datum

As described earlier, the turret datum is positioned by the turret's home point. However, the turret datum may not be on the center of rotation of the turret, but may be offset. You can specify this offset of the rotational centre of the turret from turret datum, in the More Turret Parameters Sheet, shown in Figure 2-6.

If this offset is present, the turret datum remains stationary when the turret is incrementally moved about its rotational axis. The turret datum does not move when the turret is incrementally moved around to change the active station.

The following figure shows the turret centre offsets for an axial rotating turret. The same offsets also apply to vertical and radial turrets.

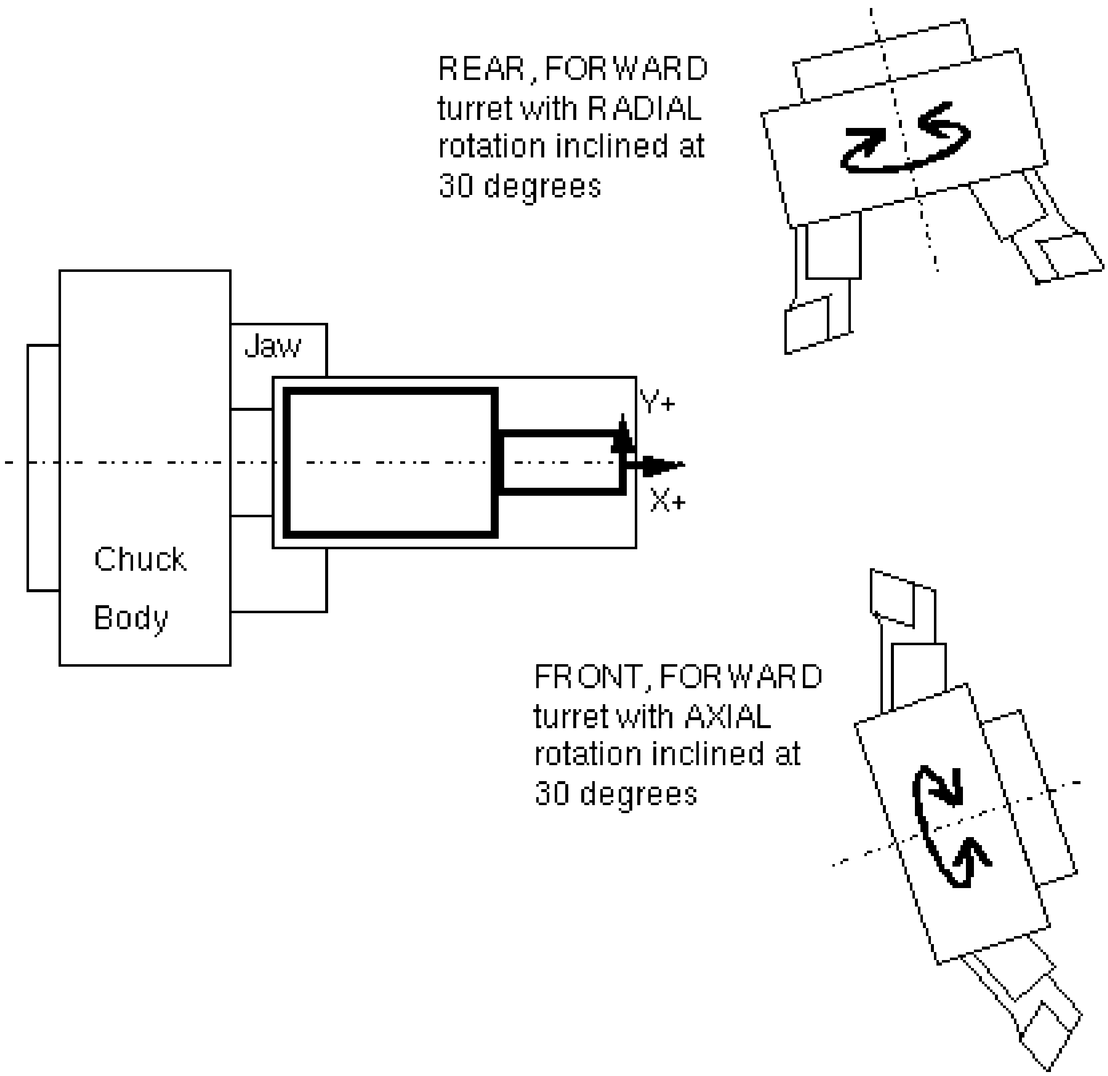
Figure 2-11 Turret Center Position



Angle of Rotary Axis

In the More Turret Parameters Sheet, you can also specify an angle by which the turret's rotary axis may be inclined and is valid for radial, axial, and vertical rotating turrets. Positive angles are measured in the counter clockwise direction, as shown in the following figure.

Figure 2-12 Radial and Axial Turrets with Inclined Rotary Axis



Number of Stations

In the Turret Parameters Sheet, you can specify any number of stations for each turret that is configured. These stations are the location at which a tool assembly, consisting of the tool box, holder, and tip is installed. The definition of a station is provided in the following section.

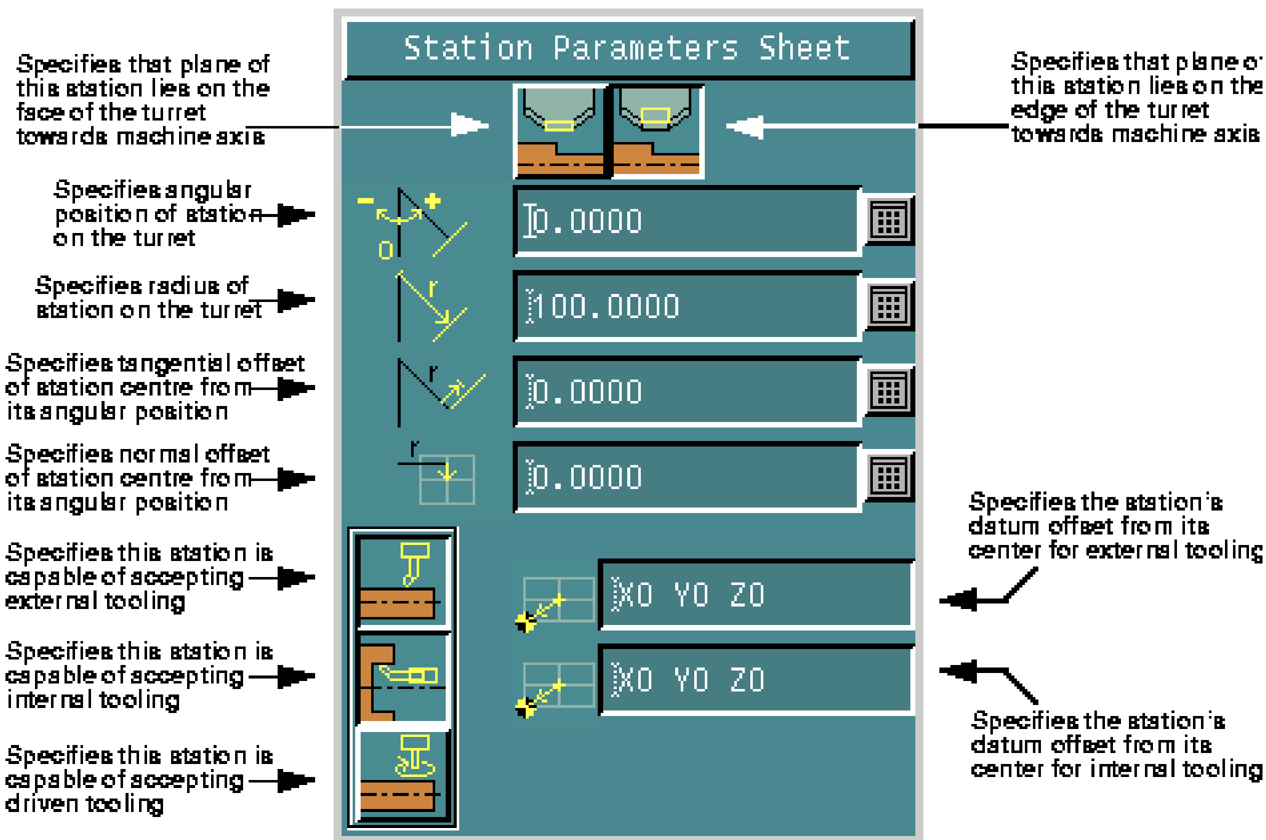
Individual Station Configuration for a Turret

You can use the Station Parameters Sheet to configure each station in the turret with the following options:

- Station number
- Station centre from turret centre
- Station alignment (edge or face)
- Station capability (external, internal, or driven tooling)
- Station datum offset

Each of these options is described in detail in the following sections.

Figure 2-13 Station Parameters Sheet



Station Number

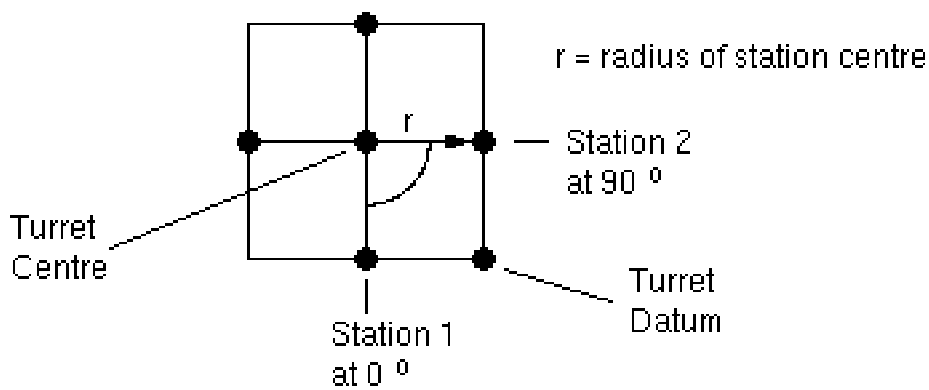
Each station is assigned a station number. You can configure a turret to have any number of stations. The stations are numbered with the first station as number 1.

Station Center from Turret Center

Each station has a centre location defined from the turret centre. You can use the Station Parameters Sheet to specify the angle, and the radius of the station centre from the turret centre.

The angle you specify must be measured around the turret in such a way that a counterclockwise angle is positive. The angles are measured from the home rotation position, which means a station defined at angle zero is active when the turret is at its zero rotation position. This definition is true for all rotational turrets, whether axial, radial, or vertical rotating turrets. See the following figure.

Figure 2-14 Station Center Definition from Turret Center



The plane in which these rotations are measured is the plane of the turret normal to its rotary axis, and on which the tools are mounted, as shown in Figure 2-7, Figure 2-8, and Figure 2-9. This plane is the face of the turret for radial turrets, and the edge of the turret for vertical and axial turrets.

Station Alignment

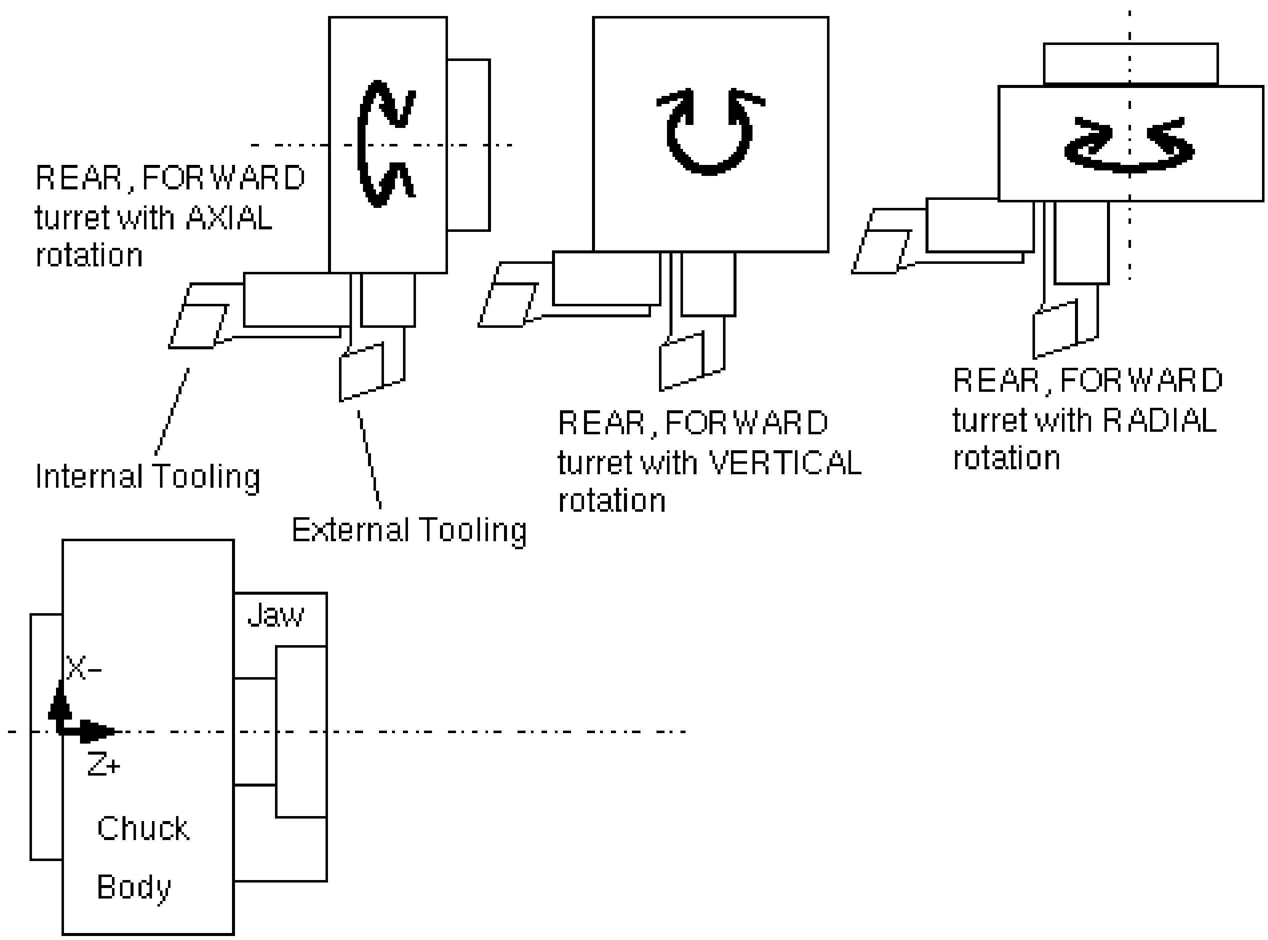
You can use the Station Parameters Sheet to configure each station position as either a face, or edge alignment. In a face alignment, the tool assembly locates on the face of the turret towards the machine z-axis, while in an edge alignment, the tool assembly locates on the edge of the turret towards the machine z-axis.

Figure 2-7, Figure 2-8, and Figure 2-9 show tool assemblies in both the edge and face alignment configurations for all turret configurations.

Station Capability

You can use the Station Parameters Sheet to configure each station to accept external, or internal tooling. You can also configure a station to accept any one, or a combination of external, and internal tooling. This is illustrated in the following figure which shows station configurations having external and internal tools, with face alignment for all rear, forward turret configurations.

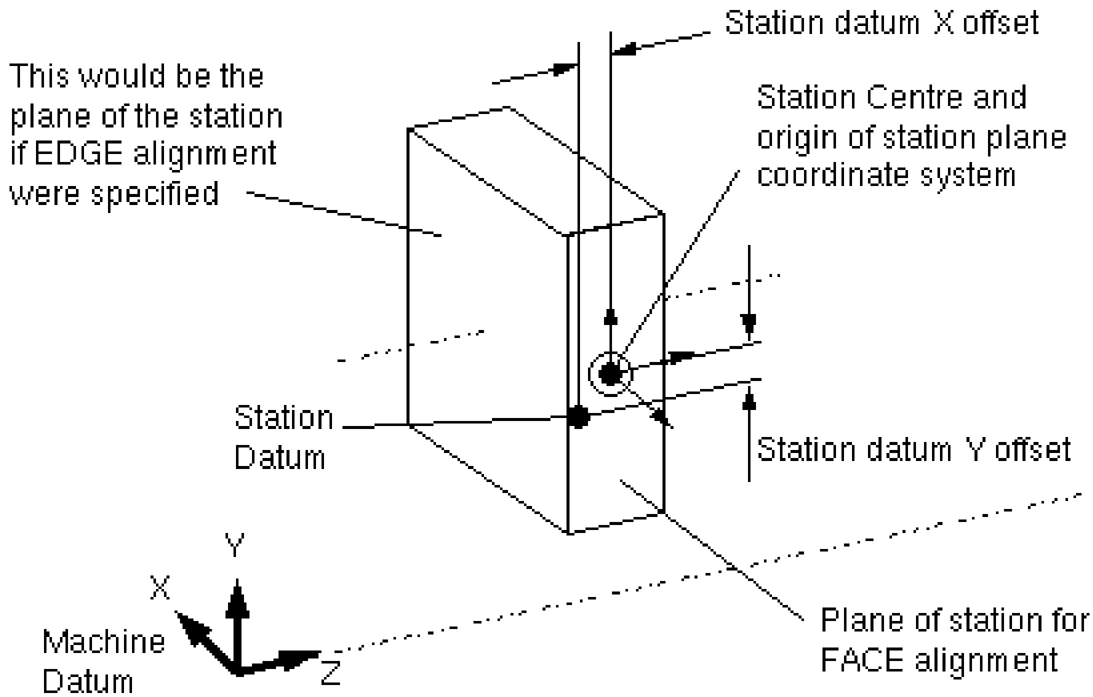
Figure 2-15 Various Station Configurations



Station Datum Offset

Using the Station Parameters Sheet, you can specify a station datum for each station, and station capability, whether external, internal, or driven. The station datum is defined from the station center relative to the plane of the station, and is the locating point for the tool assembly datum on the tool box The plane of the station is the plane of the station's face or edge alignment, as shown in the following figure.

Figure 2-16 Station with Face Alignment



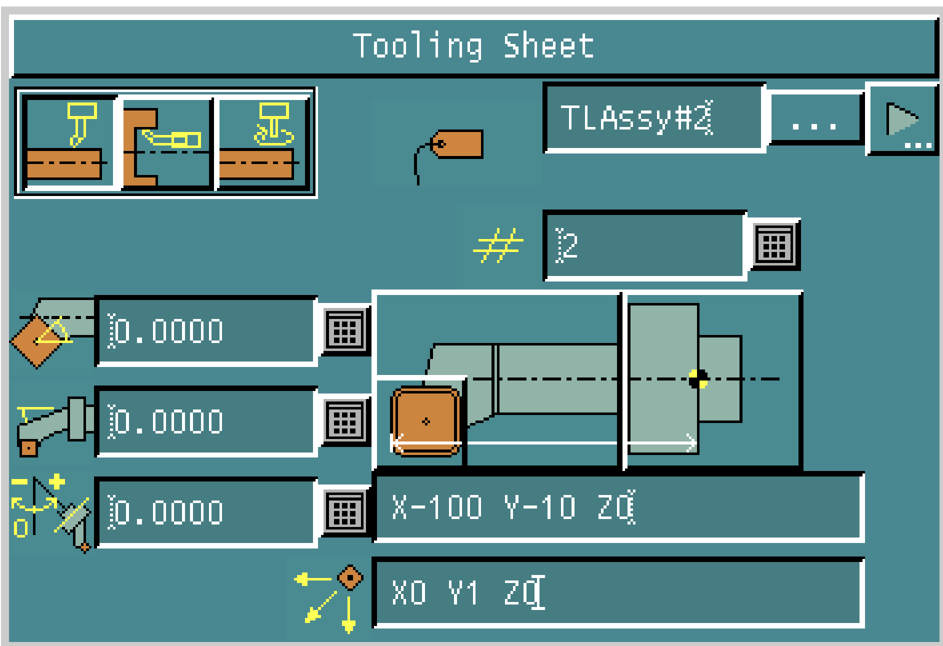
Assignment of Tool Assemblies to Stations

You can use the Tooling sheet to specify the tool assembly for the currently active station with the following options.

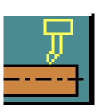
Using this Option

Use the Tooling sheet to:

Figure 2-17 Tooling Sheet

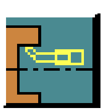


1. Specify the name of the tool assembly in the Tool Assembly Name field.
2. Specify the tool assembly number for this tool assembly in the Tool Assembly Number field.



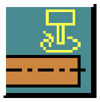
3. Click on this option to specify that the tool assembly is loaded radially into the station with respect to the machine tool.

   OR



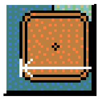
   Click on this option to specify that the tool assembly is loaded axially into the station with respect to the machine tool.

OR



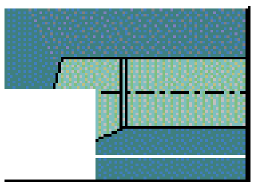
Click on this option to specify if the tool assembly is a driven tool.

4. Specify the angle of the insert to the holder in the Insert Angle field.
5. Specify the angle of the holder to the box in the Setting Angle field.
6. Specify the angle of the station for the tool assembly relative to the active station in the Working Angle field.



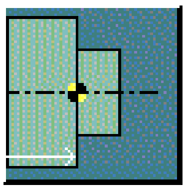
7. Click on the Insert icon to launch a runtime list of inserts defined in the tooling file, and select an Insert. See "Specifying Tool Insert Parameters" on page 2-26.
8. Click on the Holder icon to launch a runtime list of holders defined in the tooling file, and select a Holder. See "Specifying Tool Holder Parameters" on page 2-34.

Figure 2-18 Holder Icon



9. Click on the Box icon to launch a runtime list of tool boxes defined in the tooling file, and select a tool box. See "Specifying Tool Box Parameters" on page 2-36.

Figure 2-19 Box Icon

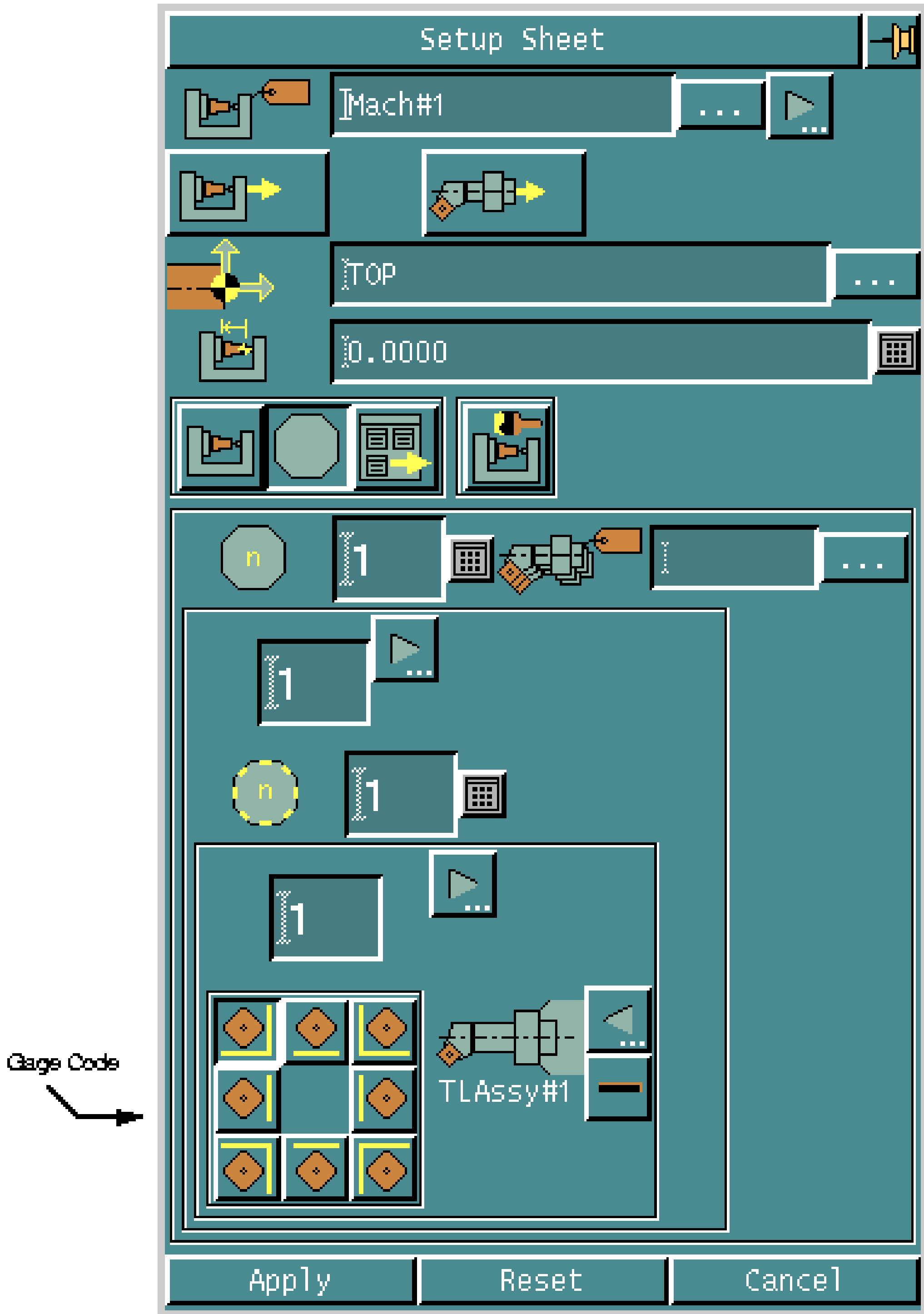


10. Specify the gage length of the tool assembly in the Gage Length field. See "Tool Gage Lengths" on page 2-37 for details.
11. Specify the default drive vector for the tool assembly relative to its definition orientation in the Drive Vector field.
12. Click Apply.

Once you have defined the tool assembly, select the gage code for the assembly in the currently active station, on the Setup Sheet as shown in the following figure.

Please note: You can define tool assemblies in a common orientation, and then use them in any orientation.

Figure 2-20 Setup Sheet



Specifying Tool Insert Parameters

When you click the Insert icon on the Tooling sheet, a runtime list of inserts, or the Insert Selector is launched, where you can select the type of insert you want.

Figure 2-21 Insert Selector

Inserts are defined in the tooling file as shown in the following figure, and appear in the Insert Selector from which you can select the Insert you want.

Figure 2-22 Inserts Defined in the Tooling File

```
//////////////////////////////////
//          INSERTS           //
//////////////////////////////////
// standard   class           name      length length2 radius angle thickness radii          centres
Insert:ISO   ROUND           "R6"      ~      ~       6      ~     2         ~              ~
Insert:ISO   ROUND           "R8"      ~      ~       8      ~     2         ~              ~
Insert:ISO   TRIANGULAR      "TR0"     16     ~       0.05   ~     2         ~              ~
Insert:ISO   SQUARE          "SQ1"     12.7   ~       0.4    ~     2         ~              ~
Insert:ISO   SQUARE          "SQ2"     12.7   ~       0.8    ~     2         ~              ~
Insert:ISO   DIAMOND         "RH1"     16     ~       0.4    35    2         ~              ~
Insert:ISO   DIAMOND         "RH2"     16     ~       0.8    35    2         ~              ~
Insert:ISO   RECTANGLE       "REC1"    12.7   19.05   2.4    ~     2         ~              ~
Insert:ISO   PARALLELOGRAM   "PARA1"   12.7   19.05   2.4    80    2         ~              ~
Insert:ISO   SPECIAL         "SP1"     ~      ~       ~      ~     2   3.18,3.18,3.18 0,0,0,0,6,0,6,0,0
```

CVNC supports insert shapes having a minimum of 1 tip, and a maximum of 4 discrete tips.

A tip is defined as the “fillet” radius or intersection of two of the “cutting” edges of the polygonal shape bounding the complete tooltip geometry. Each tip may be of different radii, including zero radius.

If you have defined one or more tips in such a way that it creates concave regions in the tool shape, CVNC treats these regions as “flat regions”. For example, if you place an elastic band around the defined tool shape, the flat region would be the resultant non-concave polygonal shape.

CVNC supports the following recognized ISO tool shapes, in addition to any 2- to 4-tip tool shape defined by you:

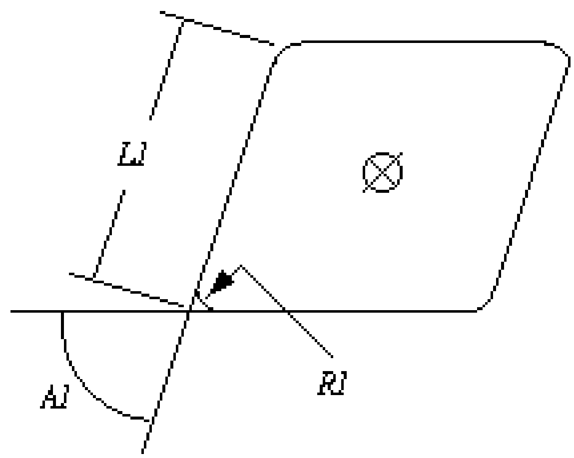
Table 2-1 ISO Codes for Tool Shapes

Tool Shape	Angle	ISO Code
Round		R
Square		S
Triangular		T
Rhomboid/Diamond	35°	V
	55°	D
	80°	C
	86°	M
Parallelogram	55°	K
	70°	
	82°	B
	85°	A
Rectangle		L

ISO Standard

Under the ISO standard, the Insert size is defined by supplying the maximum theoretical length of the cutting edge. This length is converted from a real into a 2 digit code. Combined with the included tip angle and tip radius, this information is sufficient to define the complete ISO standard tip shape. It is necessary that you enter explicit values.

Figure 2-23 Insert under ISO Standard

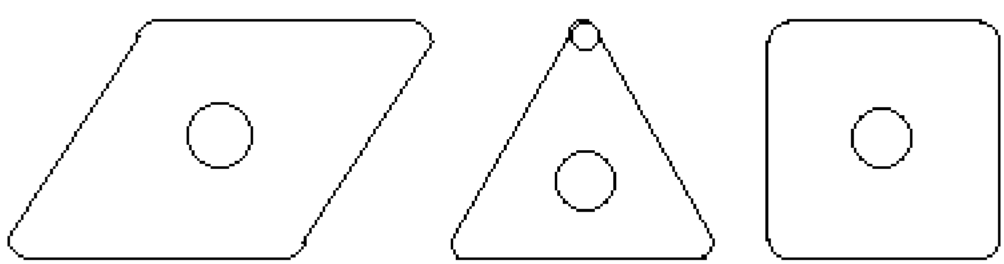


ISO Tool Shape Definition and Orientation

CVNC has a well defined method to control your input during the definition of ISO cutting tool inserts.

For all insert shapes from the previous ISO list, other than round, CVNC displays the main Primary insert radius at the bottom left with one of the two main cutting edges positioned horizontally, running left to right as shown in the following diagram. From this definition you can then apply the necessary rotation angle to orient the insert into the holder.

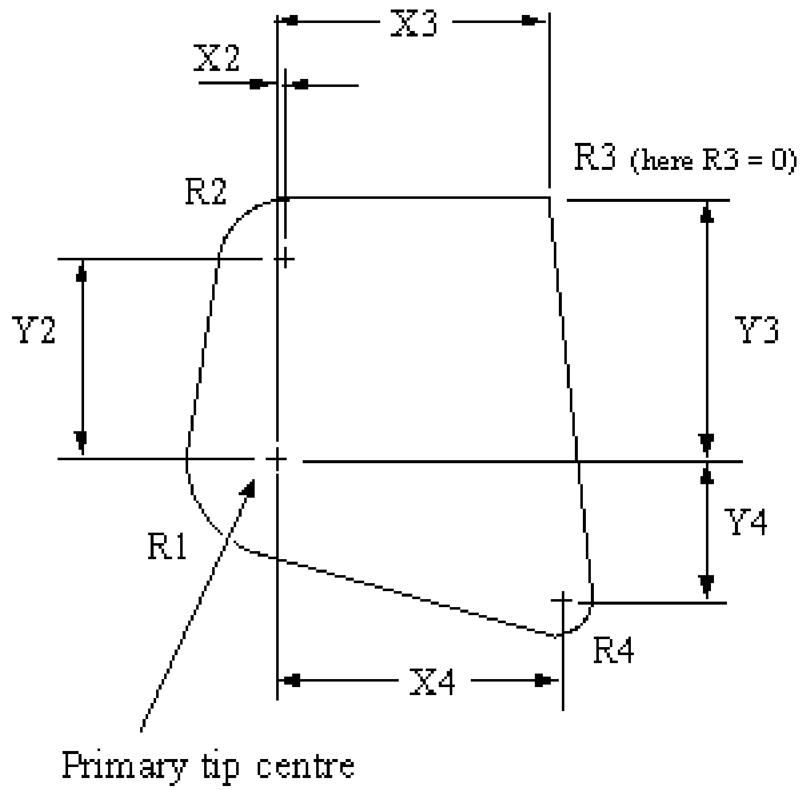
Figure 2-24



Due to the multiple requirements of generating output, and working with different tips, the first tip from which all others are defined is considered as the Primary tip from which both CVNC default toolpath graphics for SHOW TOOL and OUTPUT generation occur. However, you may need to select one of the other insert tips for part programming. This would require the output point to be changed from the Primary tip to the current tip, as explained later.

The following illustration shows a user defined multiple tip special.

Figure 2-25 Example of a Four-tip Tool



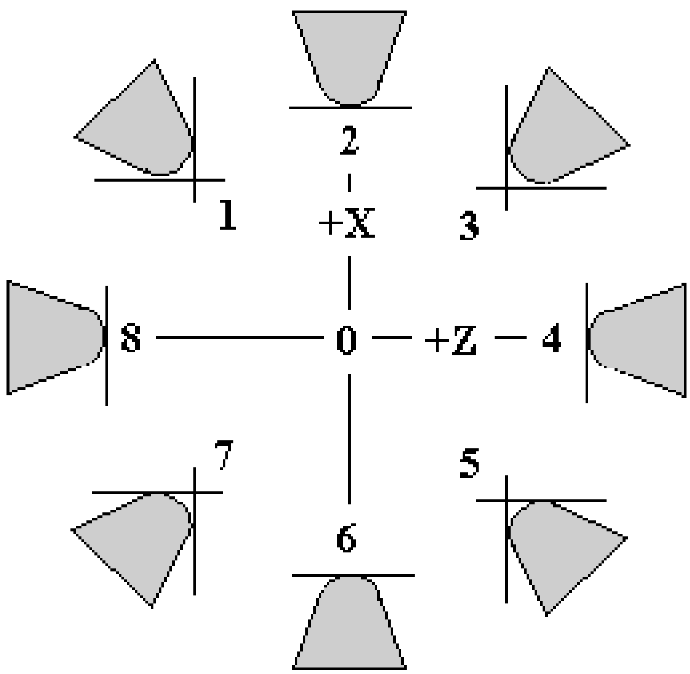
Tool Insert Orientation Values

You must define all ISO, and user defined single tip tools as shown in the previous illustrations. Once you define the tool, the insert or tip needs to be rotated into an orientation applicable to the manufacturing process and tool assembly. To do this, an angular value must be applied about which the whole insert is rotated relative to this axis. Positive values generate a counterclockwise rotation. User defined multi-tip tools rotate about this angle, around the primary tip center and a horizontal axis passing through this location.

A common ISO standard, defined by the number 0-8, is used to define the actual program point relative to the orientation of the tool tip. The standard is as shown in the following table and illustration:

Table 2-2 Common ISO Standard

Code	Tool tip direction	
0	Tip centre point	
1	-X	+Z
2	-X	
3	-X	-Z
4	-Z	
5	+X	-Z
6	+X	
7	+X	+Z
8	+Z	

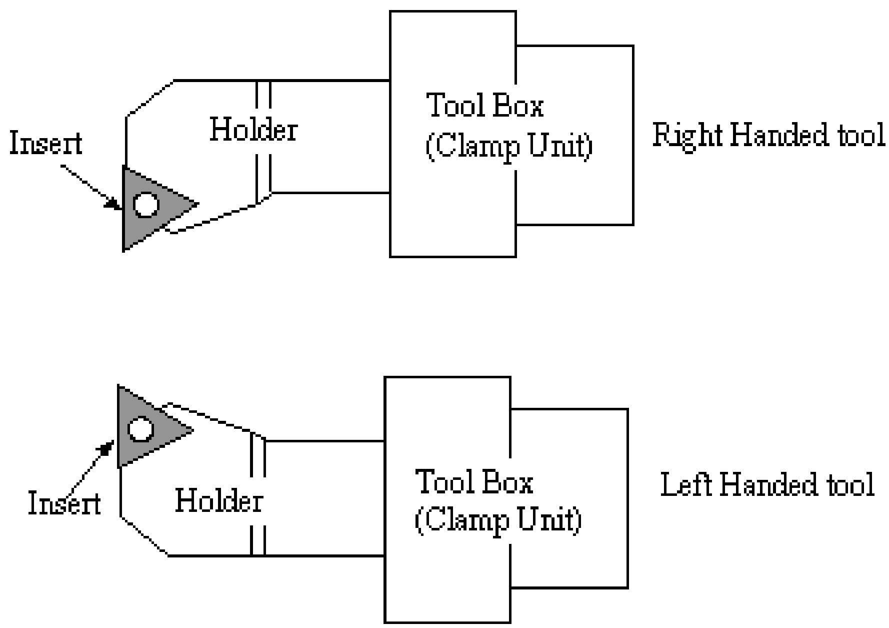


You must identify the type of tool assembly you wish to define, that is, a Right- or Left-handed tooling assembly.

An example of common tool definition orientation for a right-handed tool would be with the tool insert at the lower left relative to the holder, with tool holder and box building away from left to right.

Left handed tools build with the insert at upper left, again relative to the holder, building left to right. With reference to the previous orientation code chart, this example defines a non oriented tooling assembly of orientation code 3 for the right hand tooling, and an orientation code of type 5 for the left hand tool. This code changes as the tool is oriented into its machine tool or turret setting.

Figure 2-26 Right- and Left-handed tool



Definition of Tool Primary Cutting Axis Vector

Producing grooves in any direction is not entirely definable from the geometric definition of a groove. Typical grooves have 2 sides and a base, each side may taper with respect to the machine axial system, and may have different angular taper values, as may the base of the groove. Hence it is not always possible to extract the required direction for machining from the geometry. Also, during manufacture, certain turning processes may well utilize common tool types each of which would have a predetermined cutting direction

During definition of the tool assembly, you can build into the tool definition the main cutting direction vector for each assembly. This alleviates the need to continually re-input the drive vector controlling the main tool motion for each and every tool assembly, every time turning, grooving or facing is required.

However, you can redefine this vector for each task to ensure that optimal material cutting conditions are achieved. The vector is mapped into the tool assemblies working orientation from the common definition orientation.

User Definable Primary Tip

It is essential for you to identify the primary tip. In defining an ISO tip shape, if you follow the rules laid down for tip definition in “Specifying Tool Insert Parameters” on page 2-26, you can assume that the lower left hand tip will always be the primary tip. All others will be secondary tips.

When defining a user special, if you follow the rules laid down in “Specifying Tool Insert Parameters” on page 2-26, you can assume that the complete insert shape is to be defined around a reference point from which all tips are offset. You can then use a common method of defining tips, either clockwise or counter clockwise from the first tip which is considered the primary tip. The order of the tips is always clockwise for ISO, or special inserts.

The true centre of the primary tip is always used as the default point from which the OUTPUT data, either CLFILE or APTSOURCE is generated. This is also the default point used during entry into CVNC by SHOW TOOL during tool display.

User Selectable Tip for Programming

During programming it is not always convenient to program using only the defined primary tip. It may be necessary to switch the programmed tip to better utilize the supplied model geometry, for example, a final series of operations defined by you on a feature like groove. While machining, the tool needs to contact the left hand side of the groove with the left hand edge of the tool, and vice versa when machining the right hand edge. Therefore, if you wish to switch tips during programming so that you can more easily monitor the toolpath being produced, CVNC produces toolpath graphics representing the centre of the selected tip, and not the centre of the complete insert. Output is to the centre of the primary tip.

User Selectable Tip for Output

The default for OUTPUT is always the centre of the primary tip. However, when producing finishing operations, it is necessary to produce output that reflects the location of the selected secondary tip, and not the primary tip. In such cases, the values you need to specify are the radius of the primary tip, the radius of the secondary tip, and the true machine axial and radial gage lengths for the newly selected tip.

Separate Z and X Registers per Tip

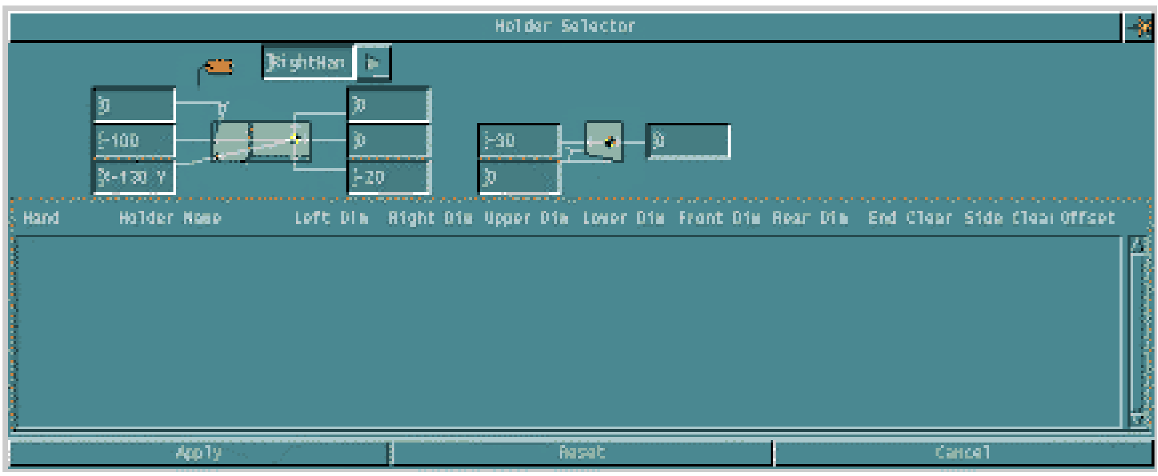
It is necessary to maintain an upto-date machine tool axial and radial co-ordinate value for the primary tip for each tool assembly. This is to ensure that at a CVNC tool change, the correct offset values can be placed in the output file.

CVNC also maintains upto-date machine tool axial and radial co-ordinate offsets for each secondary tip. This is to ensure that as you select a secondary tip for programming from which the output is to be generated, CVNC can then generate the correct axial and radial gage length output.

Specifying Tool Holder Parameters

The Tooling sheet displays a 2D outline of the Insert, Holder, and the Tool Box. When you click the Holder icon on the Tooling sheet, a runtime list of holders, or the Holder Selector is launched, where you can select the type of holder you want.

Figure 2-27 Holder Selector



Holders are defined in the tooling file as shown in the following figure, and appear in the Holder Selector from which you can select the Holder you want.

Figure 2-28 Holders Defined in the Tooling File

```
//////////////////////////////////
//        HOLDERS             //
//////////////////////////////////
//hand             name   left  right  upper  lower  front  rear  endclear  sideclear  offset
Holder:RIGHT     "H1"   -120   0     20     -30     0     -40    5          4        -110,-20,0
Holder:RIGHT     "H2"   -100   10    0      -40     0     -30    2          3        -95,-35,0
Holder:LEFT      "L1"   -120   0     20     -30     0     -40    5          4        -110,-20,0
Holder:LEFT      "L2"   -100   10    0      -40     0     -30    2          3        -95,-35,0
Holder:NEUTRAL   "N1"   -100   0     25     -25     0     -40    0          0        -95,0,0
Holder:NEUTRAL   "N2"   -100   0     25     -25     0     -40    0          0        -150,0,0
```

In the overall assembly definition, each insert tip has a true axial and radial gage length. Due to tolerance stack up during assembly builds, it is physically impossible to achieve all item to item dimensions as well as the overall assembly dimension, that is, the axial and radial gage lengths. Therefore, the Holder to Box dimension is used to take up any float or discrepancy when gage lengths and assembly build values do not match.

While defining the tool assembly, you can define suitable holders, locate it, and graphically see it inserted into the tool assembly.

Specifying Tool Box Parameters

The Tooling sheet displays a 2D outline of the Insert, Holder, and the Tool Box. When you click the Tool Box icon on the Tooling sheet, a runtime list of tool boxes, or the Box Selector is launched, where you can select the type of tool box you want.

Figure 2-29 Box Selector

Tool Boxes are defined in the tooling file as shown in the following figure, and appear in the Box Selector from which you can select the Tool box you want.

Figure 2-30 Tool Boxes Defined in the Tooling File

```
/////////////////////////////////
//         BOX           //
/////////////////////////////////
//name     left  right  upper  lower  front  rear
Box:"B1"    0     80     0     -60    30     0
Box:"B2"   -20    40     0     -40    80     0
```

While defining the tool assembly, you can define suitable boxes, such that after having selected the required insert and holder, you can select the required tool box and graphically see this inserted into the tool assembly.

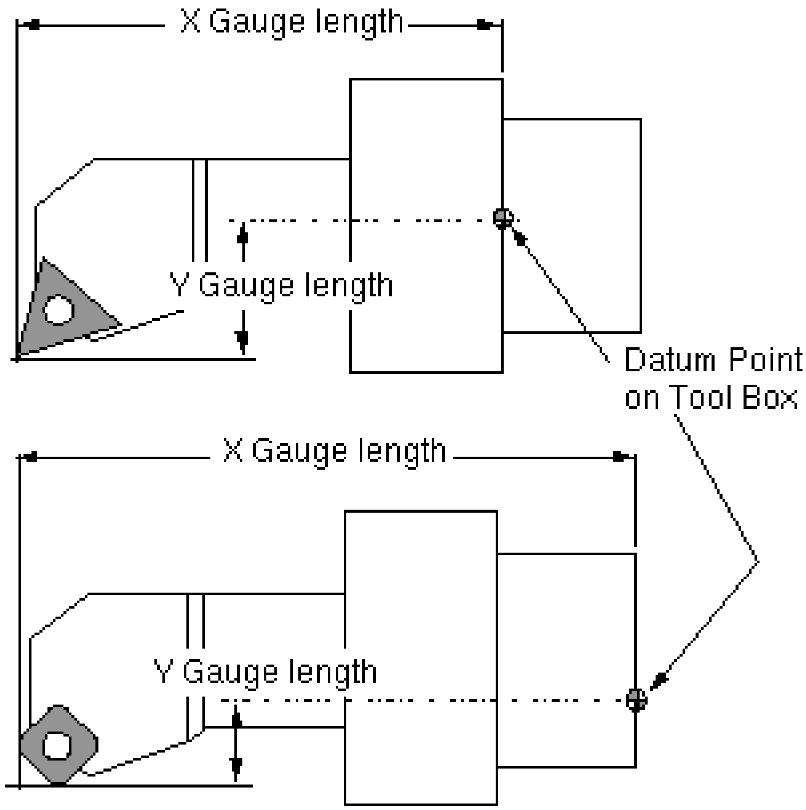
Tool Gage Lengths

You need to specify the controlling gage lengths while setting up the overall tool assembly. Each tool assembly has two values. They are the axial and radial dimensions of the tool tip extremities to the datum point of the tool box, and not to the center of the primary or secondary tip centers. These values, specified during tool assembly generation, are then maintained by CVNC as the correct machine orientation axial and radial values, so that they can be passed through at the time of tool change to the output file by the TURRET command. You can also access these values at any time.

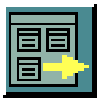
The assembly build process uses locations to locate the insert onto the holder and the holder into the tool box. This may not mathematically equal the required gage lengths. The gage length is of far more importance than the overall build up dimensions, and therefore, once built, the holder slides in axial and radial definition space to ensure that the gage length is met.

The following illustration shows examples of the gage length for a single tip single quadrant tool and a multiple tip two quadrant tool. Both examples use right handed tool definitions.

Figure 2-31 Examples of Gage Lengths

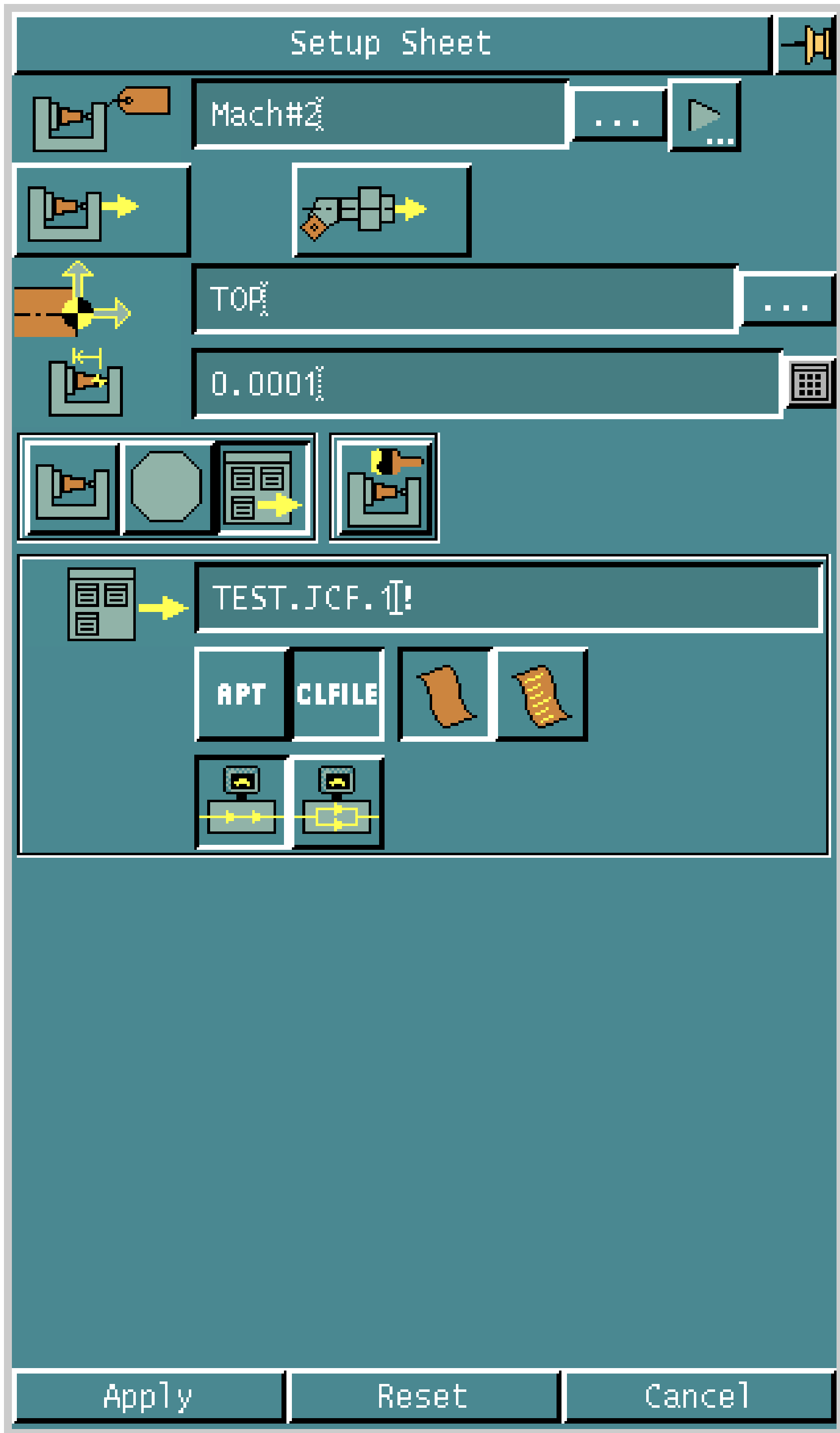


Specifying Output Generation Parameters



You can use this option to specify output generation parameters on the Setup sheet as shown in the following figure.

Figure 2-32 Setup Sheet for Output Generation Parameters



1. Specify the name of the output file in which the output for all jobs in the JCF is to be created, in the Output File field.

2. Specify whether output is APT source or CLFile. If the output is CLFile, specify whether only binary, or both binary and the text version of the CLFile output is to be created.
3. Specify that the output is to be created as part of the local CADDS 5 process when the JCF is exited, or output is to be created as a background process when the JCF is exited.
4. Click Apply if you have finished specifying all parameters.

Chapter 3

Discrete Tool Motion

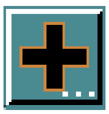
This chapter details various functionalities that support discrete tool motion such as location based motion, entity based motion, and absolute and incremental coordinate motion. The following sections describe these processes in detail.

- Specifying Tasks
- Specifying Absolute Coordinate Motion Parameters
- Specifying Incremental Coordinate Motion Parameters
- Specifying Location Motion Parameters
- Specifying Entity Based Motion Parameters
- Specifying XY-Direction Entity Based Motion Parameters
- Executing the Job

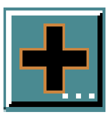
Specifying Tasks

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.

4. Choose the Job Block option from the NCBuilder task set to launch the NCBuilder property sheet or Job Sheet.

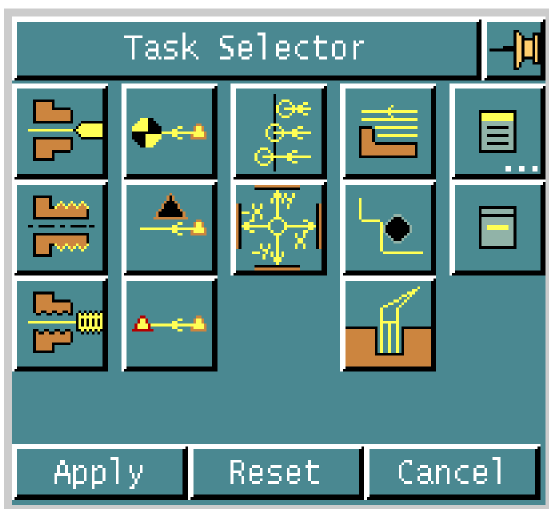


5. Choose the Add Operation option from the NCBuilder property sheet. The Operation Sheet appears.



6. Choose the Add Task option from the operation sheet. The Task Selector palette appears.

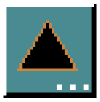
Figure 3-1 Task Selector



Please note: You should define the machine tool parameters and tooling parameters before specifying any task. See Chapter 2, “Machine Tool Definitions” for details.

7. Choose an option from the Task Selector palette. The task name and its icon appear in the Task Selector palette. The sequence number of the task also appears. For example, if the task is the second profile task in the operation, it has the number 2.
8. Click Apply.

   The task now appears in the Operation sheet which changes when the task is highlighted.

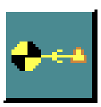


9. Click the Edit option on the Operation sheet to launch the appropriate task sheet to edit the selected task.

   For more details on options on the Operation sheet, see “Options on the Operation Sheet” on page 5-5. You can also refer to the *NC Builder User Guide and Menu Reference* for more information.

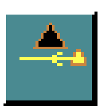
This chapter discusses the various discrete tool motion options available to you in the Task Selector.

Discrete Tool Motion Options



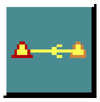
SPECIFYING ABSOLUTE COORDINATE MOTION PARAMETERS
Displays a property sheet that enables you to specify parameters for an absolute coordinate motion task to add to the current operation.

See “Specifying Absolute Coordinate Motion Parameters” on page 3-5 for details.



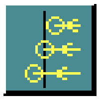
SPECIFYING INCREMENTAL COORDINATE MOTION PARAMETERS
Displays a property sheet that enables you to specify parameters for an incremental coordinate motion task to add to the current operation.

See “Specifying Incremental Coordinate Motion Parameters” on page 3-11 for details.



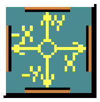
SPECIFYING LOCATION MOTION PARAMETERS
Displays a property sheet that enables you to specify parameters for a location motion task to add to the current operation.

See “Specifying Location Motion Parameters” on page 3-17 for details.



SPECIFYING ENTITY BASED MOTION PARAMETERS
Displays a property sheet that enables you to specify parameters for an entity based motion task to add to the current operation.

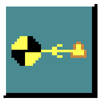
See “Specifying Entity Based Motion Parameters” on page 3-25 for details.



SPECIFYING XY-DIRECTION ENTITY BASED MOTION PARAMETERS
Displays a property sheet that enables you to specify parameters for an X, or Y direction entity based motion task to add to the current operation.

See “Specifying XY-Direction Entity Based Motion Parameters” on page 3-39 for details.

Specifying Absolute Coordinate Motion Parameters

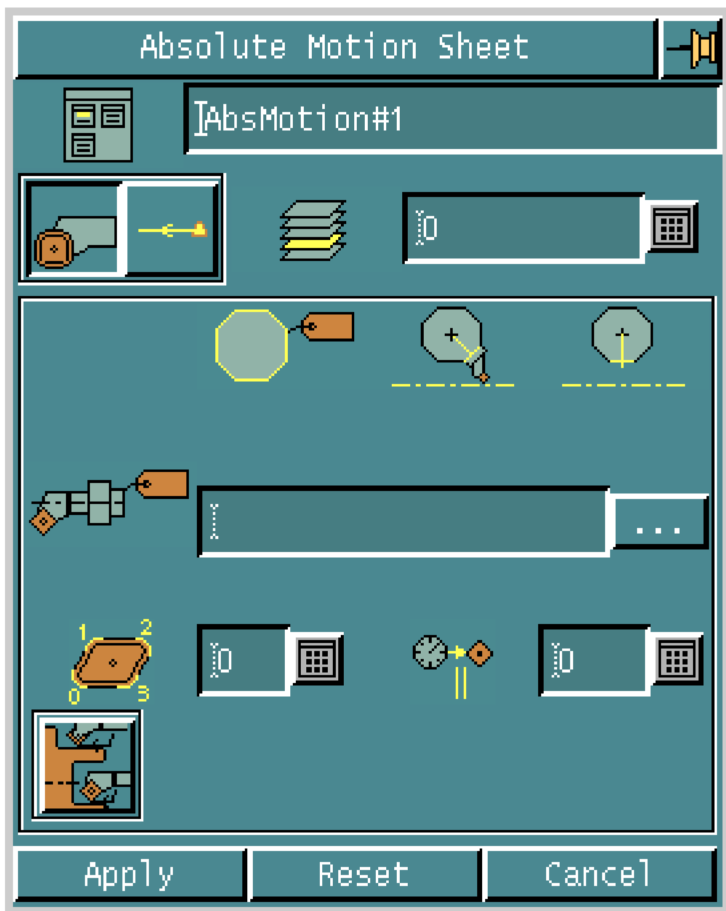


You can use the Absolute Coordinate Motion option on the Task Selector to add an absolute coordinate motion task to the current operation.

Using this Option

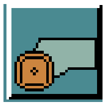
1. Highlight the task from the list of tasks on the Operation sheet, and choose the Edit option to launch the Absolute Motion Sheet.

Figure 3-2 Absolute Motion Sheet



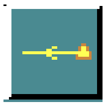
2. Specify the name of the task in the Task Name field.
3. Specify the layer on which the tool path will be placed in the Layer field.

Options on the Absolute Motion Sheet



SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 3-6 for details.

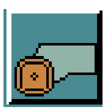


SPECIFYING GENERAL MACHINING PARAMETERS
Displays a property sheet that enables you to specify general machining parameters for the current task.

See “Specifying General Machining Parameters” on page 3-9 for details.

Please note: For each of these options, a different set of parameters appears on the same Absolute Motion sheet.

Specifying Tool Assembly Selection Parameters

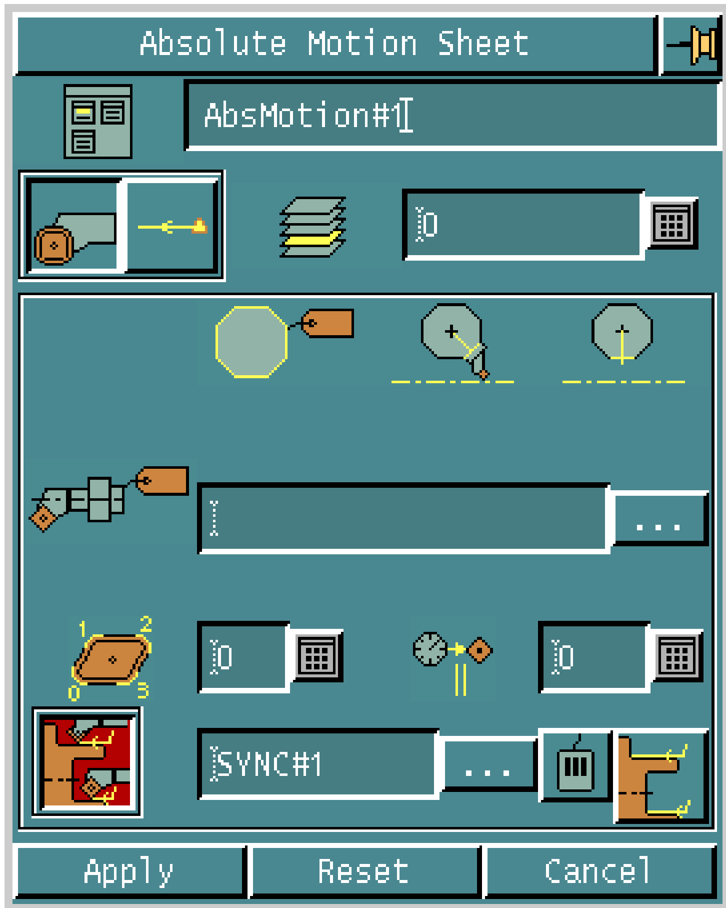


You can use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

Using this Option

1. Choose the Tool Parameters option from the Absolute Motion sheet. The set of parameters for the tool assembly appears on the sheet.

Figure 3-3 Absolute Motion Sheet with Tool Assembly Selection Parameters

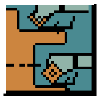


2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field.

   OR

   Choose a tool assembly from the Tool Selector scroll list.

3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



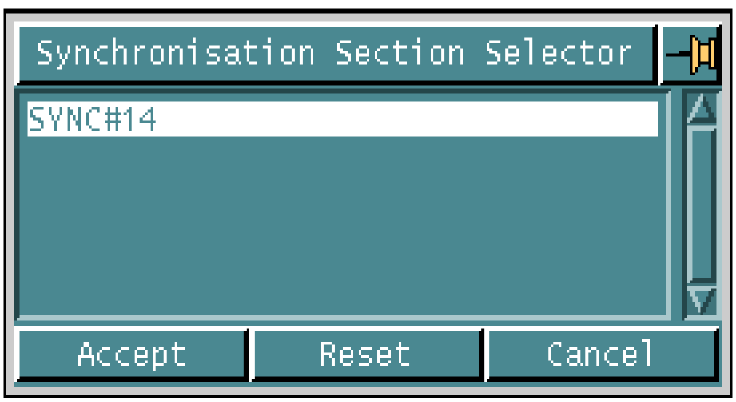
5. Choose the Sync option if you want to add the current task to the most recent synchronization section.

A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

6. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 3-4 Synchronization Section Selector Scroll List

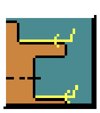


- Choose a synchronization section from the list.
- Click Accept.

OR

You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

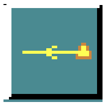
OR



Choose the New Sync Section option to create a new synchronization section to add the task.

7. Click Apply if you have finished entering all parameters for the task.

Specifying General Machining Parameters

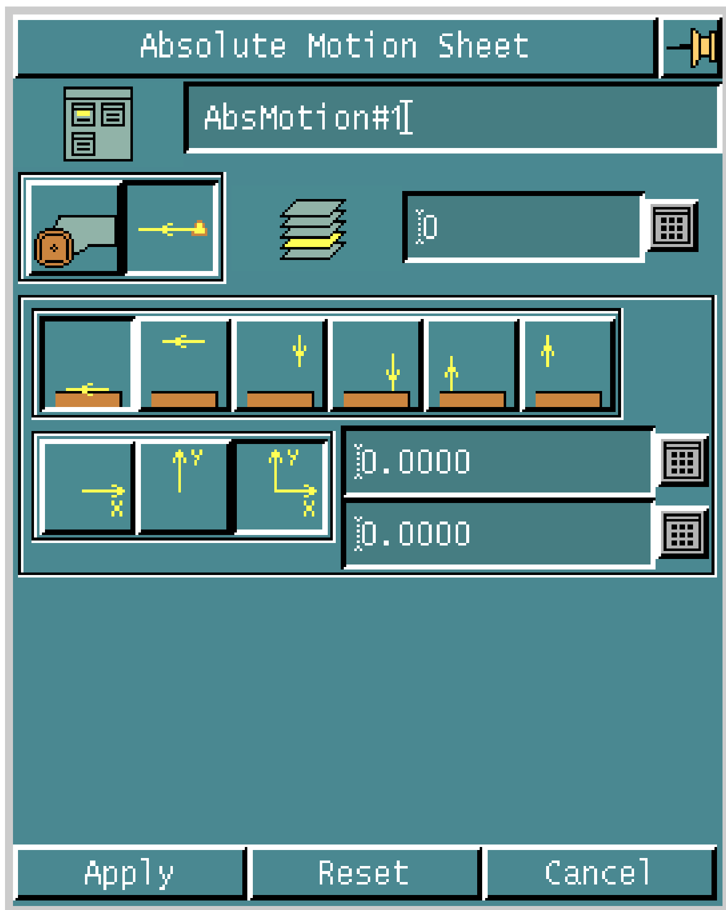


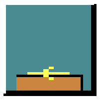
You can use the MACHINING PARAMETERS option to specify general machining parameters for the current task.

Using this Option

1. Choose the Machining Parameters option from the Absolute Motion sheet. The set of general machining parameters appears on the sheet.

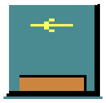
Figure 3-5 Absolute Motion Sheet with General Machining Parameters





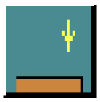
2. Click on the Cut Feed Motion option to specify that motion is to be made at cut feed rate.

   OR



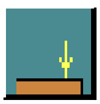
   Click on the Rapid Feed Motion option to specify that motion is to be made at rapid feed rate.

   OR



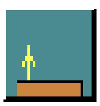
   Click on the Approach Feed Motion option to specify that motion is to be made at approach feed rate.

   OR



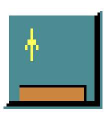
   Click on the Plunge Feed Motion option to specify that motion is to be made at plunge feed rate.

   OR

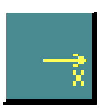


   Click on the Retract Feed Motion option to specify that motion is to be made at retract feed rate.

   OR

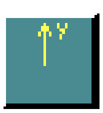


   Click on the Clear Feed Motion option to specify that motion is to be made at clear feed rate.



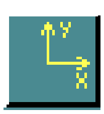
3. Click on the XAbsolute Motion option to specify absolute motion in X, and enter the absolute motion value in the XLoc Motion Value field.

   OR



   Click on the YAbsolute Motion option to specify absolute motion in Y, and enter the absolute motion value in the YLoc Motion Value field.

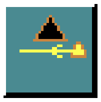
   OR



   Click on the XYAbsolute Motion option to specify absolute motion in X, and Y, and enter the absolute motion values in both the XLoc Motion Value field, and the YLoc Motion Value field.

4. Click Apply if you have finished entering all parameters for the task.

Specifying Incremental Coordinate Motion Parameters

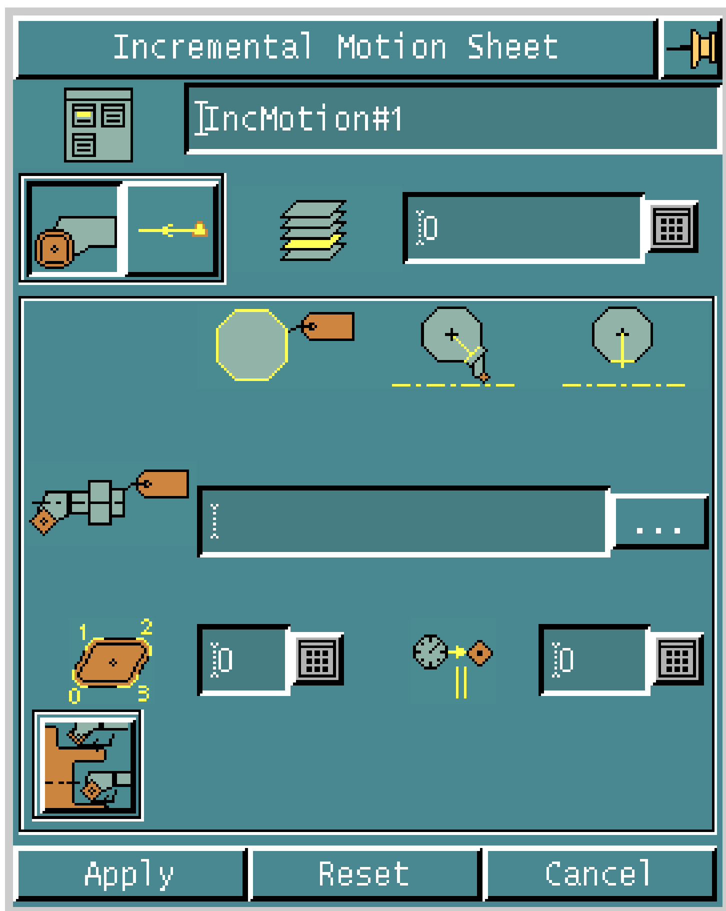


You can use the Incremental Coordinate Motion option on the Task Selector to add an incremental coordinate motion task to the current operation.

Using this Option

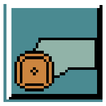
1. Highlight the task from the list of tasks on the Operation sheet, and choose the Edit option to launch the Incremental Motion Sheet.

Figure 3-6 Incremental Motion Sheet



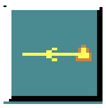
2. Specify the name of the task in the Task Name field.
3. Specify the layer on which the tool path will be placed in the Layer field.

Options on the Incremental Motion Sheet



SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 3-12 for details.

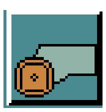


SPECIFYING GENERAL MACHINING PARAMETERS
Displays a property sheet that enables you to specify general machining parameters for the current task.

See “Specifying General Machining Parameters” on page 3-15 for details.

Please note: For each of these options, a different set of parameters appears on the same Incremental Motion sheet.

Specifying Tool Assembly Selection Parameters

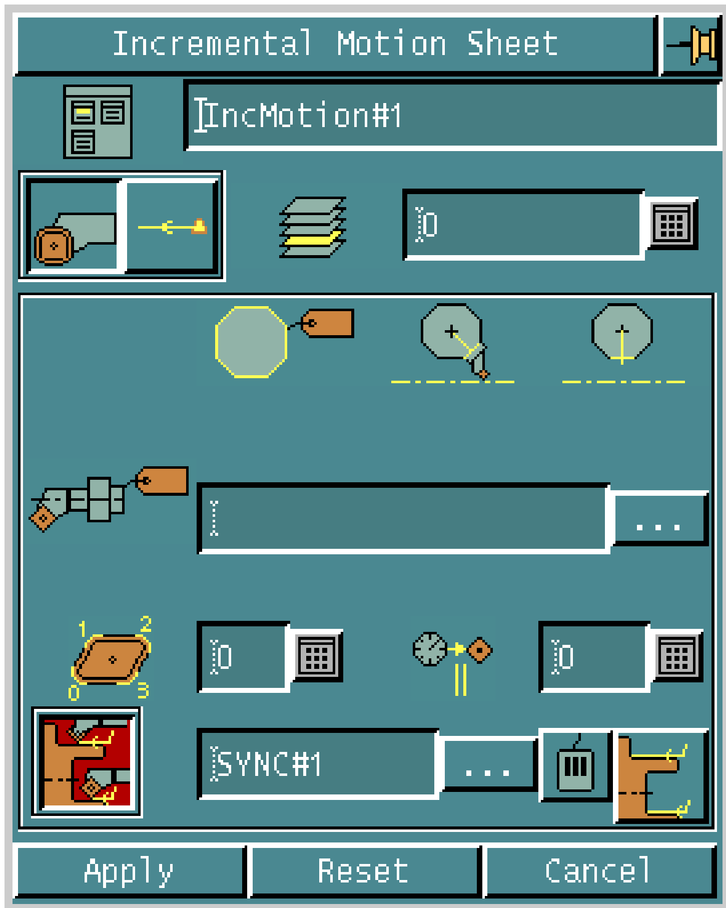


You can use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

Using this Option

1. Choose the Tool Parameters option from the Incremental Motion sheet. The set of parameters for the tool assembly appears on the sheet.

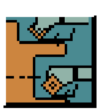
Figure 3-7 Incremental Motion Sheet with Tool Assembly Selection Parameters



2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field.

   OR

   Choose a tool assembly from the Tool Selector scroll list.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



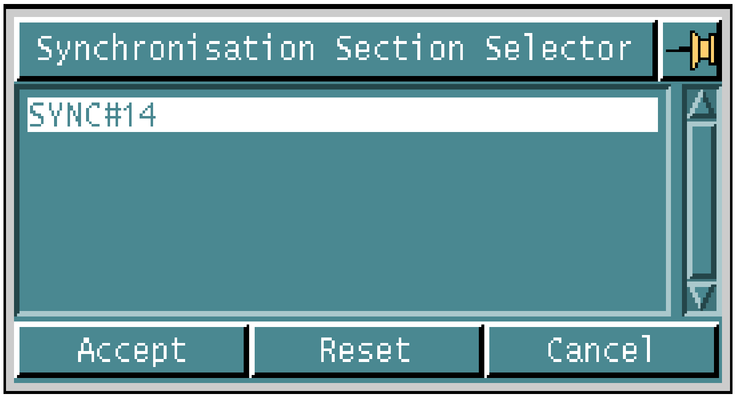
5. Choose the Sync option if you want to add the current task to the most recent synchronization section.

A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

6. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 3-8 Synchronization Section Selector Scroll List

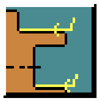


- Choose a synchronization section from the list.
- Click Accept.

OR

You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NC Builder. The current task is then added to the same synchronization section as that of the digitized task.

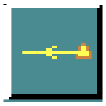
OR



Choose the New Sync Section option to create a new synchronization section to add the task.

7. Click Apply if you have finished entering all parameters for the task.

Specifying General Machining Parameters

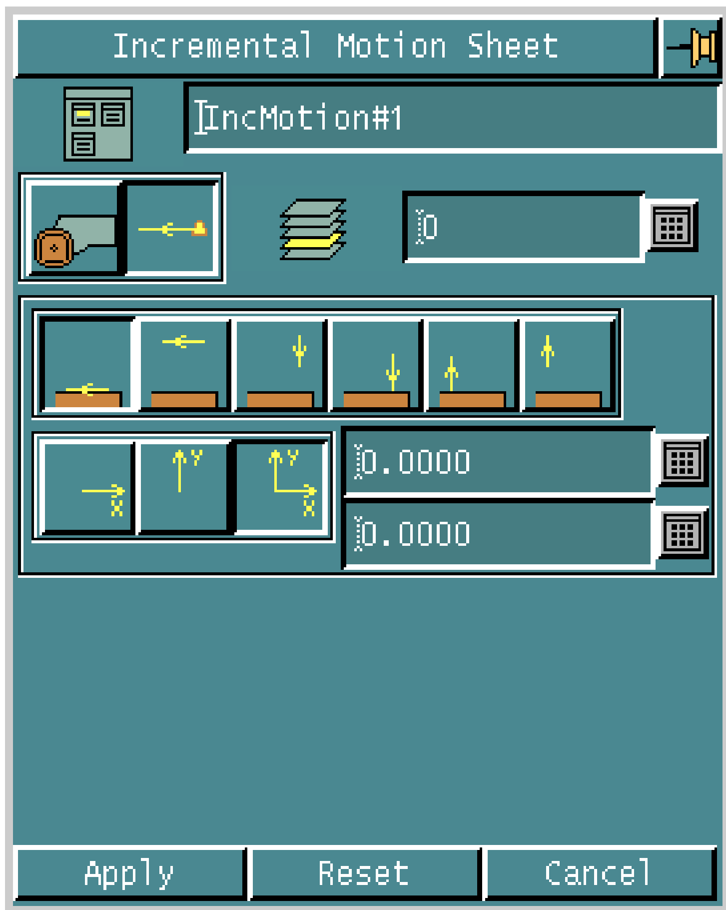


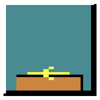
You can use the MACHINING PARAMETERS option to specify general machining parameters for the current task.

Using this Option

1. Choose the Machining Parameters option from the Incremental Motion sheet. The set of general machining parameters appears on the sheet.

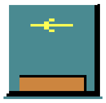
Figure 3-9 Incremental Motion Sheet with General Machining Parameters





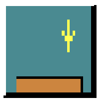
2. Click on the Cut Feed Motion option to specify that motion is to be made at cut feed rate.

   OR



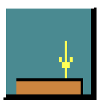
   Click on the Rapid Feed Motion option to specify that motion is to be made at rapid feed rate.

   OR



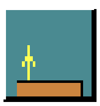
   Click on the Approach Feed Motion option to specify that motion is to be made at approach feed rate.

   OR



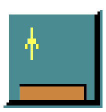
   Click on the Plunge Feed Motion option to specify that motion is to be made at plunge feed rate.

   OR

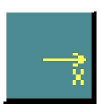


   Click on the Retract Feed Motion option to specify that motion is to be made at retract feed rate.

   OR

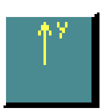


   Click on the Clear Feed Motion option to specify that motion is to be made at clear feed rate.



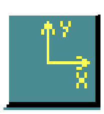
3. Click on the XAbsolute Motion option to specify absolute motion in X, and enter the absolute motion value in the XLoc Motion Value field.

   OR



   Click on the YAbsolute Motion option to specify absolute motion in Y, and enter the absolute motion value in the YLoc Motion Value field.

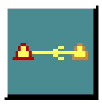
   OR



   Click on the XYAbsolute Motion option to specify absolute motion in X, and Y, and enter the absolute motion values in both the XLoc Motion Value field, and the YLoc Motion Value field.

4. Click Apply if you have finished entering all parameters for the task.

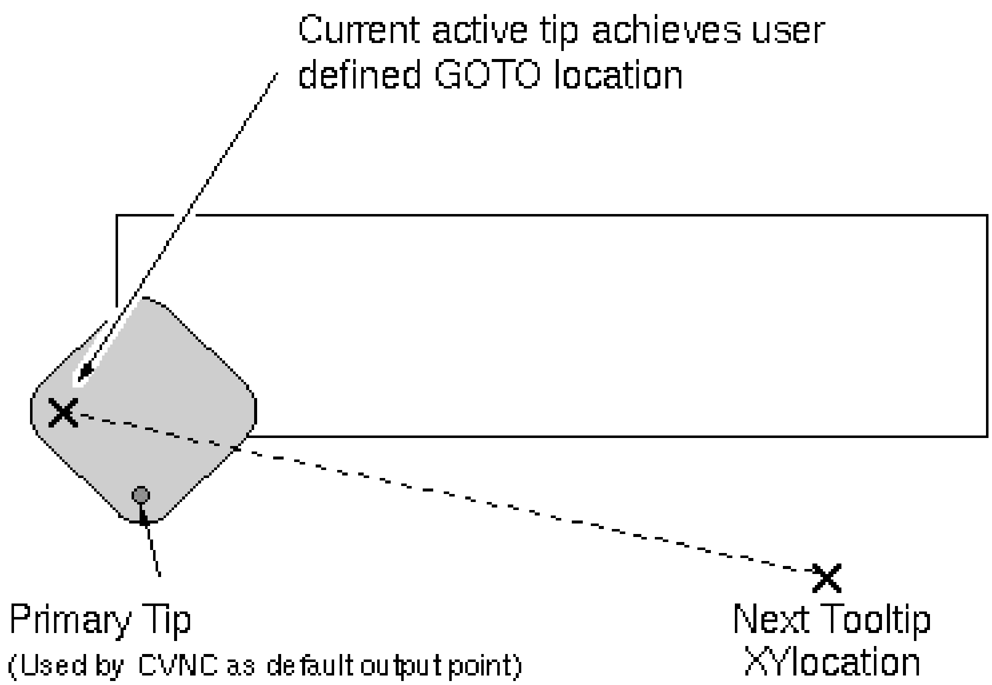
Specifying Location Motion Parameters



You can use the Location Motion option on the Task Selector to add a location motion task to the current operation.

All discrete motion controlled by the selection or provision of location data is used to control the final destination of the centre of the current active tip.

Figure 3-10

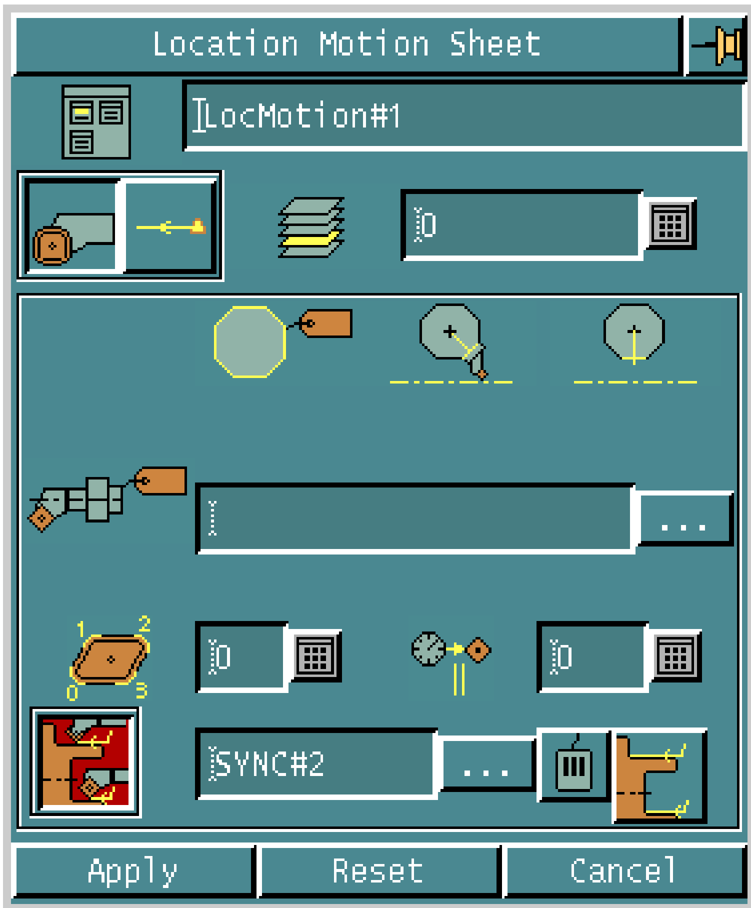


Please note: Motion is direct from the current location to the identified location.

Using this Option

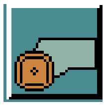
1. Highlight the task from the list of tasks on the Operation sheet, and choose the Edit option to launch the Location Motion Sheet.

Figure 3-11 Location Motion Sheet



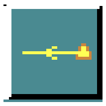
2. Specify the name of the task in the Task Name field.
3. Specify the layer on which the tool path will be placed in the Layer field.

Options on the Location Motion Sheet



SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 3-19 for details.

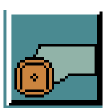


SPECIFYING GENERAL MACHINING PARAMETERS
Displays a property sheet that enables you to specify general machining parameters for the current task.

See “Specifying General Machining Parameters” on page 3-22 for details.

Please note: For each of these options, a different set of parameters appears on the same Location Motion sheet.

Specifying Tool Assembly Selection Parameters

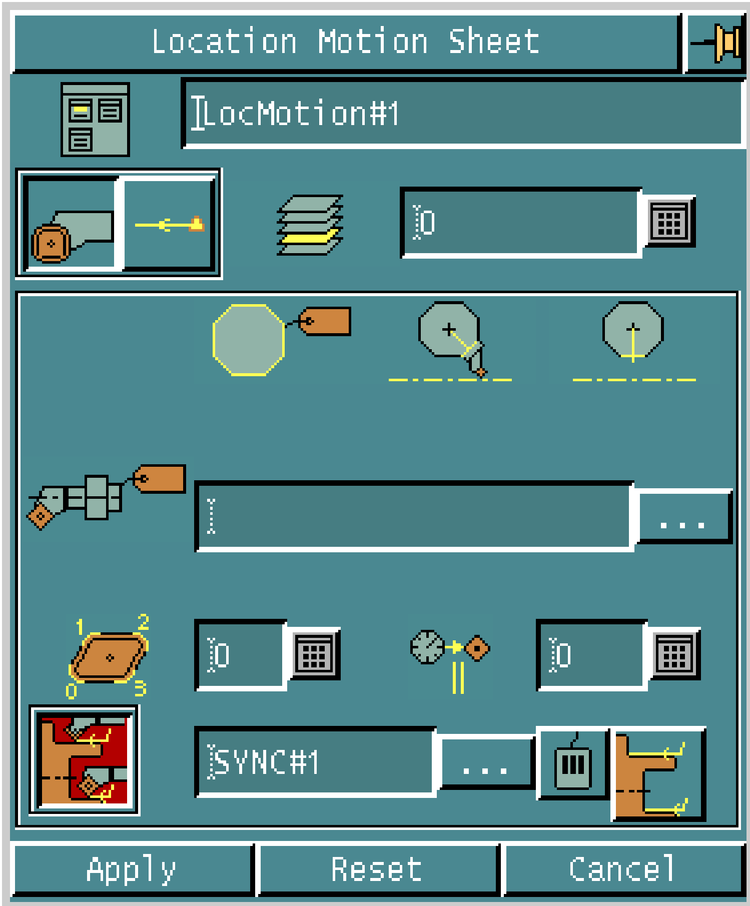


You can use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

Using this Option

1. Choose the Tool Parameters option from the Location Motion sheet. The set of parameters for the tool assembly appears on the sheet.

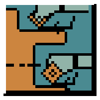
Figure 3-12 Location Motion Sheet with Tool Assembly Selection Parameters



2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field.

   OR

   Choose a tool assembly from the Tool Selector scroll list.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



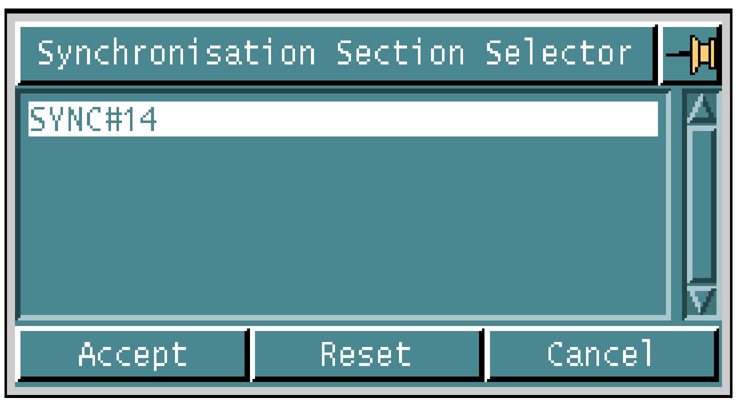
5. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

6. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 3-13 Synchronization Section Selector Scroll List

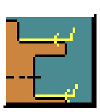


- Choose a synchronization section from the list.
- Click Accept.

OR

You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

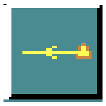
OR



Choose the New Sync Section option to create a new synchronization section to add the task.

7. Click Apply if you have finished entering all parameters for the task.

Specifying General Machining Parameters

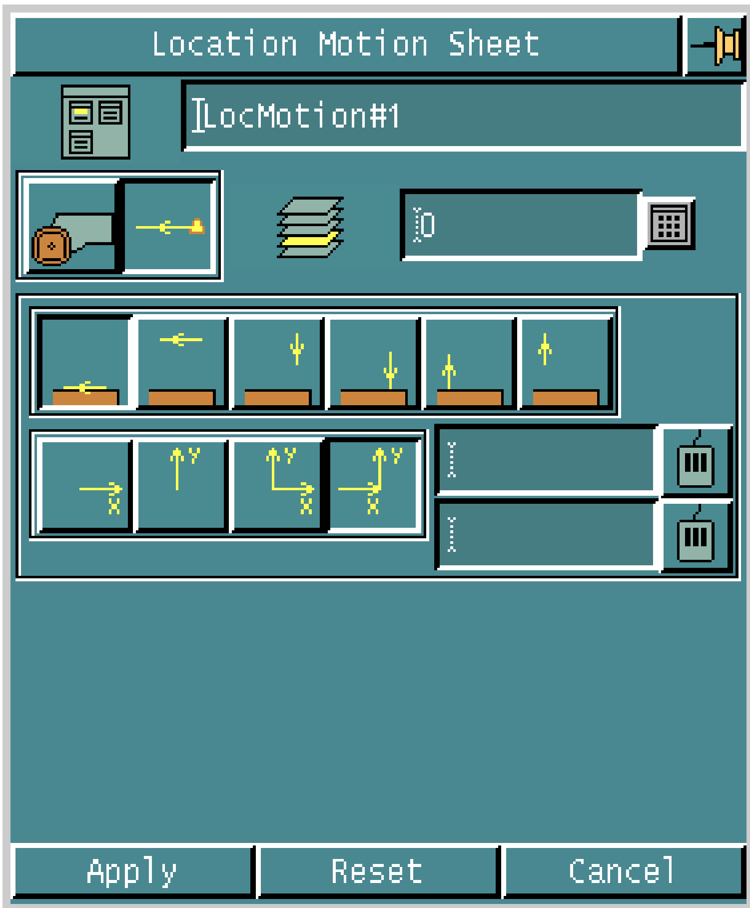


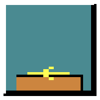
You can use the MACHINING PARAMETERS option to specify general machining parameters for the current task.

Using this Option

1. Choose the General Machining Parameters option from the Location Motion sheet. The set of general machining parameters appears on the sheet.

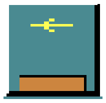
Figure 3-14 Location Motion Sheet with General Machining Parameters





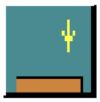
2. Click on the Cut Feed Motion option to specify that motion is to be made at cut feed rate.

   OR



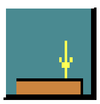
   Click on the Rapid Feed Motion option to specify that motion is to be made at rapid feed rate.

   OR



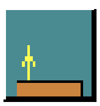
   Click on the Approach Feed Motion option to specify that motion is to be made at approach feed rate.

   OR



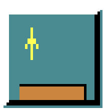
   Click on the Plunge Feed Motion option to specify that motion is to be made at plunge feed rate.

   OR

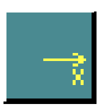


   Click on the Retract Feed Motion option to specify that motion is to be made at retract feed rate.

   OR

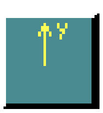


   Click on the Clear Feed Motion option to specify that motion is to be made at clear feed rate.



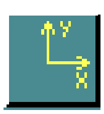
3. Click on the XLocation Motion option to specify motion to the location in X, and enter the location in the XMotion Location field.

   OR



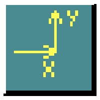
   Click on the YLocation Motion option to specify motion to the location in Y, and enter the location in the YMotion Location field.

   OR



   Click on the XYLocation Motion option to specify motion to the location in X, and Y, and enter the location in the XMotion Location field.

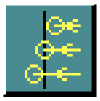
OR



Click on the XLocation YLocation Motion option to specify motion to the location in X, and then in Y, and enter the first location in the XMotion Location field, and the second location in the YMotion Location field.

4. Click Apply if you have finished entering all parameters for the task.

Specifying Entity Based Motion Parameters

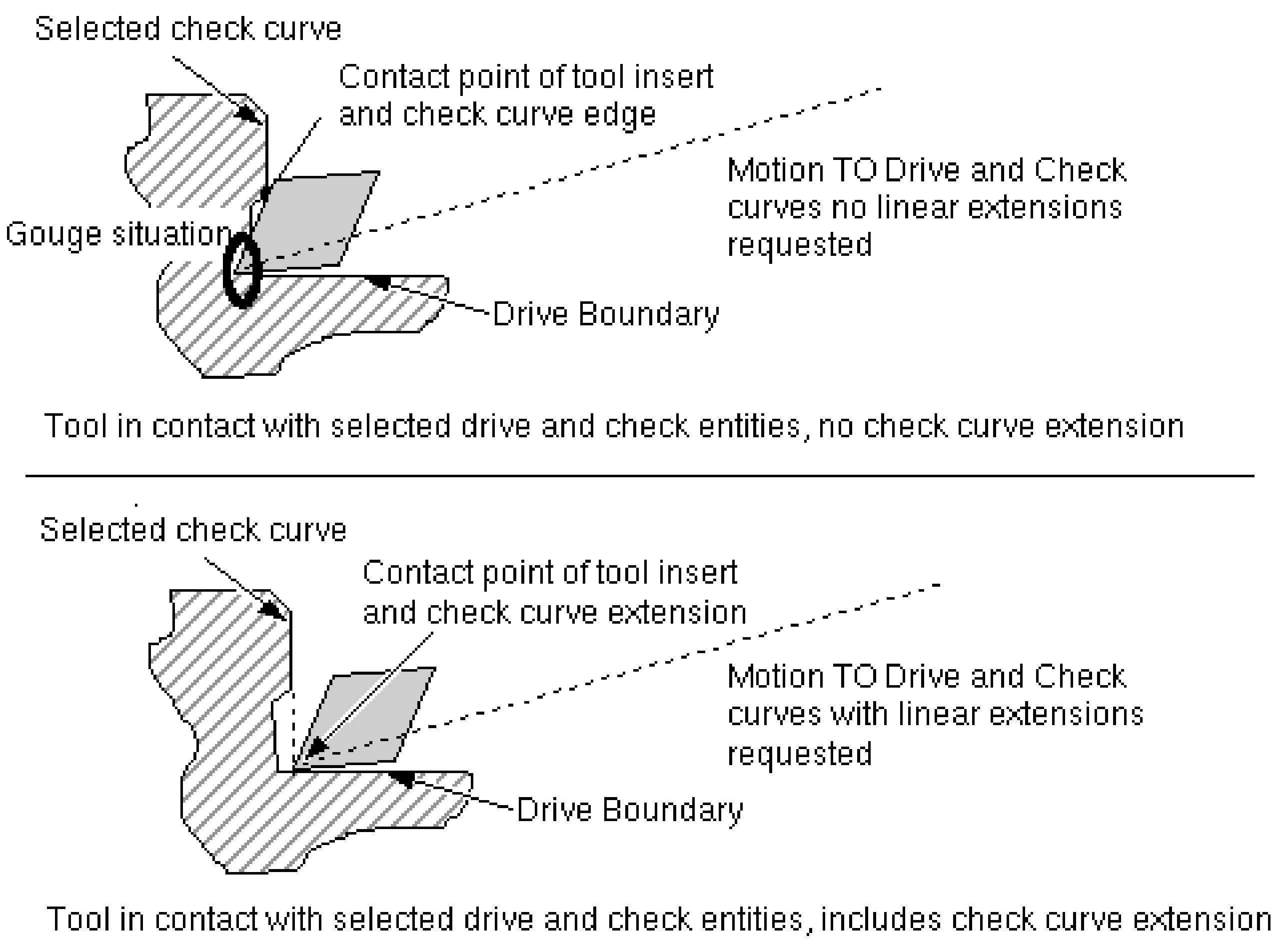


You can use the Entity Motion option on the Task Selector to add an entity based motion task to the current operation.

Entity based motion is distinct from location based motion, and requires the shape of the current insert for positioning of the tool.

Please note: Motion to the defined location is in a straight line from the current tool location.

Figure 3-15

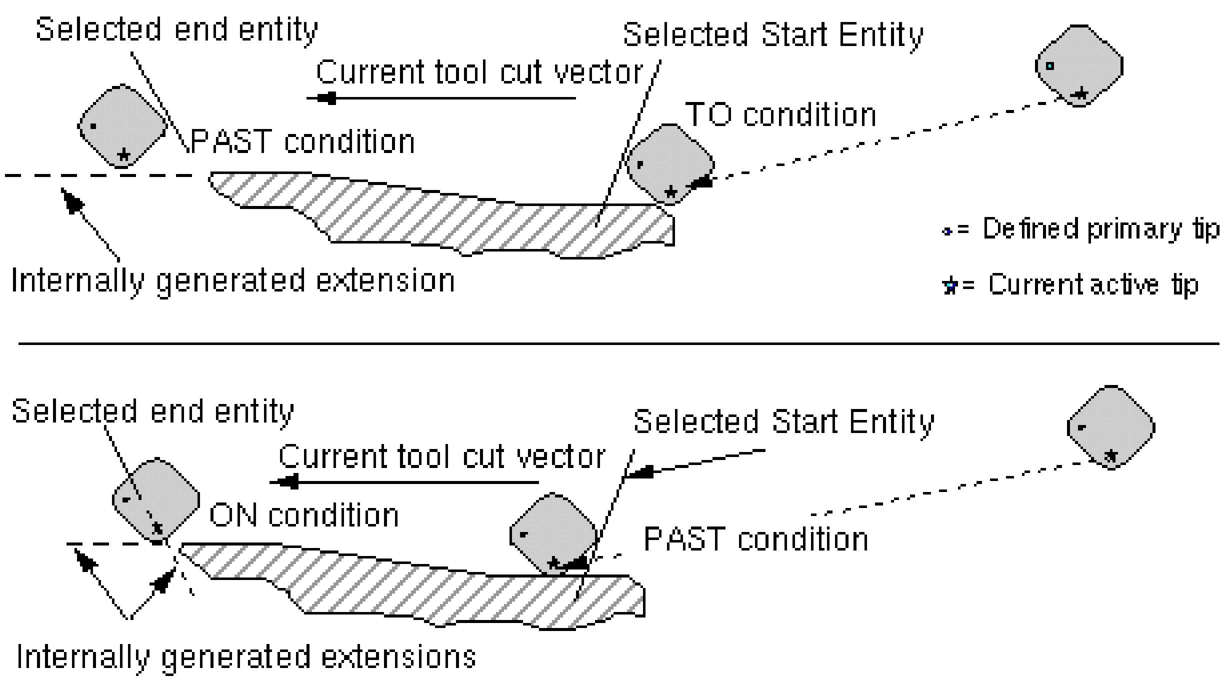


The current CVNC-T2 concept of TO, ON, and PAST is maintained when controlling the insert shape with respect to the geometry. TO and PAST are defined by reference to the current tool cutting vector associated with the current tool assembly, and not the current tool location. This vector is then used to determine the TO/PAST condition for the selected entity.

In cases where the vector and the selected entity are mathematically parallel, a rigid Right Hand Side rule is used. According to this rule, if you are looking along the entity in the direction of the cutting vector, the Right Hand Side of the entity is considered the TO side, while the Left Hand Side is the PAST side. This uses the complete insert shape for TO and PAST, and not the current tip. ON always places the center of the current active tool tip on the selected geometry.

The following illustration shows examples of TO, ON, and PAST for a check boundary, with the part boundary assuming a TO condition.

Figure 3-16 Example of TO, ON, and PAST for a Check Boundary



Discrete motion with respect to entities is also possible when you provide a single entity and then provide additional motion controls necessary to achieve the required motion result to the selected entity, for example NORMAL, DIRLIN etc.

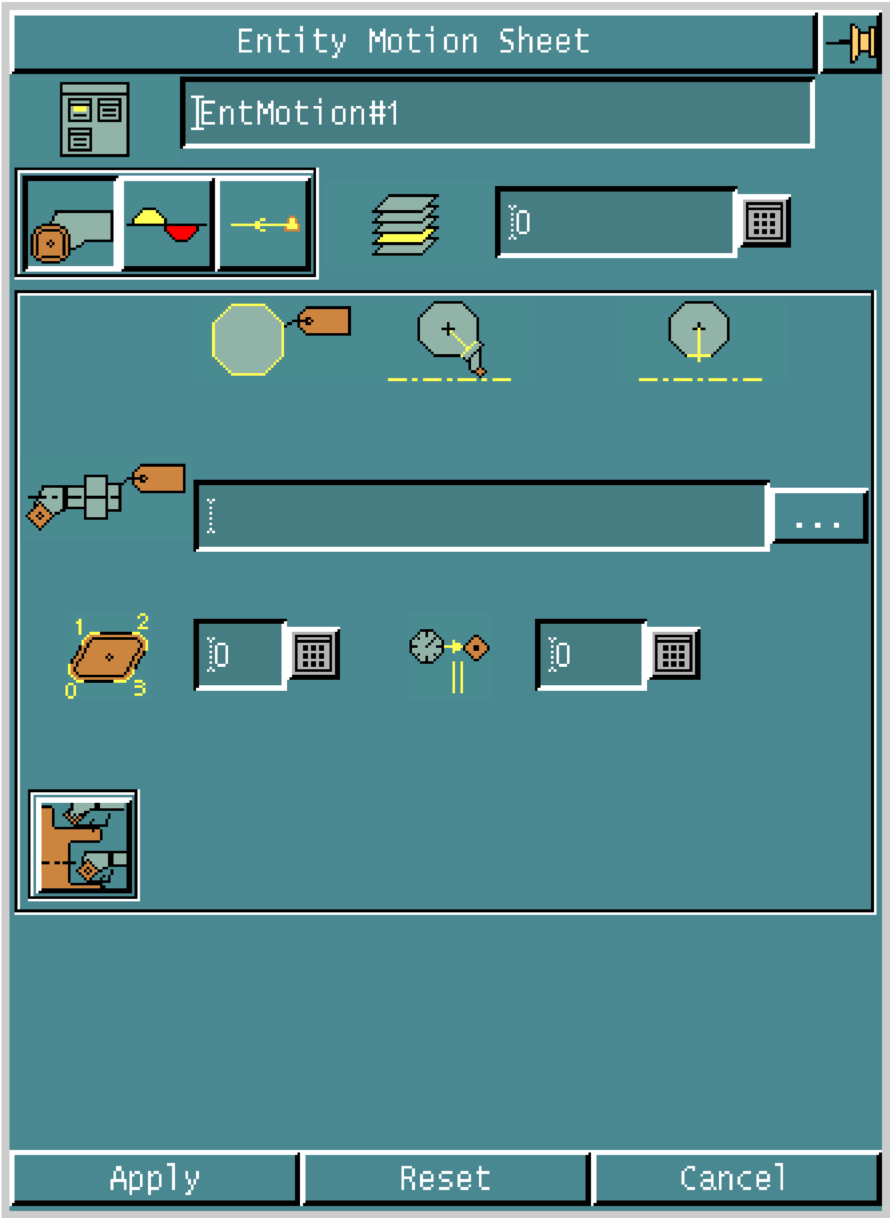
If you select a single entity for motion TO, ON, or PAST control modifiers, then this entity is considered the DRIVE curve for global stock purposes. This condition applies when considering either optional extension of part boundary with check curves, or global stock application.

Discrete motion functionality fully supports the use of Stock since it is essential for discrete stock applied to entities. You can apply different discrete stock values to both faces and diameters. However, if the current global stock values differ from the discrete stock value applied to the current entity by more than the current machining tolerance, CVNC issues a warning indicating this possible stock requirement conflict.

Using this Option

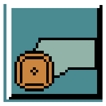
1. Highlight the task from the list of tasks on the Operation sheet, and choose the Edit option to launch the Entity Motion Sheet.

Figure 3-17 Entity Motion Sheet



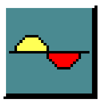
2. Specify the name of the task in the Task Name field.
3. Specify the layer on which the tool path will be placed in the Layer field.

Options on the Entity Motion Sheet



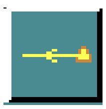
SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 3-29 for details.



SPECIFYING TOLERANCE PARAMETERS
Displays a property sheet that enables you to specify tolerance, and stock parameters for the current task.

See “Specifying Tolerance Parameters” on page 3-32 for details.

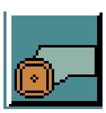


SPECIFYING GENERAL MACHINING PARAMETERS
Displays a property sheet that enables you to specify general machining parameters for the current task.

See “Specifying General Machining Parameters” on page 3-34 for details.

Please note: For each of these options, a different set of parameters appears on the same Entity Motion sheet.

Specifying Tool Assembly Selection Parameters

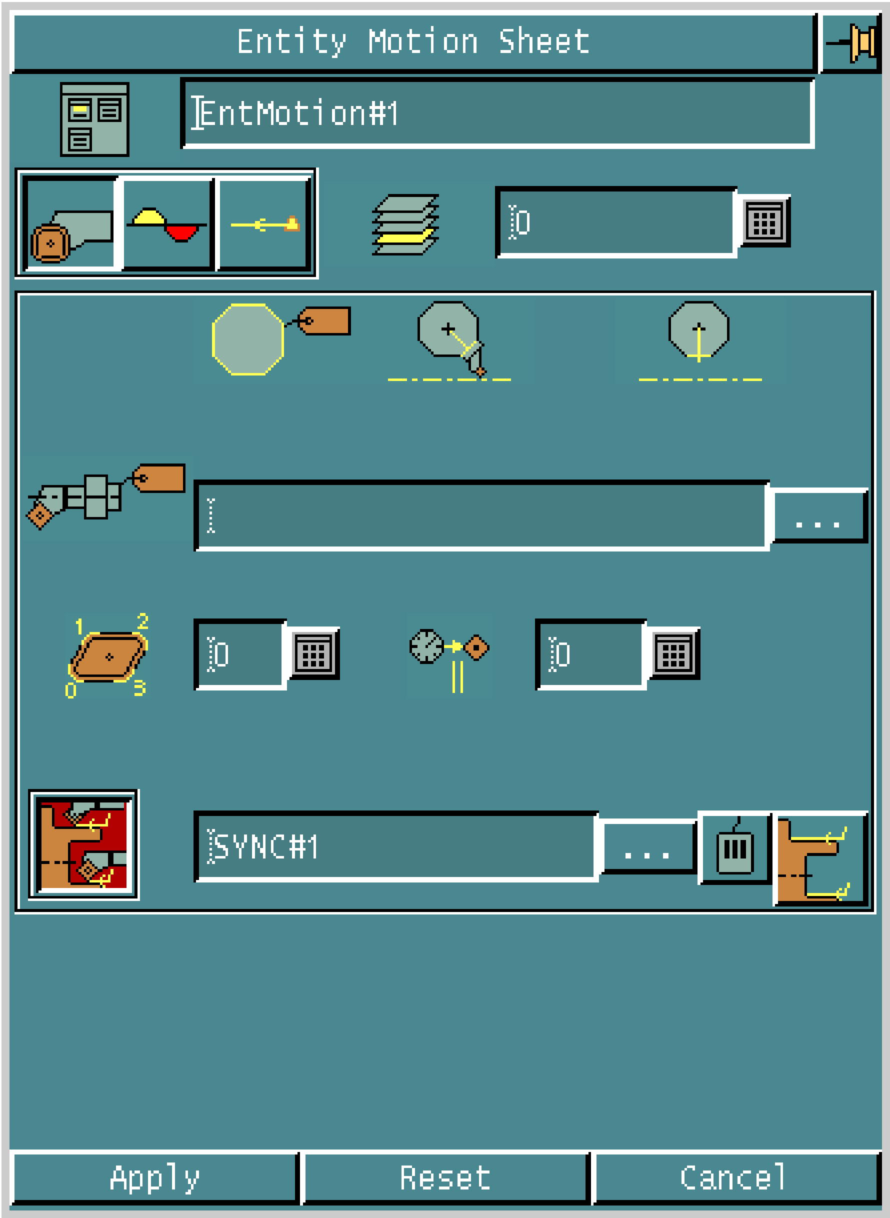


You can use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

Using this Option

1. Choose the Tool Parameters option from the Entity Motion sheet. The set of parameters for the tool assembly appears on the sheet.

Figure 3-18 Entity Motion Sheet with Tool Assembly Selection Parameters

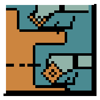


2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field.

   OR

   Choose a tool assembly from the Tool Selector scroll list.

3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



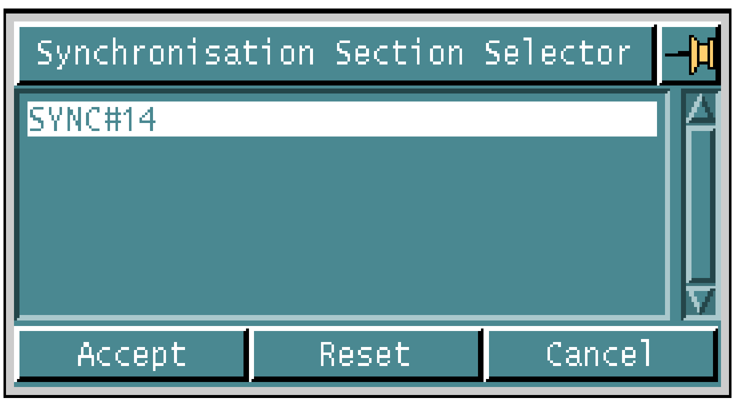
5. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

6. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 3-19 Synchronization Section Selector Scroll List

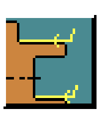


- Choose a synchronization section from the list.
- Click Accept.

OR

You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

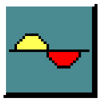
OR



Choose the New Sync Section option to create a new synchronization section to add the task.

7. Click Apply if you have finished entering all parameters for the task.

Specifying Tolerance Parameters



You can use the TOLERANCE PARAMETERS option to specify tolerance, and stock parameters for the current task.

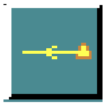
Using this Option

1. Choose the Tolerance Parameters option from the Entity Motion sheet. The set of parameters for the tool assembly appears on the sheet.

Figure 3-20 Entity Motion Sheet with Tolerance Parameters

2. Specify the inner tolerance for the current task in the Inner Tolerance field.
3. Specify the outer tolerance for the current task in the Outer Tolerance field.
4. Specify the stock offset for the current task in the Stock Offset field.
5. Click Apply if you have finished entering all parameters for the task.

Specifying General Machining Parameters

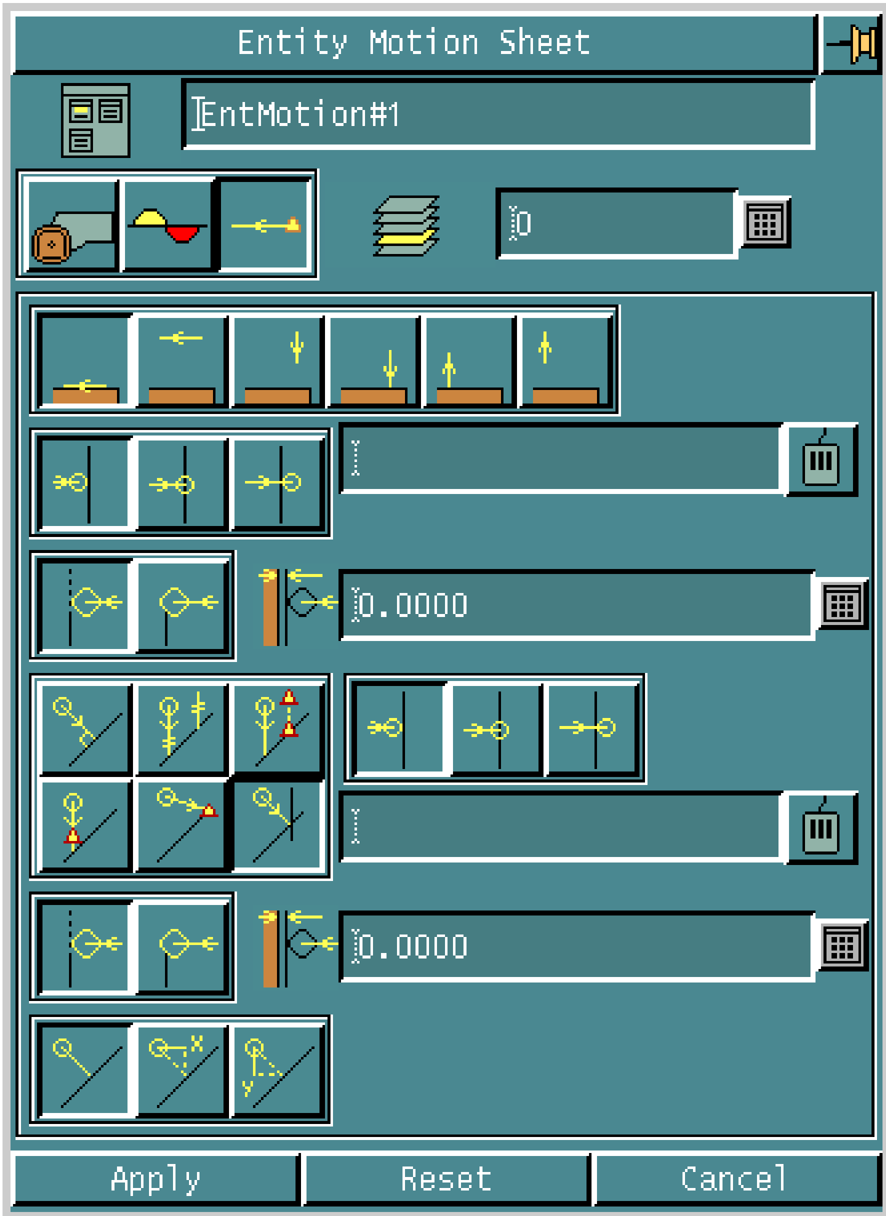


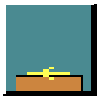
You can use the MACHINING PARAMETERS option to specify general machining parameters for the current task.

Using this Option

1. Choose the Machining Parameters option from the Entity Motion sheet. The set of general machining parameters appears on the sheet.

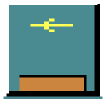
Figure 3-21 Entity Motion Sheet with General Machining Parameters





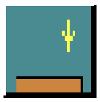
2. Click on the Cut Feed Motion option to specify that motion is to be made at cut feed rate.

OR



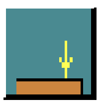
Click on the Rapid Feed Motion option to specify that motion is to be made at rapid feed rate.

OR



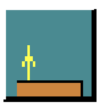
Click on the Approach Feed Motion option to specify that motion is to be made at approach feed rate.

OR



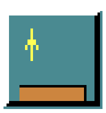
Click on the Plunge Feed Motion option to specify that motion is to be made at plunge feed rate.

OR

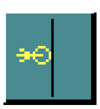


Click on the Retract Feed Motion option to specify that motion is to be made at retract feed rate.

OR

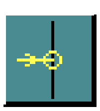


Click on the Clear Feed Motion option to specify that motion is to be made at clear feed rate.



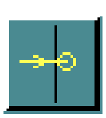
3. Click on the TO Entity option to specify motion will be TO the entity based on the drive vector.

OR



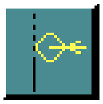
Click on the ON Entity option to specify motion will be ON the entity.

OR



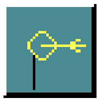
Click on the PAST Entity option to specify motion will be PAST the entity based on the drive vector.

4. Specify the entity in the Drive Entity field by digitizing the entity.



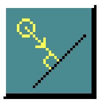
5. Click on the Extend ON option to specify that the entity will be extended.

   OR



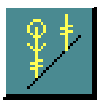
   Click on the Extend OFF option to specify that the entity will not be extended.

6. Specify an offset to the entity for this motion in the Entity Offset field.



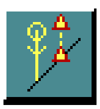
7. Click on the Entity Normal option to specify that direction of motion will be normal to the entity.

   OR



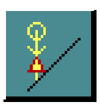
   Click on the Entity Parallel option to specify that direction of motion will be parallel to an entity.

   OR



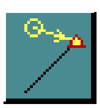
   Click on the Vector Parallel option to specify that direction of motion will be parallel to a vector.

   OR



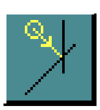
   Click on the Location Direction option to specify that motion will be in the direction of a location.

   OR



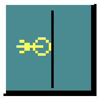
   Click on the Entity End option to specify that motion will be in the direction of the end of an entity.

   OR



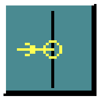
   Click on the Entity Intersection option to specify that motion will be in the direction of the intersection with an entity.

Note the following five options appear on the Entity motion Sheet only if you click on the Entity Intersection option.



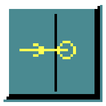
- Click on the TO Intersection Entity option to specify motion will be TO the intersection entity based on the drive vector.

  OR

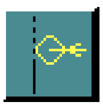


  Click on the ON Intersection Entity option to specify motion will be ON the intersection entity.

  OR

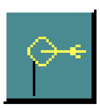


  Click on the PASTIntersection Entity option to specify motion will be PAST the intersection entity based on the drive vector.



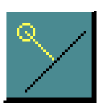
- Click on the Extend ON option to specify that the intersection entity will be extended.

  OR



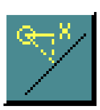
  Click on the Extend OFF option to specify that the intersection entity will not be extended.

  Specify an offset to the intersection entity for this motion in the Intersection Entity Offset field.



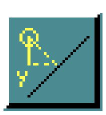
8. Click on the Complete Motion option to specify that the complete motion will be created

   OR



   Click on the XOnly option to specify that only the X component of the motion will be created.

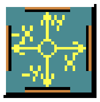
   OR



   Click on the YOnly option to specify that only the Y component of the motion will be created.

9. Click Apply if you have finished entering all parameters for the task.

Specifying XY-Direction Entity Based Motion Parameters

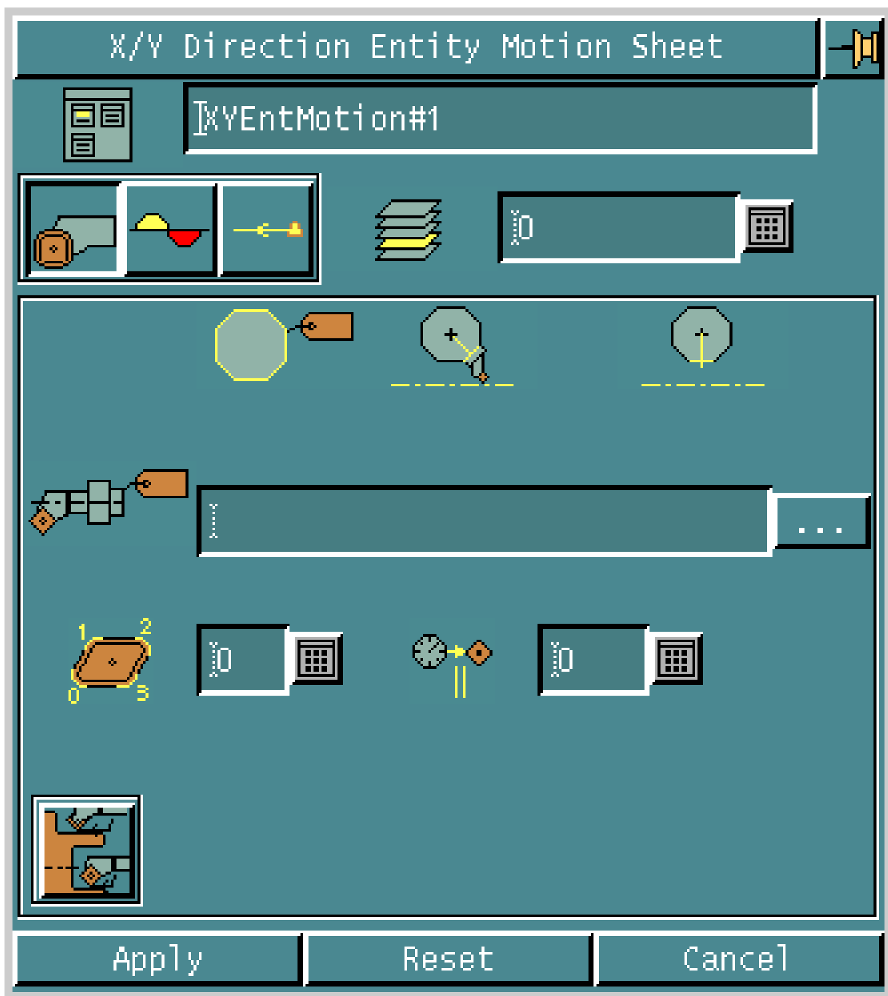


You can use the XYDirection Entity based Motion option on the Task Selector to add an XY-direction entity based motion task to the current operation.

Using this Option

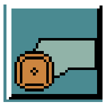
1. Highlight the task from the list of tasks on the Operation sheet, and choose the Edit option to launch the X/Y Direction Entity Motion Sheet.

Figure 3-22 X/Y Direction Entity Motion Sheet



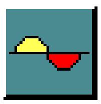
2. Specify the name of the task in the Task Name field.
3. Specify the layer on which the tool path will be placed in the Layer field.

Options on the X/Y Direction Entity Motion Sheet



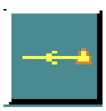
SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See "Specifying Tool Assembly Selection Parameters" on page 3-40 for details.



SPECIFYING TOLERANCE PARAMETERS
Displays a property sheet that enables you to specify tolerance, and stock parameters for the current task.

See "Specifying Tolerance Parameters" on page 3-43 for details.

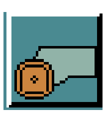


SPECIFYING GENERAL MACHINING PARAMETERS
Displays a property sheet that enables you to specify general machining parameters for the current task.

See "Specifying General Machining Parameters" on page 3-44 for details.

Please note: For each of these options, a different set of parameters appears on the same X/Y Direction Entity Motion sheet.

Specifying Tool Assembly Selection Parameters

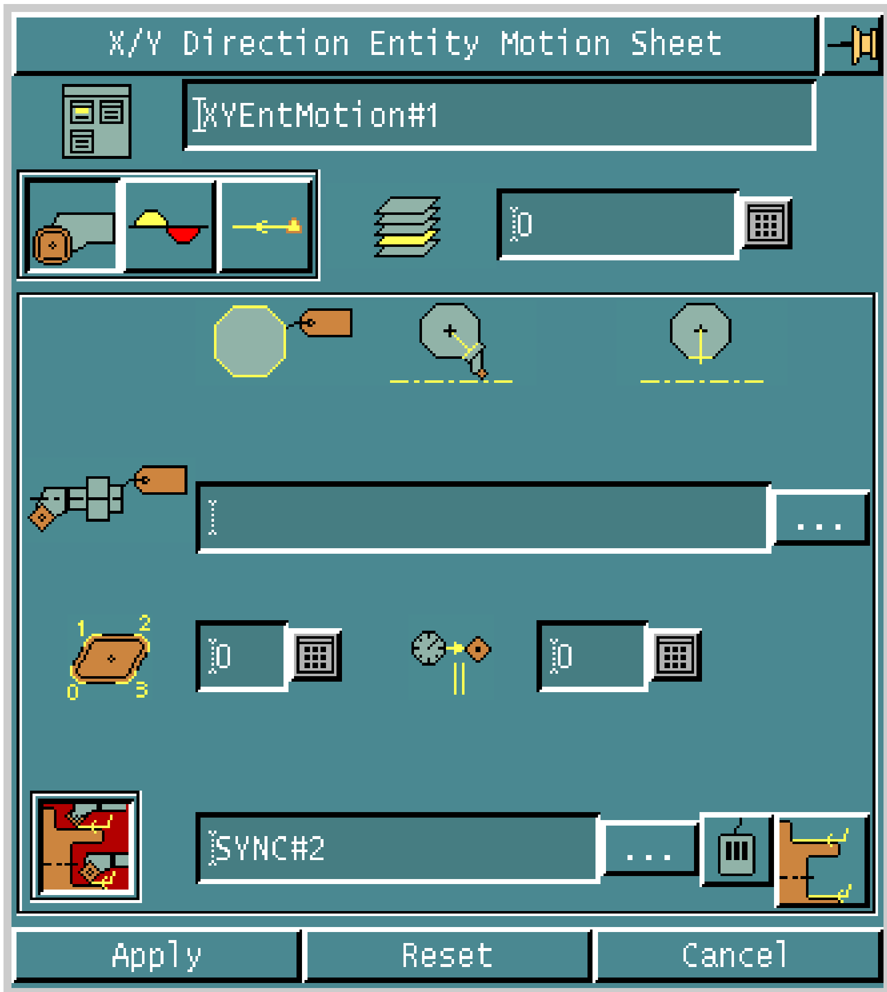


You can use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

Using this Option

1. Choose the Tool Parameters option from the X/Y Direction Entity Motion sheet. The set of parameters for the tool assembly appears on the sheet.

Figure 3-23 X/Y Direction Entity Motion Sheet with Tool Assembly Selection Parameters

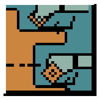


2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field.

   OR

   Choose a tool assembly from the Tool Selector scroll list.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.

4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



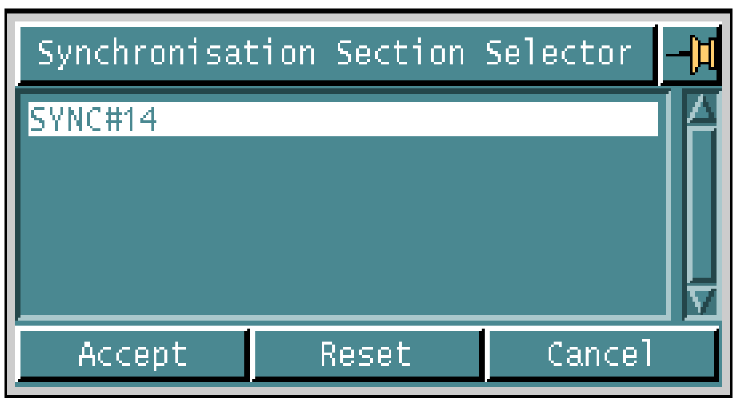
5. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

6. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 3-24 Synchronization Section Selector Scroll List

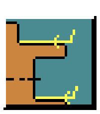


   - Choose a synchronization section from the list.
   - Click Accept.

   OR

   You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

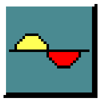
   OR



   Choose the New Sync Section option to create a new synchronization section to add the task.

7. Click Apply if you have finished entering all parameters for the task.

Specifying Tolerance Parameters

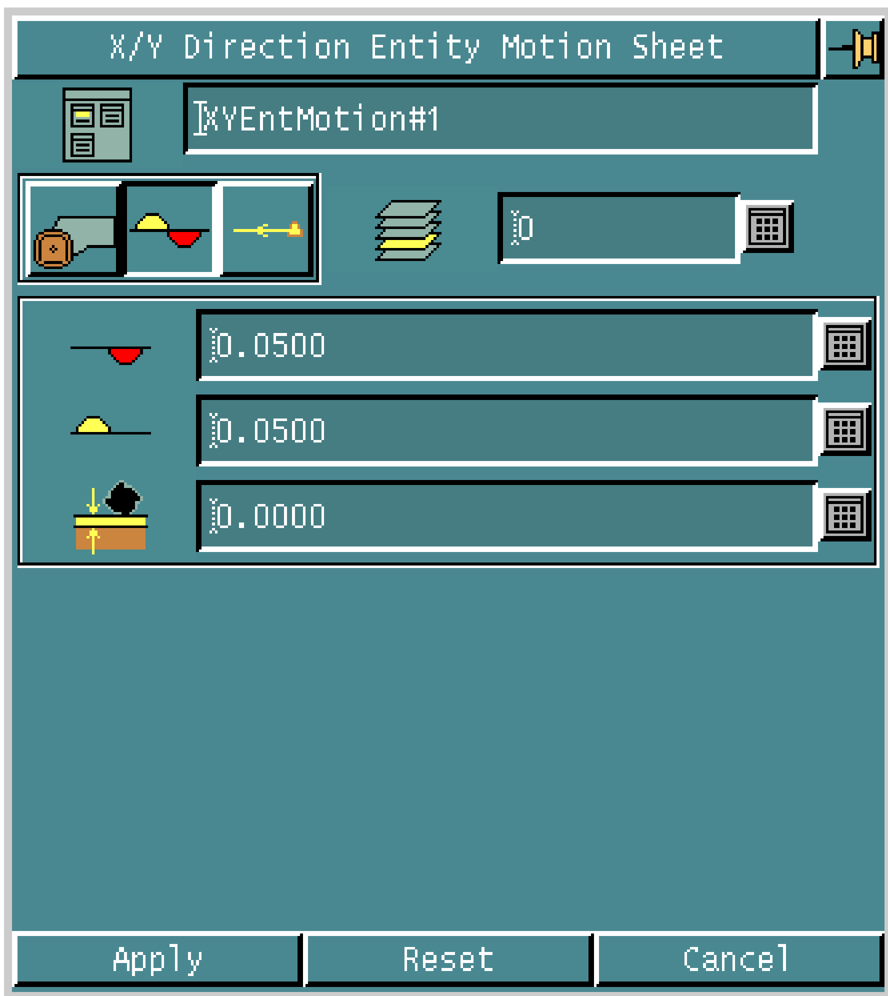


You can use the TOLERANCE PARAMETERS option to specify tolerance, and stock parameters for the current task.

Using this Option

1. Choose the Tolerance Parameters option from the X/Y Direction Entity Motion sheet. The set of parameters for the tool assembly appears on the sheet.

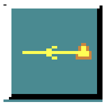
Figure 3-25 X/Y Direction Entity Motion Sheet with Tolerance Parameters



2. Specify the inner tolerance for the current task in the Inner Tolerance field.
3. Specify the outer tolerance for the current task in the Outer Tolerance field.
4. Specify the stock offset for the current task in the Stock Offset field.

5. Click Apply if you have finished entering all parameters for the task.

Specifying General Machining Parameters

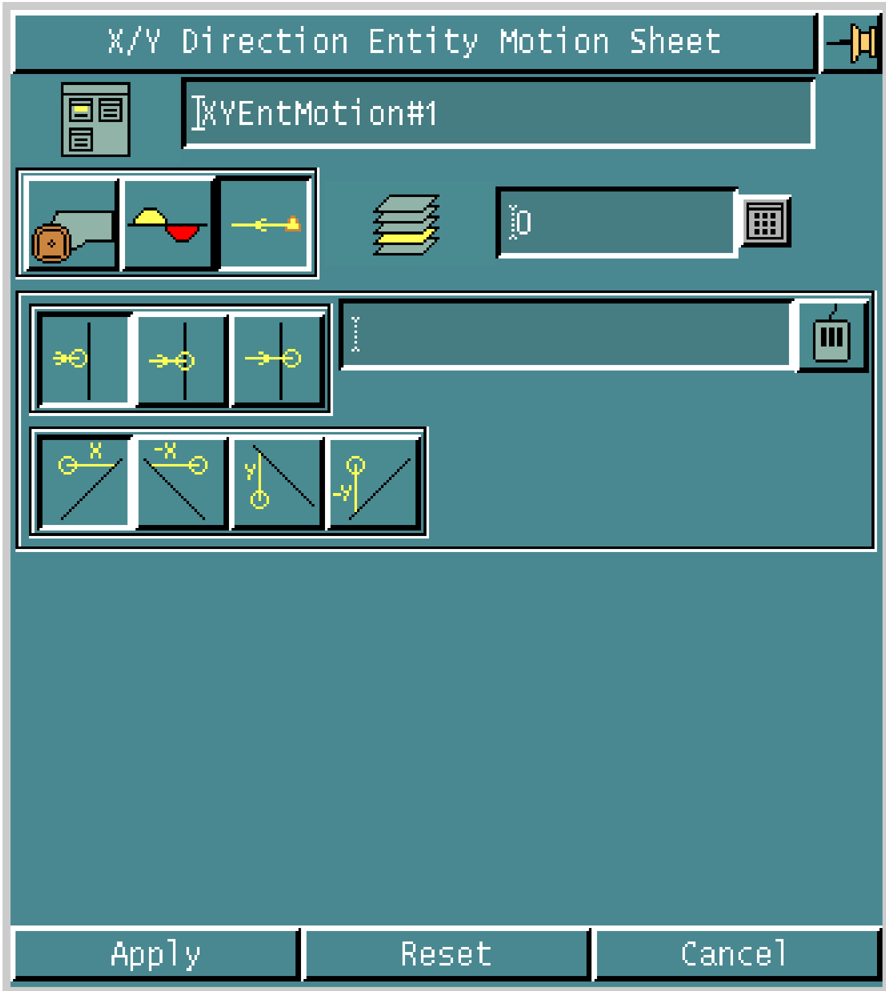


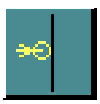
You can use the MACHINING PARAMETERS option to specify general machining parameters for the current task.

Using this Option

1. Choose the Machining Parameters option from the X/Y Direction Entity Motion sheet. The set of general machining parameters appears on the sheet.

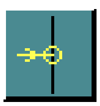
Figure 3-26 X/Y Direction Entity Motion Sheet with General Machining Parameters





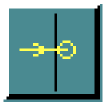
2. Click on the TO Entity option to specify motion will be TO the entity based on the drive vector.

   OR



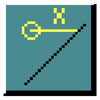
   Click on the ON Entity option to specify motion will be ON the entity.

OR



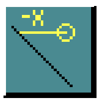
Click on the PAST Entity option to specify motion will be PAST the entity based on the drive vector.

3. Specify the entity in the Drive Entity field.



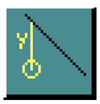
4. Click on the XDirection option to specify that the motion will be in the positive X direction.

OR



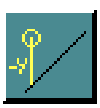
Click on the YDirection option to specify that the motion will be in the positive Y direction.

OR



Click on the NXDirection option to specify that the motion will be in the negative X direction.

OR



Click on the NYDirection option to specify that the motion will be in the negative Y direction.

5. Click Apply if you have finished entering all parameters for the task.

Executing the Job

You have to execute the task after specifying parameters for it. After you have clicked Apply on the appropriate sheet, use the following steps for executing your job:

1. Click Apply on the Operation sheet after specifying all the tasks for the current operation.
2. If you have used synchronization sections, you can see the Sync Master option in the NCBuilder sheet.

Figure 3-27 NCBuilder Sheet to Choose the Master Turret

- Choose this option to select the master turret. The Synchronization Master Sheet appears.

Figure 3-28 Synchronization Master Sheet

- Choose the master turret for each synchronization section from the option list. For each synchronization section, CVNC uses feed rates and spindle speeds of the master turret.
- Click Apply.

3. Click Apply on the NCBuilder sheet after specifying all the operations for the current job. The following pulldown menu appears:

Figure 3-29 Menu for Execution of a Job

4. Click Create Job and process.

Please note: If you have used synchronization sections, CVNC sorts the job control file (JCF) when you apply the job, such that there is minimum lead between turrets.

Chapter 4 Profiling Tasks

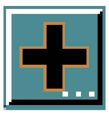
This chapter describes in detail about how you can use the NC Builder Graphical User Interface (GUI) to add a profiling task to a current operation. The following sections discuss the options available to you while defining a profile task:

- Accessing the Profiling Option
- Specifying the Profiling Parameters
- Specifying Tool Assembly Selection Parameters
- Specifying Feed Rates
- Specifying Spindle Speed
- Specifying Tolerance Parameters
- Specifying General Machining Parameters
- Specifying Leadin/Leadout Parameters
- Executing the Job

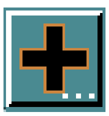
Accessing the Profiling Option

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.

4. Choose the Job Block option from the NCBuilder task set. The NCBuilder property sheet appears.

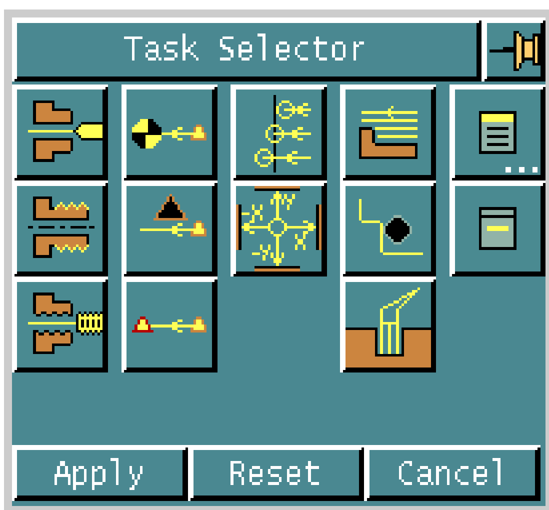


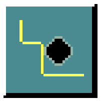
5. Choose the Add Operation option from the NCBuilder property sheet. The Operation Sheet appears.



6. Choose the Add Task option from the operation sheet. The Task Selector palette appears.

Figure 4-1 Task Selector





7. Choose the Profile option from the Task Selector palette. The task name and its icon appear in the Task Selector palette. The sequence number of the task also appears. For example, if the task is the second profile task in the operation, it has the number 2.

   You can use the Profile option on the Task Selector to add a profiling task to the current operation.

8. Click Apply.

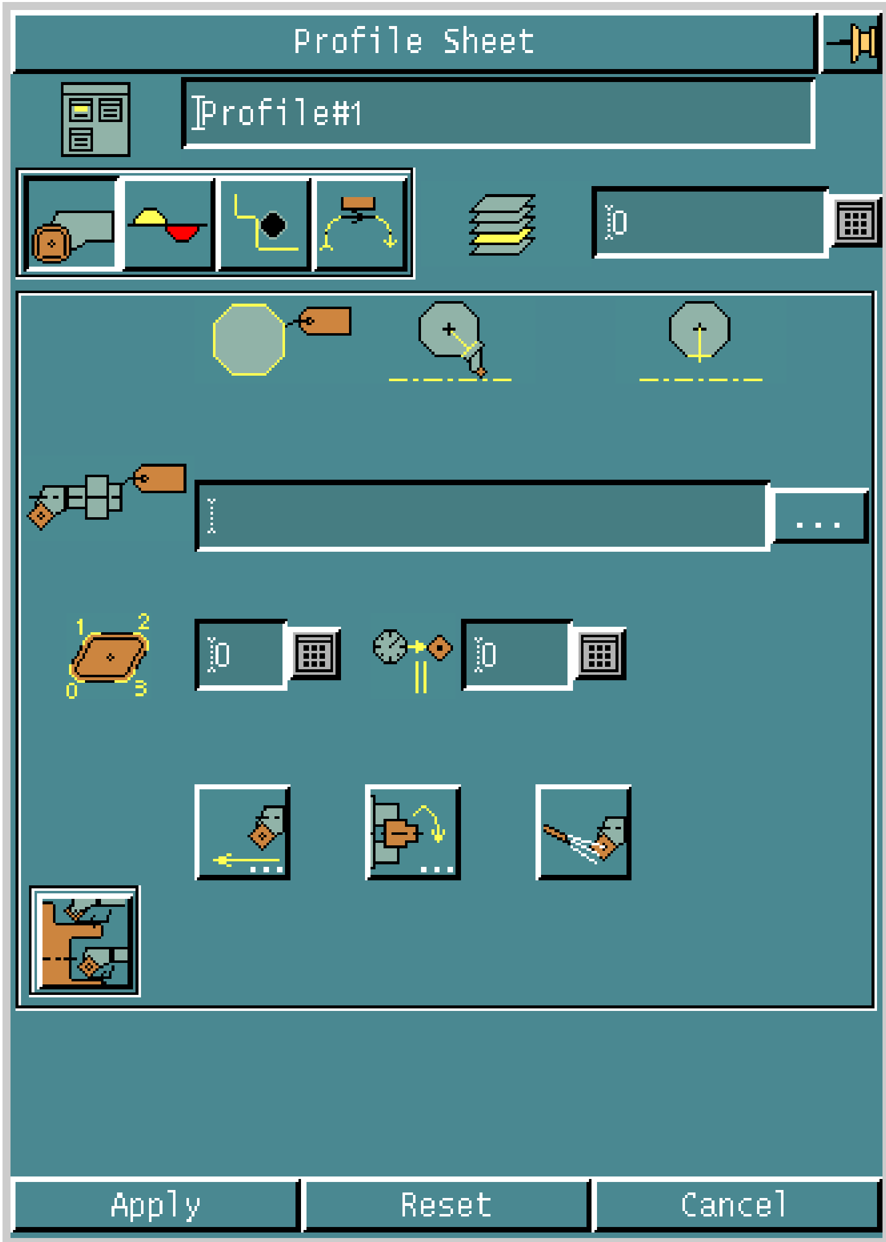
Specifying the Profiling Parameters

Use the EDIT option on the Operation sheet to specify parameters for the profiling process.

Using this Option

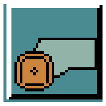
1. Highlight the Profile task from the list of tasks on the Operation sheet, and choose the Edit option to launch the Profile Sheet.

Figure 4-2 Profile Sheet



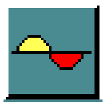
2. Specify the name of the task in the Task Name field.
3. Specify the layer for creating the tool path in the Layer field.

Options on the Profile Sheet



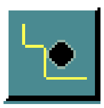
SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See "Specifying Tool Assembly Selection Parameters" on page 4-7 for details.



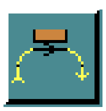
SPECIFYING TOLERANCE PARAMETERS
Displays a property sheet that enables you to specify tolerance, and stock parameters for the current task.

See "Specifying Tolerance Parameters" on page 4-14 for details.



SPECIFYING GENERAL MACHINING PARAMETERS
Displays a property sheet that enables you to specify general machining parameters for the current task.

See "Specifying General Machining Parameters" on page 4-17 for details.

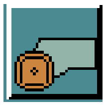


SPECIFYING LEAD IN/LEAD OUT PARAMETERS
Displays a property sheet that enables you to specify lead in/lead out parameters for the current task.

See "Specifying Leadin/Leadout Parameters" on page 4-29 for details.

Please note: For each of these options, a different set of parameters appears on the same Profile sheet.

Specifying Tool Assembly Selection Parameters

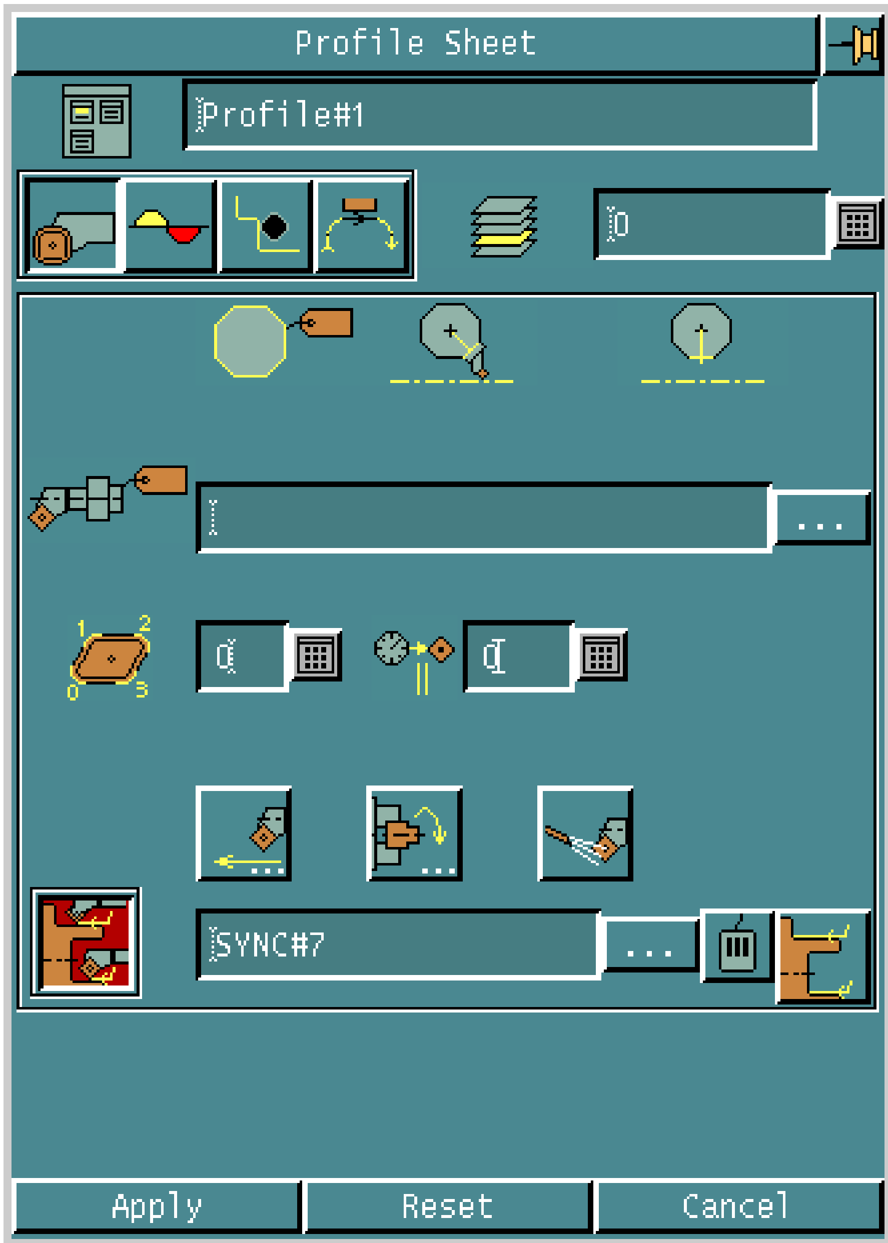


You can use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

Using this Option

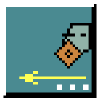
1. Choose the Tool Parameters option from the Profile sheet. The set of parameters for the tool assembly appears on the Profile sheet.

Figure 4-3 Profile Sheet with Tool Assembly Selection Parameters

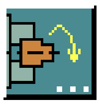


2. Choose a tool assembly from the Tool Selector scroll list.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.

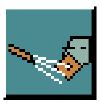
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



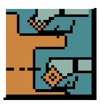
5. Choose the Feed Parameters option to launch the Feeds Sheet. Enter the feed rates for the current task. See “Specifying Feed Rates” on page 4-11 for details.



6. Choose the Spindle Speed Parameters option to launch the Spindle Speed Sheet. Enter the spindle speed parameters for the current task. See “Specifying Spindle Speed” on page 4-13 for details.



7. Choose the Coolant option and specify the coolant setting to either ON, OFF, MIST, or FLOOD.



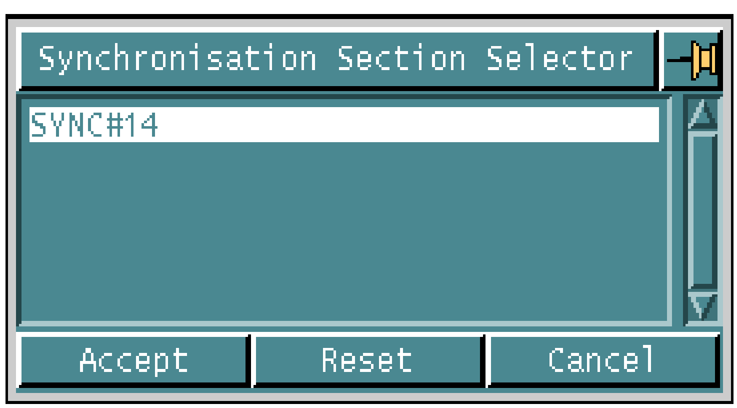
8. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

9. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 4-4 Synchronization Section Selector Scroll List

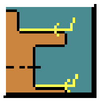


- Choose a synchronization section from the list.
- Click Accept.

OR

You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

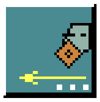
OR



Choose the New Sync Section option to create a new synchronization section to add the task.

10. Click Apply if you have finished entering all parameters for the task.

Specifying Feed Rates

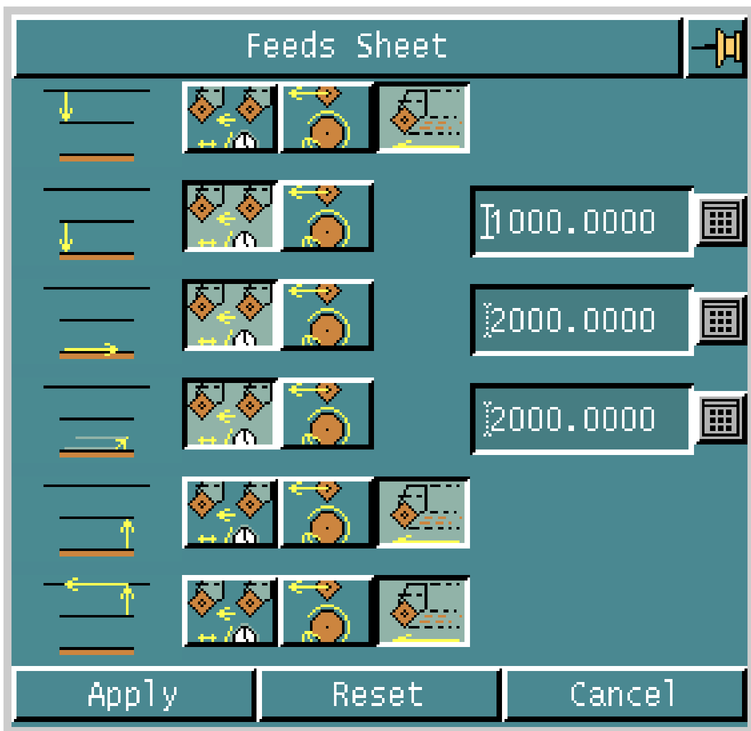


You can use the Feed Parameters option to specify the feed rates for the current operation.

Using this Option

1. Choose the Feed Parameters option from the Profile sheet as shown in Figure 4-3. The Feeds Sheet appears.

Figure 4-5 Feeds Sheet



2. Click and enter the approach feed rate. You can specify it in units of mm/min or mm/rev.

   OR

   Specify it to be the same as the rapid feed rate.

3. Click and enter the plunge feed rate. You can specify it in units of mm/min or mm/rev.
4. Click and enter the cut feed rate. You can specify it in units of mm/min or mm/rev.

5. Click and enter the connect feed rate. You can specify it in units of mm/min or mm/rev.
6. Click and enter the retract feed rate. You can specify it in units of mm/min or mm/rev.

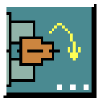
   OR

   Specify it to be the same as the rapid feed rate.
7. Click and enter the clear feed rate. You can specify it in units of mm/min or mm/rev.

   OR

   Specify it to be the same as the rapid feed rate.
8. Click Apply.

Specifying Spindle Speed



You can use the Spindle Speed Parameters option to specify the spindle speed parameters for the current operation.

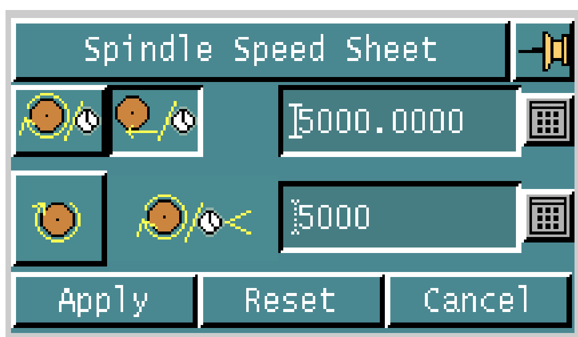
You can specify separate spindle speeds for the following:

- The general cutting motion.
- The optional final profile cut.

Using this Option

1. Choose the Spindle Speed Parameters option from the Profile sheet as shown in Figure 4-3. The Spindle Speed Sheet appears.

Figure 4-6 Spindle Speed Sheet

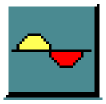


2. Click and enter the spindle speed in revolutions per minute.

   OR

   Click and enter the spindle speed in surface meters per minute. If you choose this option you must specify the maximum spindle speed in revolutions per minute.
3. Click and specify the direction of rotation of the spindle.
4. Click Apply.

Specifying Tolerance Parameters

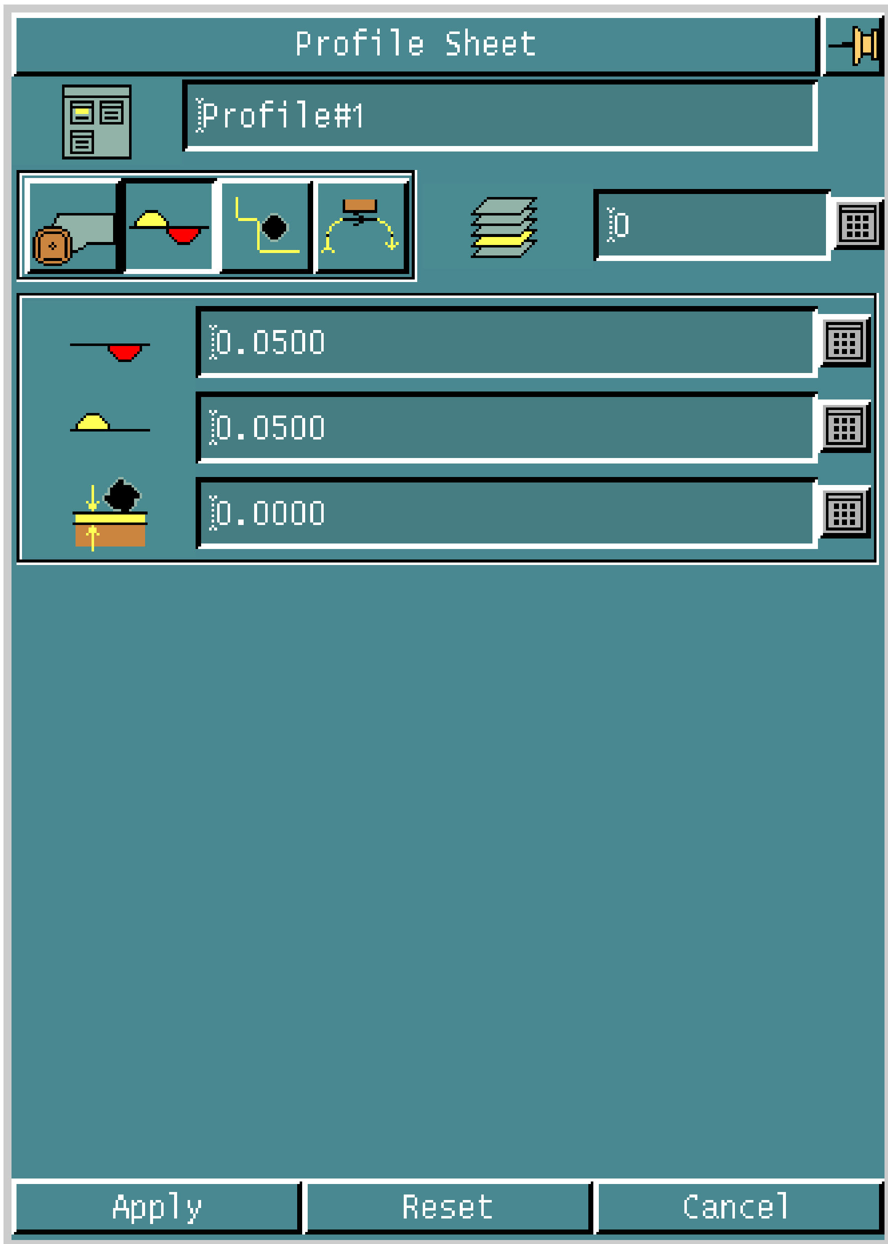


You can use the TOLERANCE PARAMETERS option to specify tolerance, and stock parameters for the current task.

Using this Option

1. Choose the Tolerance Parameters option from the Profile sheet. The set of parameters for tolerance appears on the sheet.

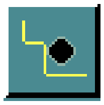
Figure 4-7 Profile Sheet with Tolerance Parameters



2. Specify the inner tolerance for the current task in the Inner Tolerance field.
3. Specify the outer tolerance for the current task in the Outer Tolerance field.

4. Specify the stock offset for the current task in the Stock Offset field.
5. Click Apply if you have finished entering all parameters for the task.

Specifying General Machining Parameters



You can use the MACHINING PARAMETERS option to specify general machining parameters for the current task.

Controlling Start and End Locations

To fully control machining operations, it is necessary to have full control over the start and end locations of the profile cutting motion. In all cases, the tool is positioned, wherever possible, with the current active tip tangential to the profile boundary at both the start, and end positions. The complete tool shape is checked against the profile boundary and the requested start or end positions, and provided start or end positions that are fully degouged.

The options for control are:

- Profiling the identified boundary completely, with no limiting of the profile operation. This is the default.
- You can identify locations on or near the selected geometry to identify the start or end locations. When a supplied location does not lie exactly on the geometry, then you can use a point that lies on the geometry nearest to the selected location in the current 2D plane.

  You can provide any of the following combinations:

  - A start and an end location.

    As shown in the following illustration, the cut will start with the active tool tip tangential to a location on the profile boundary nearest to the identified start location. The cut will end tangentially to a location on the profile boundary nearest to the identified end location.

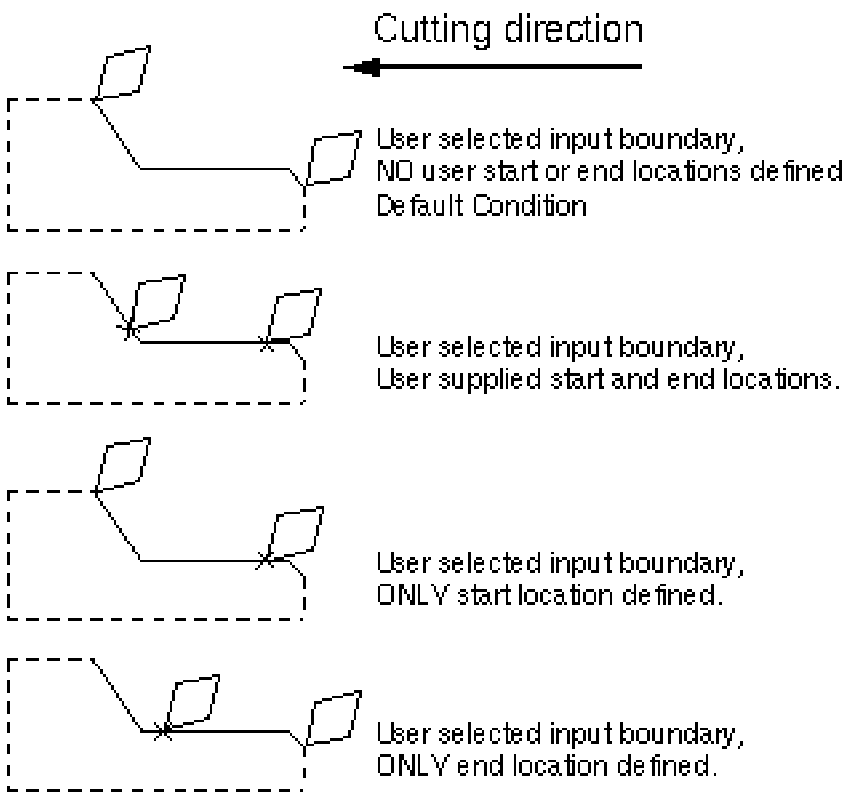
  - A start location without an end location.

    As shown in the following illustration, the profile cut will start with the active tool tip tangential to the identified location, and machine the remaining profile boundary geometry with respect to the cut direction.

  - An end location without a start location.

    A shown in the following illustration, the profile cut will start with the active tool tip tangential to the extreme of the supplied geometry and end tangential to the identified end location. Ambiguities, if any, are resolved with respect to the cut direction.

Figure 4-8 Profile Cut with Various Combinations of Start and End Locations



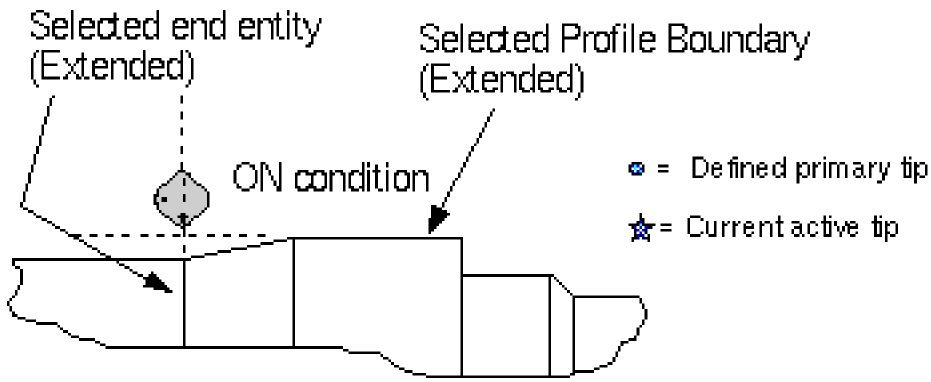
- You can identify any CADDS supported geometric wireframe entities to control the profile start and end conditions.

  The current concept of TO, PAST, and ON applies with respect to the identified control geometry, tool tip shape, and cut direction. In the case of TO and PAST, the entire tool tip insert shape is positioned against the supplied control geometry and part boundary with respect to the cut direction. When positioning the tool for TO and PAST, the tool is positioned relative to both actual non extended part boundary and check curve wherever possible.

  You have the option to extend the boundary and the check curve. In this way, you can better control the required tool position in case the supplied part boundary or check curve are of insufficient length to position the tool insert as required.

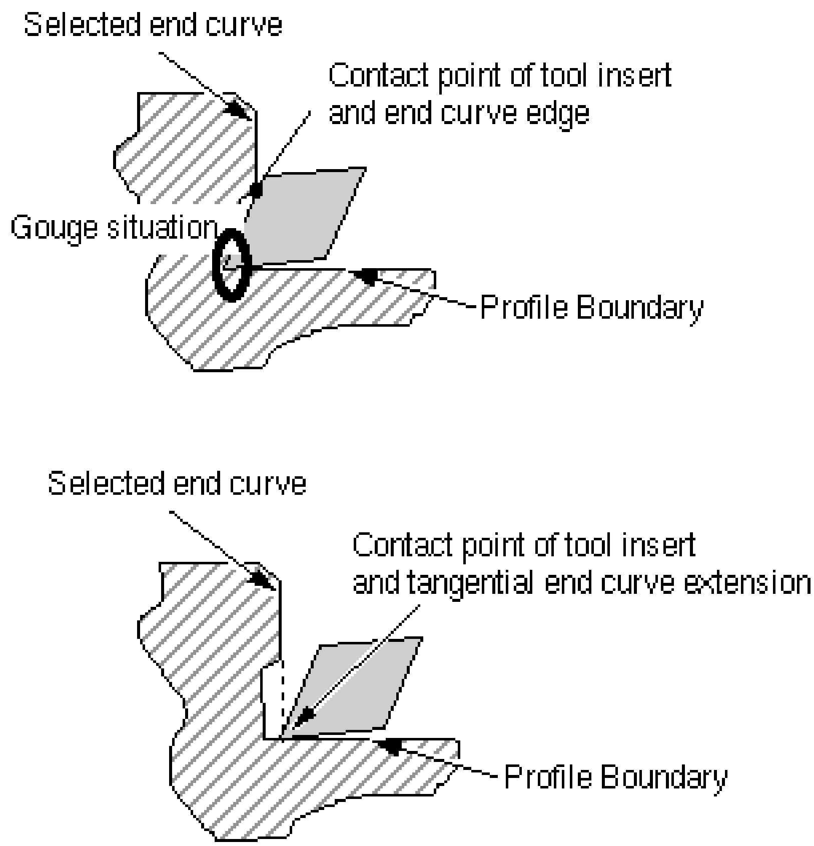
When you select ON, the center of the current active tool tip radius is used for positioning ON the check curve. As shown in the following illustration, the tool is positioned so that the center of the current active tip is positioned ON to the check curve or its linear extension generated by the profile functionality

Figure 4-9 Tool Tip Position with ON Condition



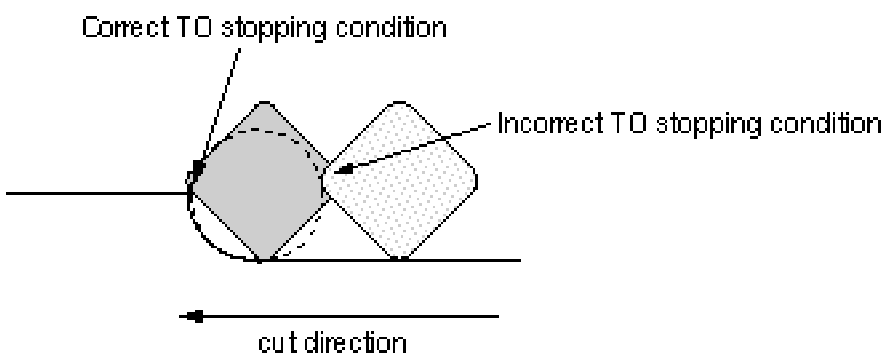
With respect to the TO and PAST concept, it is not necessary for the controlling geometry to physically intersect the selected profile boundary, or vice versa. By default, the complete tool shape is positioned relative to this geometry. This may produce gouge situations. You can prevent such situations by optionally providing internal boundary and control curve extensions, as shown in the following illustration.

Figure 4-10 Preventing Gouge Situations



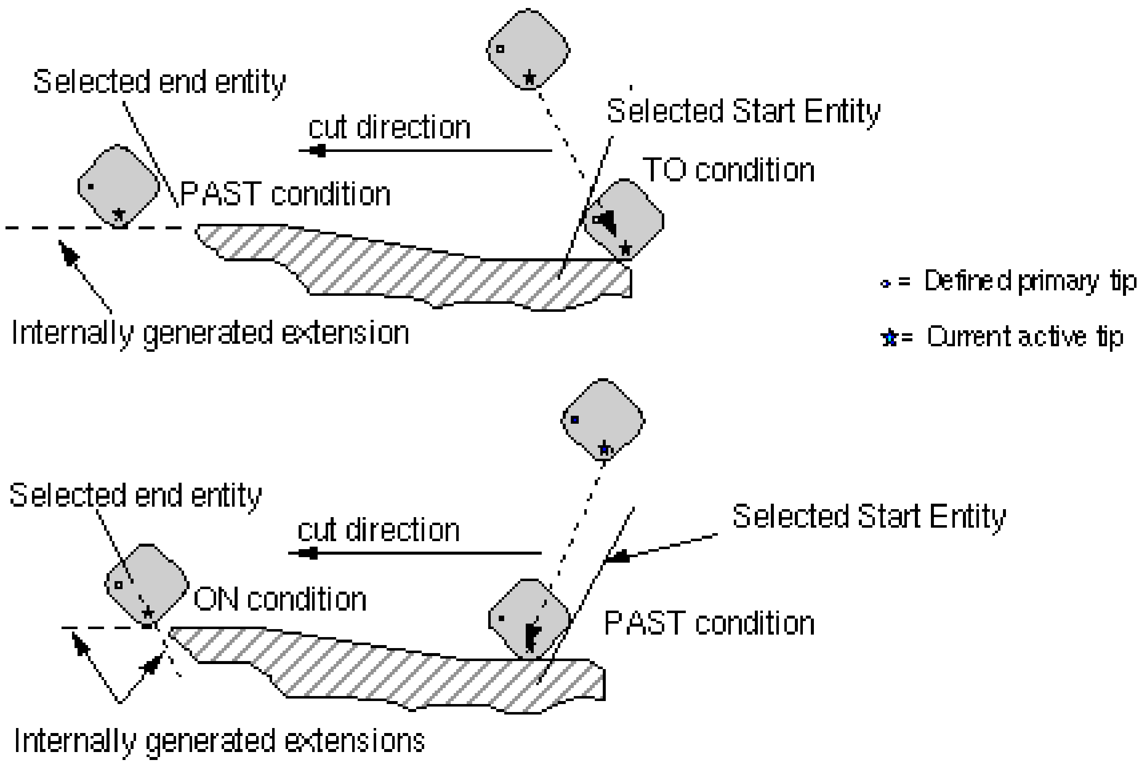
When arcs are used as the start or end control curves, then the system uses only the visible portion of the arc for correct positional testing, and not the whole circular form of the arc.

Figure 4-11 Arcs as Start and End Control Curves



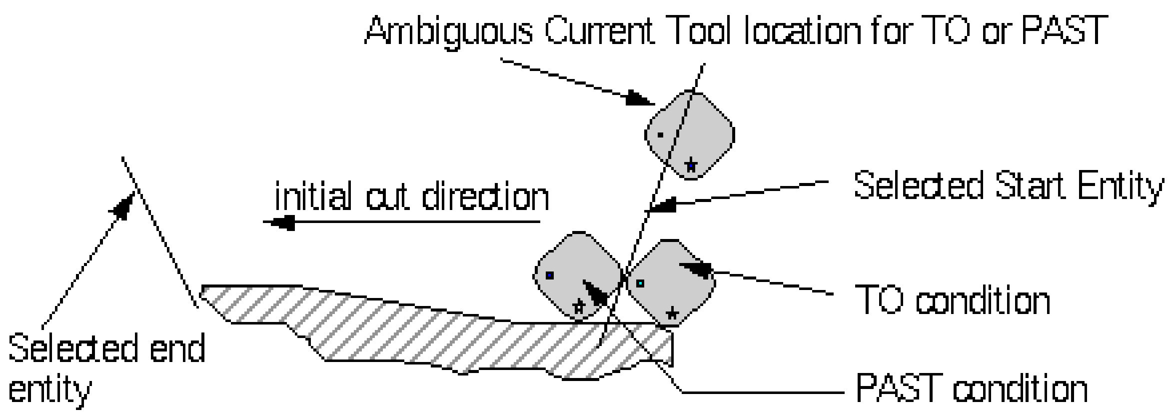
The following illustration shows the concept of TO and PAST conditions. Earlier, CVNC used the current tool location to identify TO and PAST conditions. This causes problems when you insert additional programming statements that change the location of the tool before the TO and PAST conditions are selected. To prevent such situations, the profile cutting direction is used to determine the TO and PAST conditions. Thus the tool location can be modified without adversely affecting the select TO and PAST conditions.

Figure 4-12 TO and PAST Conditions



As shown in the following illustration, ambiguous cases may exist with starting positions for both TO and PAST. In such cases, the ambiguity is resolved by evaluating the initial profile cutting motion direction and not the current tool location. This direction is then applied to define the TO and PAST conditions.

Figure 4-13 Ambiguous Tool Location for TO and PAST



You can also provide a combination of the two identified control formats, that is, a start or end location combined with a start or end geometric entity and a relevant TO, ON, or PAST orientation.

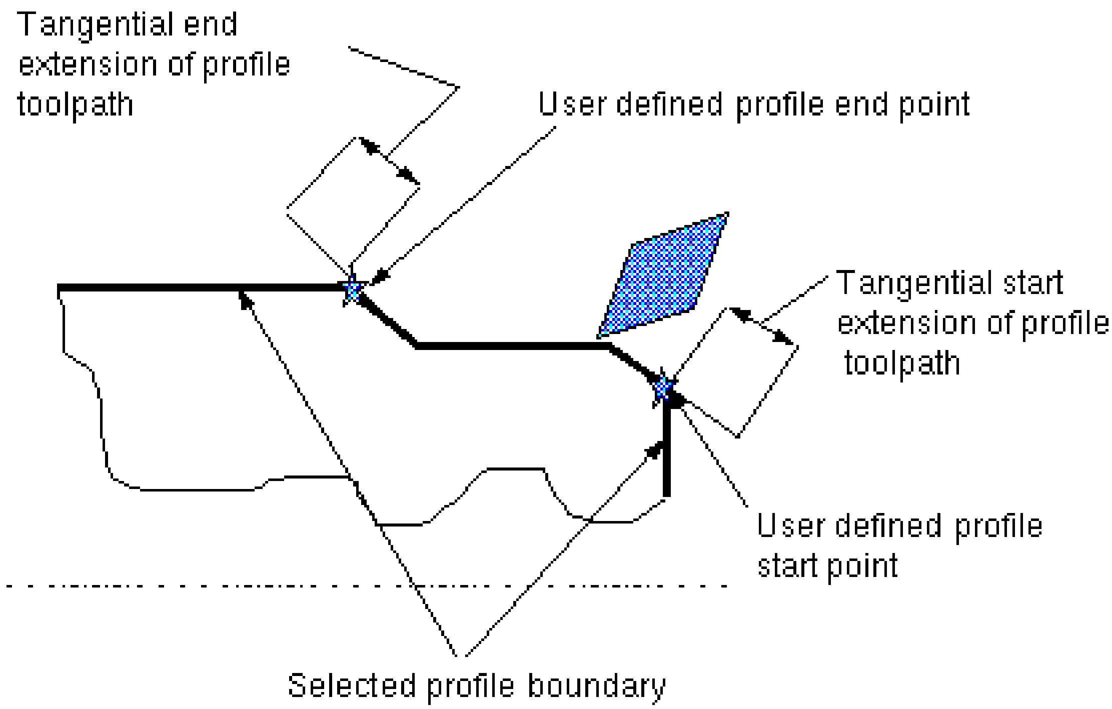
Start and End Extensions

You can apply extensions to the generated profile tool path. These extensions can be either:

- Linear extensions tangential to the tool path at the initial point for start extensions, and tangential to the tool path at their final point for end extensions.

  Output from tangential extensions are checked for redundant points, that is, when the linear extension adjoins a linear segment of the profile boundary, the output contains only a point for the start of the extension, and a point at the end of the linear boundary segment, as shown in the following illustration.

Figure 4-14 tangential Start and End Extensions

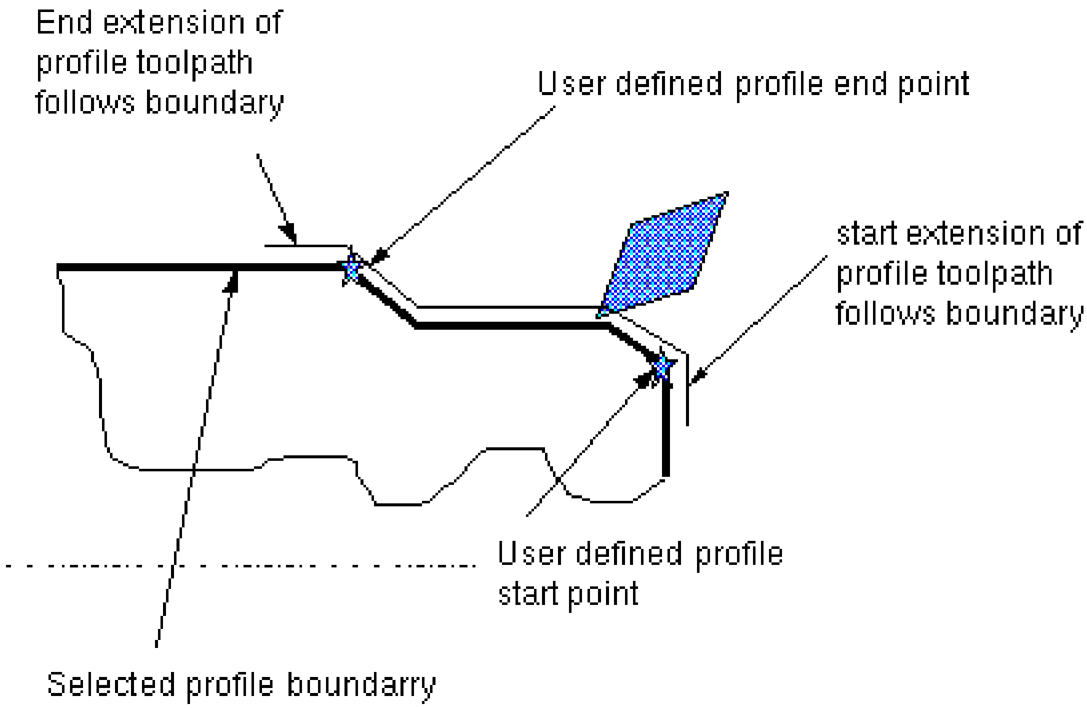


OR

- Extensions that follow the profile boundary, as specified by you.

  This is useful where the boundary is longer than the required section of the tool path, as defined by the start and end conditions. If the specified extension length is longer than the supplied boundary, the extension then reverts to a linear extension at the point where it falls off or onto the profile boundary.

Figure 4-15 Start and End Extensions Following the Profile Boundary

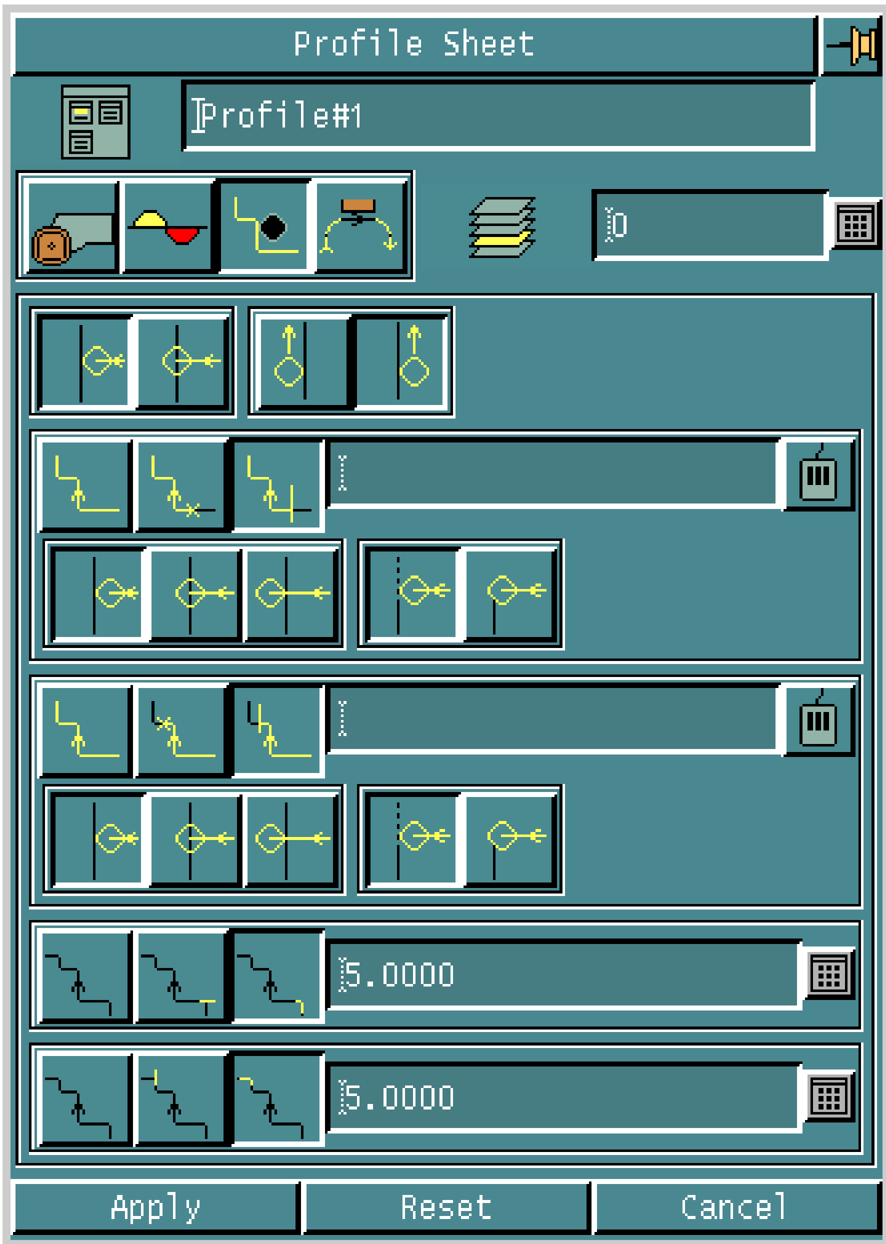


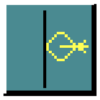
When you provide a TO, ON, or PAST for a check curve to define the start or end position, the extensions apply to the tool path from the point where the tool makes contact with both part boundary and the controlling check curve. The method of extension apply according to the above requirements.

Using this Option

1. Choose the Machining Parameters option from the Profile sheet. The set of general machining parameters appears on the sheet.

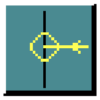
Figure 4-16



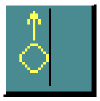


2. Click on the TO Entity option to specify that motion will be TO the entity.

   OR

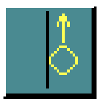


   Click on the ON Entity option to specify that motion will be ON the entity.

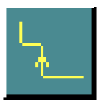


3. Click on the Left Boundary option to specify that the tool will machine to the left of the geometry, relative to the cut direction.

   OR

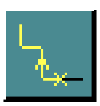


   Click on the Right Boundary option to specify that the tool will machine to the right of the geometry, relative to the cut direction.



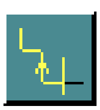
4. Click on the Start Control OFF option to specify that the open geometry is to be machined from the start. This is the default.

   OR



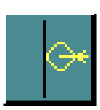
   Click on the Location Start Control option to specify that the open geometry is to be machined from a start location, and enter the location at which to start machining the geometry in the Start Location field.

   OR



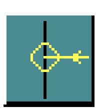
   Click on the Entity Start Control option to specify that the open geometry is to be machined from a start line, and specify the line at which to start machining the geometry in the Start Entity field.

   Note the following five options appear only if you choose the Entity Start Control option.



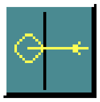
   - Click on the TO Entity option to specify that motion will be TO the entity based on the drive vector.

     OR

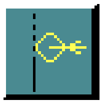


     Click on the ON Entity option to specify that motion will be ON the entity.

     OR

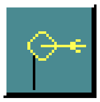


Click on the PAST Entity option to specify that motion will be PAST the entity based on the drive vector.



- Click on the Extend ON option to specify that the entity will be extended.

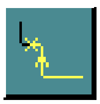
  OR



Click on the Extend OFF option to specify that the entity will not be extended.

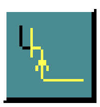
5. Click on the End Control OFF option to specify that the open geometry is to be machined to its end.

   OR



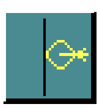
Click on the Location End Control option to specify that the open geometry is to be machined to an end location, and enter the location at which to end machining the geometry in the End Location field.

OR



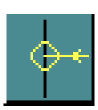
Click on the Entity End Control option to specify that the open geometry is to be machined to an end line, and specify the line at which to end machining the geometry in the End Entity field.

Note the following five options appear only if you choose the Entity End Control option.



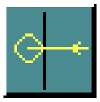
- Click on the TO Entity option to specify that motion will be TO the entity based on the drive vector.

  OR

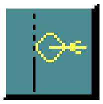


Click on the ON Entity option to specify that motion will be ON the entity.

OR

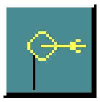


Click on the PAST Entity option to specify that motion will be PAST the entity based on the drive vector.



- Click on the Extend ON option to specify that the entity will be extended.

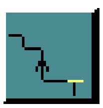
  OR



Click on the Extend OFF option to specify that the entity will not be extended.

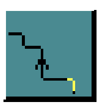
6. Click on the Start Extension Off option to specify no start extension.

   OR

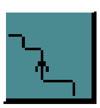


Click on the Linear Start Extension option to specify a tangential start extension, and enter the tool path's start extension for the current task in the Start Extension Length field.

OR

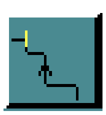


Click on the Profile Start Extension option to specify a start extension that follows the geometry, and enter the tool path's start extension for the current task in the Start Extension Length field.



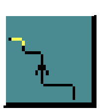
7. Click on the End Extension Off option to specify no end extension.

   OR



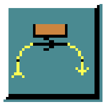
Click on the Linear End Extension option to specify a tangential end extension, and enter the tool path's end extension for the current task in the End Extension Length field.

OR



Click on the Profile End Extension option to specify an end extension that follows the geometry, and enter the tool path's end extension for the current task in the End Extension Length field.

Specifying Leadin/Leadout Parameters



You can use the LEAD PARAMETERS option to specify lead in/lead out parameters for the current task.

Leadin/Leadout Strategy

You can define the approach and retraction strategy to be used during profiling motion, the default being no leadin/leadout strategy applied. Options include radial and/or linear leadins and leadouts.

Radial leadins are positioned so that they end tangentially to the subsequent profile tool path segment.

Radial leadouts are positioned so that they start tangentially to the preceding segment of the profile tool path.

You can control the following strategies:

- Radial

  Radius of radial leadin

  Radius of radial leadout

  Angular degrees of circular tool motion for leadin

  Angular degrees of circular tool motion for leadout

- Linear

  Angle of linear leadin

  Angle of linear leadout

  Length of linear leadin

  Length of linear leadout

- Strategy selection

  Radial leadin only

  Linear radial only

  Linear leadin connected to a subsequent Radial leadin

  Radial leadout only

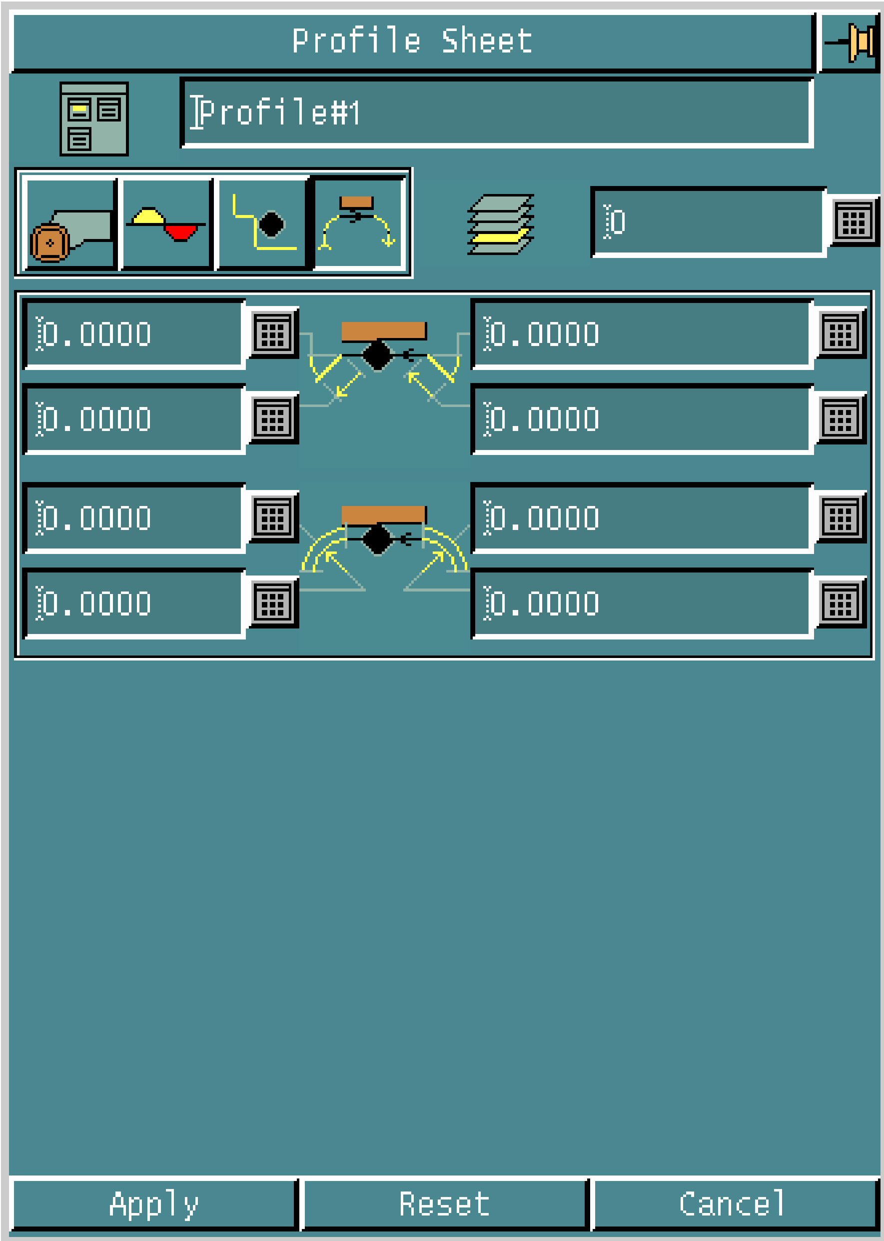
  Linear leadout only

  Radial leadout connected to a subsequent Linear leadout

Using this Option

1. Choose the Lead Parameters option from the Profile sheet. The set of leadin and leadout parameters appears on the sheet.

Figure 4-17 Profile Sheet for Leadin/Leadout Parameters



2. Specify the angle for linear leadout in the Linear Leadout Angle field.
3. Specify the angle for linear leadin in the Linear Leadin Angle field.
4. Specify the length of the linear leadout in the Linear Leadout Length field.

5. Specify the length of the linear leadin in the Linear Leadin Length field.
6. Specify the angle for radial leadout in the Radial Leadout Angle field.
7. Specify the angle for radial leadin in the Radial Leadin Angle field.
8. Specify the radius of the radial leadout in the Radial Leadout Radius field.
9. Specify the radius of the radial leadin in the Radial Leadin Radius field.
10. Click Apply if you have finished entering all parameters for the task.

Executing the Job

You have to execute the task after specifying parameters for it. After you have clicked Apply on the Profile sheet, use the following steps for executing your job:

1. Click Apply on the Operation sheet after specifying all the tasks for the current operation.
2. If you have used synchronization sections, you can see the Sync Master option in the NCBuilder sheet.

Figure 4-18 NCBuilder Sheet to Choose the Master Turret

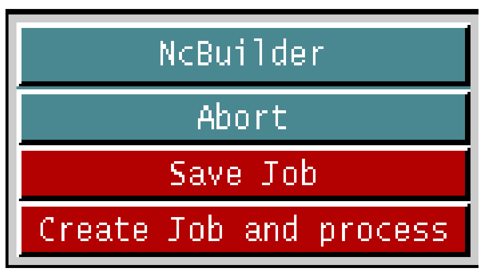
- Choose this option to select the master turret. The Synchronization Master Sheet appears.

Figure 4-19 Synchronization Master Sheet

- Choose the master turret for each synchronization section from the option list. For each synchronization section, CVNC uses feed rates and spindle speeds of the master turret.
- Click Apply.

3. Click Apply on the NCBuilder sheet after specifying all the operations for the current job. The following pulldown menu appears:

Figure 4-20 Menu for Execution of a Job



4. Click Create Job and process.

   You can now see the tool path and execution of your job.

Please note: If you have used synchronization sections, CVNC sorts the job control file (JCF) when you apply the job, such that there is minimum lead between turrets.

Chapter 5

Turning and Facing Tasks

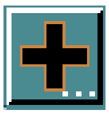
This chapter describes how you can use the NCBuilder Graphical User Interface (GUI) for turning and facing tasks. Turning and facing have some similarities though they are different operations. In conventional z-axis machines used for machining shafts, turning normally occurs linearly along the length of the component, constantly reducing the diameter. Facing occurs in a radial motion, reducing the length of the component. The sections that follow elucidate these operations. You can also refer to “TURNFACE” on page A-3 or the online help file to see the syntax of the command issued by CVNC, for turning and facing tasks.

- Accessing the Turning and Facing Options
- Specifying Geometry
- Specifying the Feed Rates for an Operation
- Specifying the Spindle Speed for an Operation
- Specifying the Coolant Setting for an Operation
- Specifying the Turning and Facing Parameters
- Specifying Tool Assembly Selection Parameters
- Specifying Tolerance Parameters
- Specifying Machining Parameters
- Specifying More Machining Parameters
- Specifying Parameters for the Final Profile Pass
- Specifying Leadin/Leadout Parameters for the Final Profile Pass
- Executing the Job

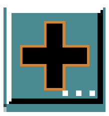
Accessing the Turning and Facing Options

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.

4. Choose the Job Block option from the NCBuilder task set. The NCBuilder property sheet appears.

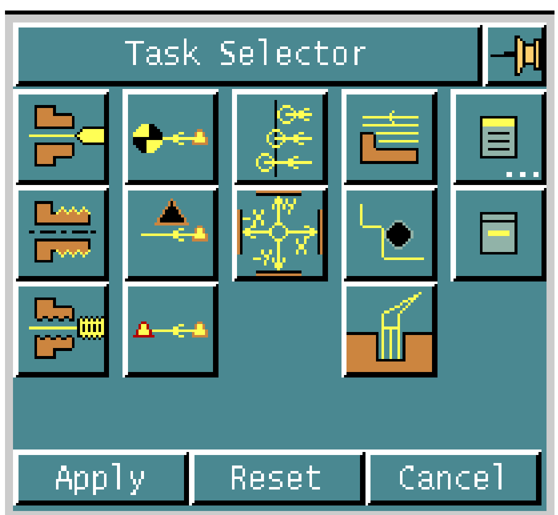


5. Choose the Add Operation option from the NCBuilder property sheet. The Operation Sheet appears.

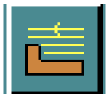


6. Choose the Add Task option from the operation sheet. The Task Selector palette appears.

Figure 5-1 Task Selector Palette

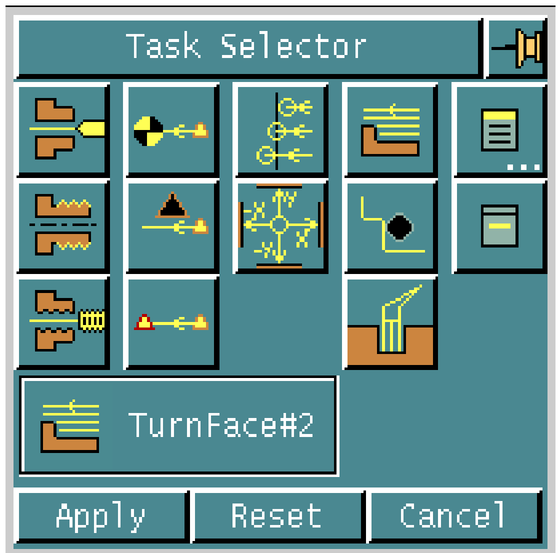


Please note: You should define the machine tool parameters and tooling parameters before specifying any task. See Chapter 2, “Machine Tool Definitions” for details.



7. Choose the TurnFace option from the Task Selector palette. The task name and its icon appear in the Task Selector palette as shown in the following figure. The sequence number of the task also appears. For example, if the task is the second turn/face task in the operation, it has the number 2.

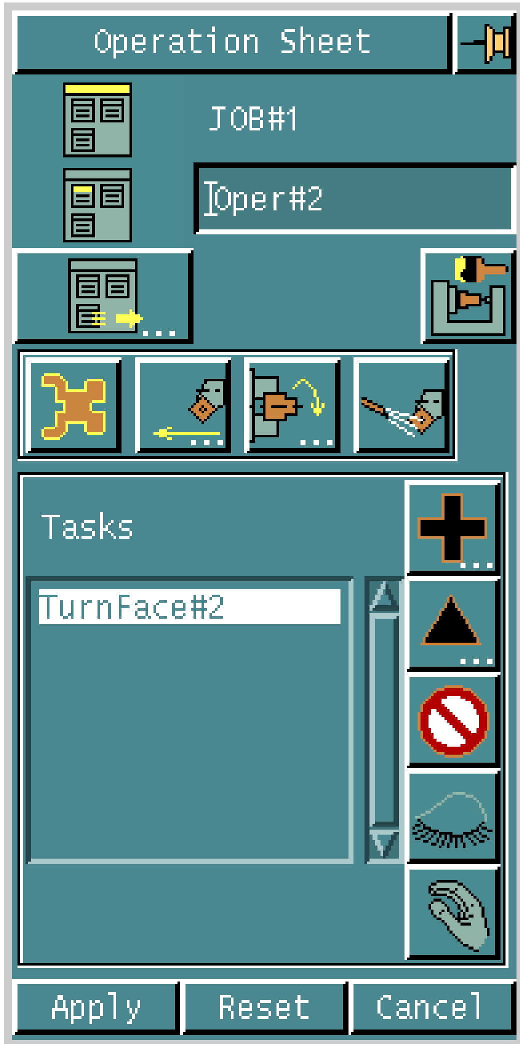
Figure 5-2 Task Selector Palette with Task Name and Icon



8. Click Apply.

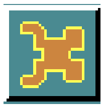
   The task now appears in the operation sheet which changes when the task is highlighted as shown in the following figure.

Figure 5-3 Operation Sheet after Adding a Task



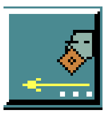
Options on the Operation Sheet

The various options on the operation sheet are:



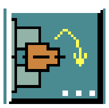
SPECIFYING GEOMETRY
Displays a property sheet that enables you to define geometry for the part and material boundaries.

See “Specifying Geometry” on page 5-6 for details.



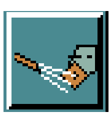
SPECIFYING FEED RATES
Displays a property sheet that enables you to specify the feed rates for the current operation.

See “Specifying the Feed Rates for an Operation” on page 5-10 for details.



SPECIFYING SPINDLE SPEED
Displays a property sheet that enables you to specify the spindle speed parameters for the current operation.

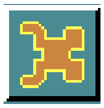
See “Specifying the Spindle Speed for an Operation” on page 5-13 for details.



SPECIFYING COOLANT SETTING
Enables you to specify the coolant setting for the current operation.

See “Specifying the Coolant Setting for an Operation” on page 5-14 for details.

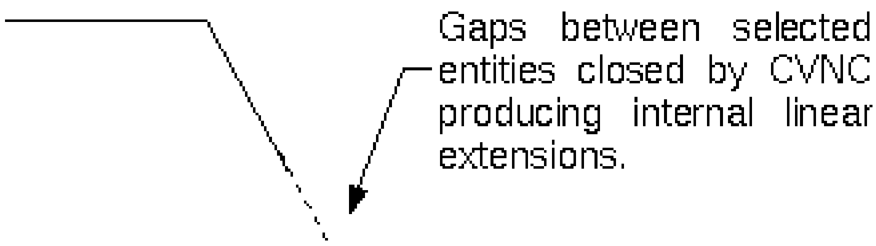
Specifying Geometry



Use the SELECT GEOMETRY option to define geometry for the part and material boundaries.

You can specify up to 512 wireframe entities for the part boundary. They must be in order. If there are gaps between the selected entities, which are greater than the current tolerance value, CVNC closes them automatically (see the figure below). CVNC also trims back overlapping geometry, if any, to form a contiguous part boundary.

Figure 5-4 Closing Gaps between Entities

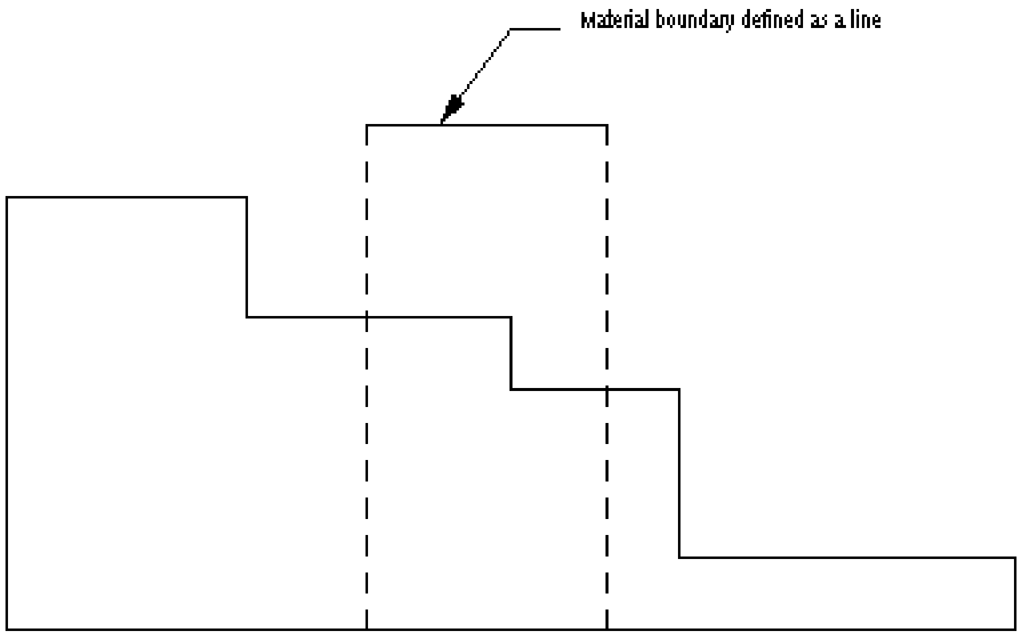


The material comprises the following:

- Forgings
- Castings
- Bar stock

You can define the material boundary using just a line. In addition, you can define open boundaries as well as closed boundaries. CVNC closes the open boundaries by dropping normals to the coordinate axes and connecting the points obtained by the intersections. Please see the following figure for details. In some cases even the part boundary is sufficient to define the material. Unlike in CVNC-T2, the material boundary can now fall within the bounds of the part. In such cases only the specified area is machined.

Figure 5-5 Open Material Boundary Within the Bounds of the Part



You can also specify up to 512 wireframe entities for the material boundary. These entities must be in order.

If there are gaps between the selected entities, which are greater than the current tolerance value, CVNC closes them automatically. CVNC also trims back overlapping geometry, if any, to form a contiguous material boundary. Alternatively you can digitize 2 locations to define the limits of a rectangular region which encloses the material boundary.

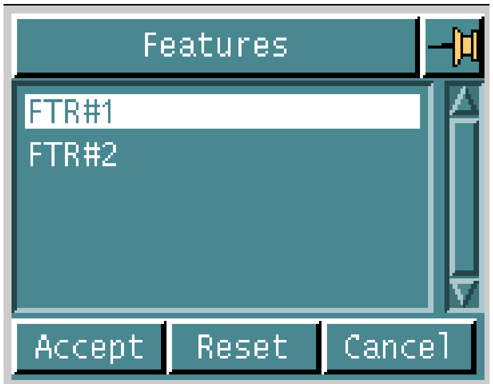
Using this Option

1. Choose the Select Geometry option from the operation sheet. The Feature Selection property sheet appears.

Figure 5-6 Feature Selection Property Sheet

2. Click the list button. The Features scroll list appears.

Figure 5-7 Features Scroll List

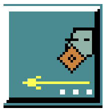


- Choose the feature from the list and click Accept. The selected feature appears in the Part Boundary field of the Feature Selection sheet.

  OR

3. Digitize the feature for the part boundary if you are selecting it for the first tine.
4. Digitize the feature for the material boundary.
5. Click Apply.

Specifying the Feed Rates for an Operation



Use the FEED PARAMETERS option to specify the feed rates for the current operation.

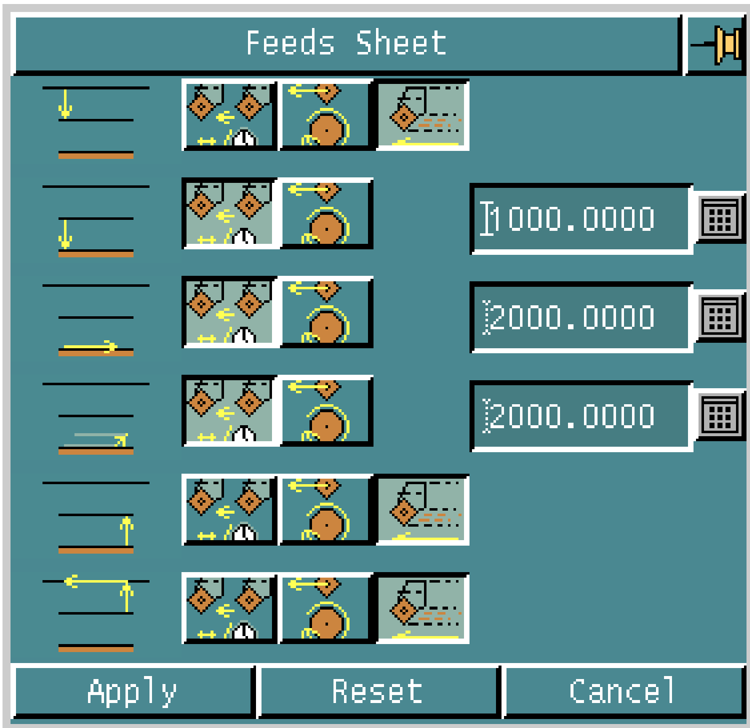
You can specify separate feed rates for each of the following:

- The general cutting motion.
- Connections between passes.
- Stepin and Stepout motion. The default for this is the Cut feed rate.
- The initial motion from the current location to the start point. The default for this is the Rapid feed rate.
- The motion from the end of a cut to the start point. The default for this is the Approach feed rate.
- The approach motion from the start point to the stepin motion of the first cut.
- The final optional profile cut.

Using this Option

1. Choose the Feed Parameters option from the operation sheet. The Feeds Sheet appears.

Figure 5-8 Feeds Sheet



2. Click and enter the approach feed rate. You can specify it in units of mm/min or mm/rev.

   OR

3. Specify it to be the same as the rapid feed rate.
4. Click and enter the plunge feed rate. You can specify it in units of mm/min or mm/rev.
5. Click and enter the cut feed rate. You can specify it in units of mm/min or mm/rev.
6. Click and enter the connect feed rate. You can specify it in units of mm/min or mm/rev.
7. Click and enter the retract feed rate. You can specify it in units of mm/min or mm/rev.

   OR

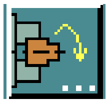
8. Specify it to be the same as the rapid feed rate.

9. Click and enter the clear feed rate. You can specify it in units of mm/min or mm/rev.

   OR

10. Specify it to be the same as the rapid feed rate.
11. Click Apply.

Specifying the Spindle Speed for an Operation



Use the SPINDLE SPEED PARAMETERS option to specify the spindle speed parameters for the current operation.

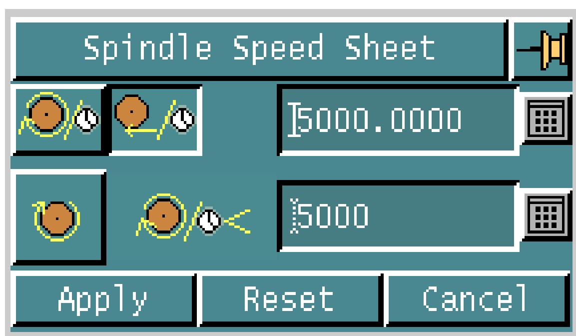
You can specify separate spindle speeds for the following:

- The general cutting motion.
- The optional final profile cut.

Using this Option

1. Choose the Spindle Speed Parameters option from the operation sheet. The Spindle Speed Sheet appears.

Figure 5-9 Spindle Speed Sheet

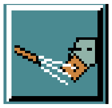


2. Click and enter the spindle speed in revolutions per minute.

   OR

3. Click and enter the spindle speed in surface meters per minute. If you choose this option you must specify the maximum spindle speed in revolutions per minute.
4. Click and specify the direction of rotation of the spindle.
5. Click Apply.

Specifying the Coolant Setting for an Operation



Use the COOLANT option to specify the coolant setting for the current operation.

Using this Option

1. Click the Coolant option on the operation sheet.
2. Choose the coolant setting to be one of ON, MIST, FLOOD or OFF.

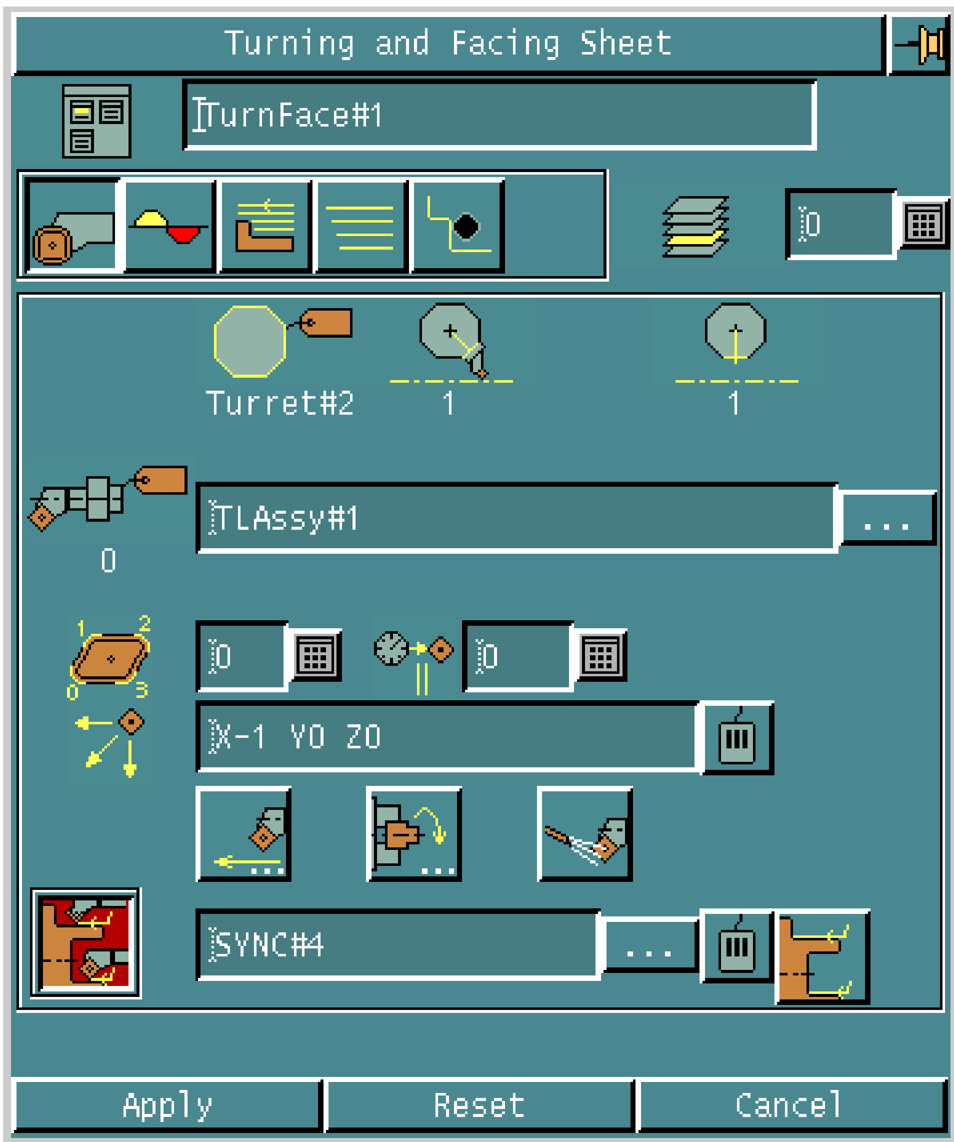
Specifying the Turning and Facing Parameters

Use the EDIT option on the operation sheet to specify parameters for turning and facing.

Using this Option

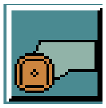
1. Highlight the task and choose the Edit option on the operation sheet. The Turning and Facing Sheet appears.

Figure 5-10 Turning and Facing Sheet



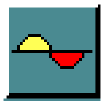
2. Specify the layer for creating the tool path in the Layer field.

Options on the Turning and Facing Sheet



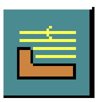
SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 5-17 for details.



SPECIFYING TOLERANCE PARAMETERS
Displays a property sheet that enables you to specify tolerance and stock parameters for the current task.

See “Specifying Tolerance Parameters” on page 5-20 for details.

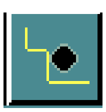


SPECIFYING MACHINING PARAMETERS
Displays a property sheet that enables you to specify machining parameters for the current task.

See “Specifying Machining Parameters” on page 5-22 for details.

SPECIFYING MORE MACHINING PARAMETERS
Displays a property sheet that enables you to specify more machining parameters for the current task.

See “Specifying More Machining Parameters” on page 5-28 for details.

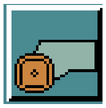


SPECIFYING PARAMETERS FOR THE FINAL PROFILE PASS
Displays a property sheet that enables you to specify parameters for the optional final profile pass.

See “Specifying Parameters for the Final Profile Pass” on page 5-35 for details.

Please note: For each of these options, a different set of parameters appears on the same turning and facing sheet.

Specifying Tool Assembly Selection Parameters

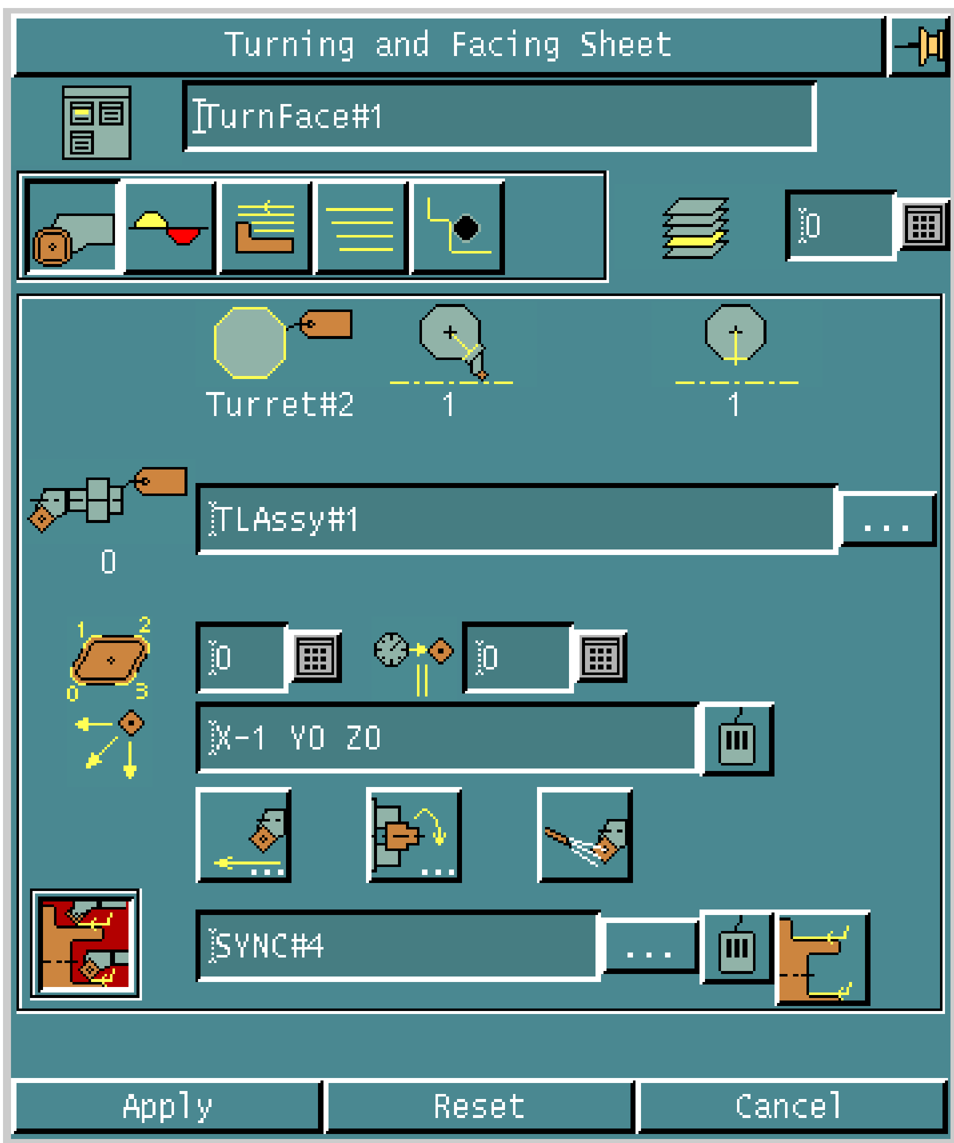


Use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

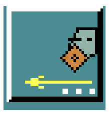
Using this Option

1. Choose the Tool Parameters option from the turning and facing sheet. The set of parameters for the tool assembly appears on the sheet.

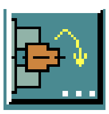
Figure 5-11 Turning and Facing Sheet for Tool Assembly Selection Parameters



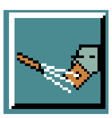
2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field. To do this,
   - Click the list button. The Tool Selector scroll list appears.
   - Choose a tool assembly from the list.
   - Click Accept.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.
5. Digitize a vector for the drive vector in the Drive Vector field. This vector specifies the cutting direction for the current task.



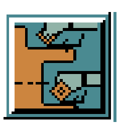
6. Choose the Feed Parameters option and enter the feed rates for the current task as explained in “Specifying the Feed Rates for an Operation” on page 5-10.



7. Choose the Spindle Speed Parameters option and enter the spindle speed parameters for the current task as explained in “Specifying the Spindle Speed for an Operation” on page 5-13.



8. Choose the Coolant option and specify the coolant setting as explained in “Specifying the Coolant Setting for an Operation” on page 5-14.



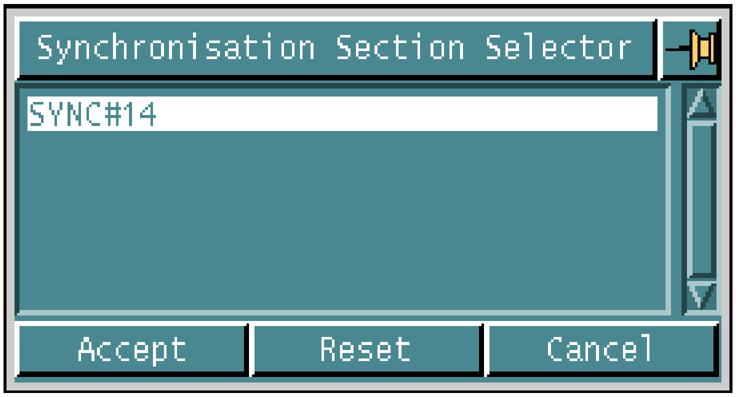
9. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

10. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 5-12 Synchronization Section Selector Scroll List

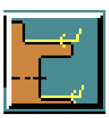


- Choose a synchronization section from the list.
- Click Accept.

  OR

11. You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

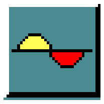
  OR



12. Choose the New Sync Section option to create a new synchronization section to add the task.

13. Click Apply if you have finished entering all parameters for the task.

Specifying Tolerance Parameters

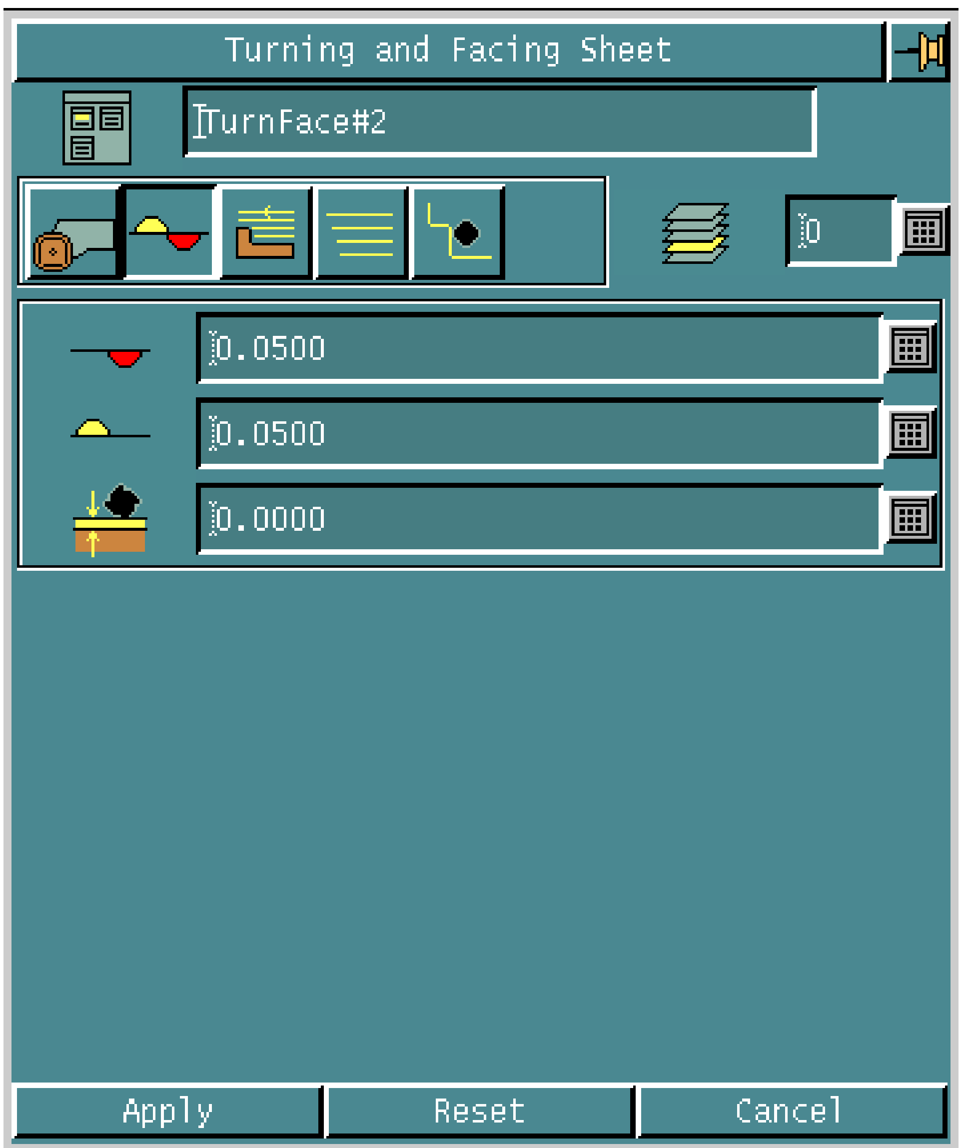


Use the TOLERANCE PARAMETERS option to specify tolerance and stock parameters for the current task.

Using this Option

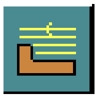
1. Choose the Tolerance Parameters option from the turning and facing sheet. The set of parameters for tolerance appears on the sheet.

Figure 5-13 Turning and Facing Sheet for Tolerance Parameters



2. Specify the inner tolerance for the current task in the Inner Tolerance field.
3. Specify the outer tolerance for the current task in the Outer Tolerance field.
4. Specify the stock offset for the current task in the Stock Offset field.
5. Click Apply if you have finished entering all parameters for the task.

Specifying Machining Parameters

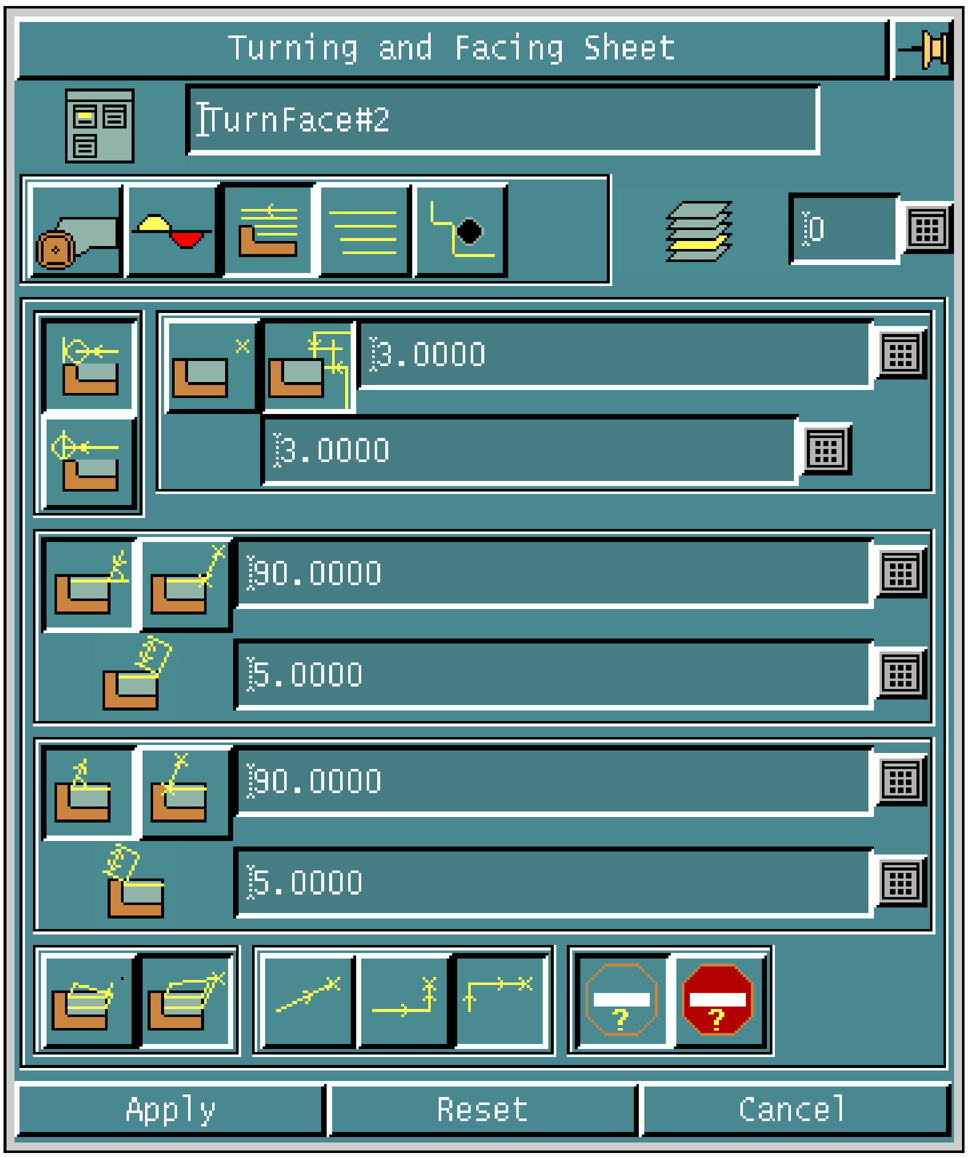


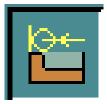
Use the MACHINING PARAMETERS option to specify machining parameters for the current task.

Using this Option

1. Choose the Machining Parameters option from the turning and facing sheet. The set of machining parameters appears on the sheet.

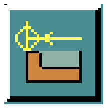
Figure 5-14 Turning and Facing Sheet for Machining Parameters





2. Choose the To option to specify that the tool insert is positioned TO the material boundary at the end of the cut.

OR



3. Choose the On option to specify that the tool insert is positioned ON the material boundary at the end of the cut.

Please note: At the start of a cut, the tool insert is always positioned TO the material boundary.

The following figures show the TO and ON conditions for turning and facing processes:

Figure 5-15 TO and ON Conditions for a Turning Process

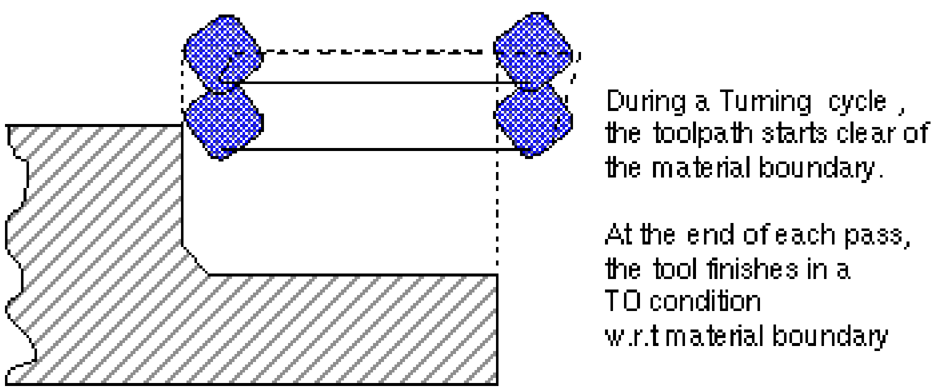
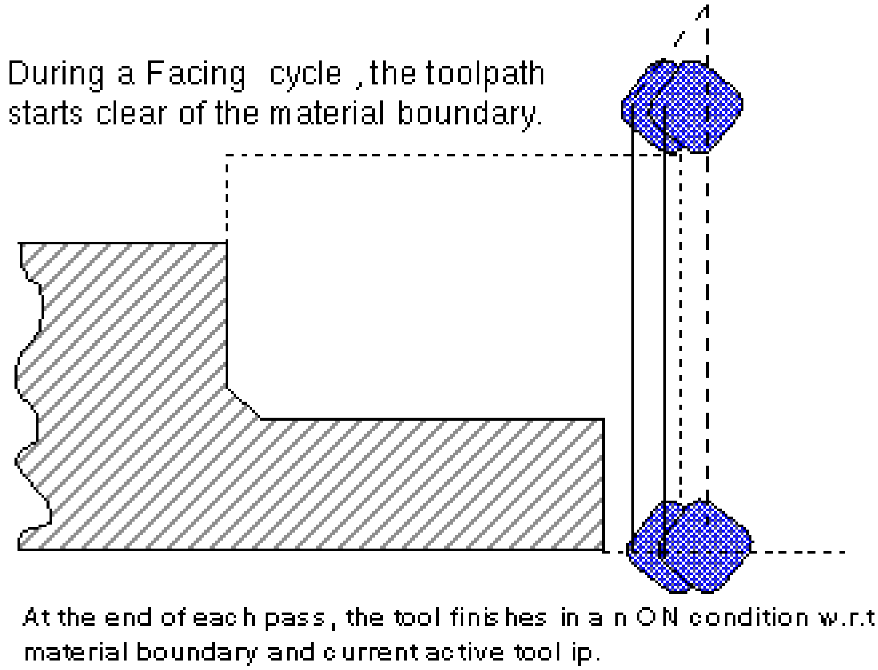
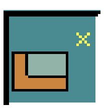


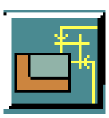
Figure 5-16 TO and ON Conditions for a Facing Process





4. Choose the `Loc` option if you want to enter a location for the start point of the tool. Alternatively, you may digitize the start point.

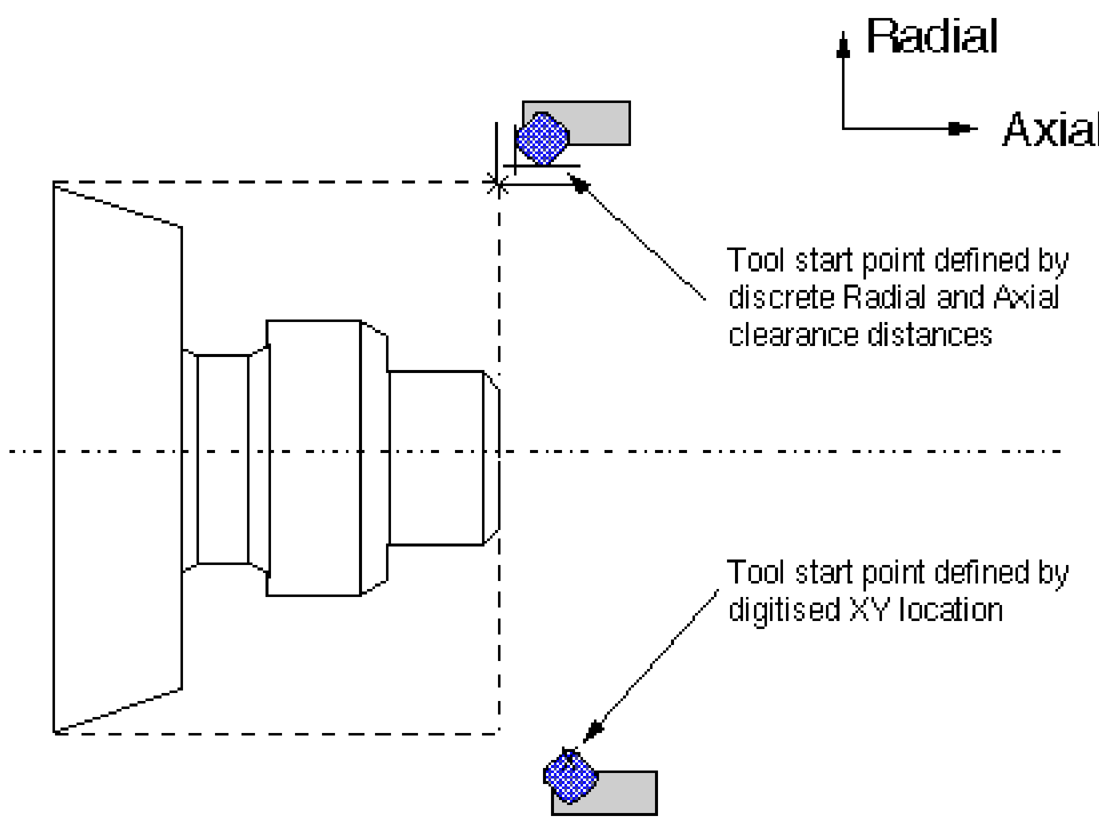
   OR

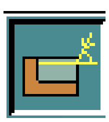


5. Choose the `Clear` option to specify the start point in terms of the clearance distance. This is the distance in x and y-coordinates by which the tool is offset from the material boundary. Enter the values for the x and y-coordinates.

   The following figure shows the two methods of defining the start point:

Figure 5-17 Defining the Start Point

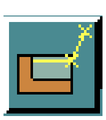




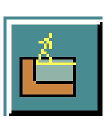
6. Choose the `Stepin by Angle` option and enter the angle of the stepin vector.

   This angle is measured relative to the cut direction vector. Thus an angle of 0 degrees means that the stepin vector is parallel to and in the same direction as the cut direction vector, whereas an angle of 180 degrees means that the stepin vector is parallel to but in the opoosite direction as the cut direction vector. A counter clockwise direction indicates a positive angle. The stepin vector is completely independent of the machine, turret, station and tool assembly configurations.

   OR



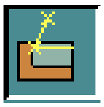
7. Choose the `Stepin by Loc` option and enter a vector for the stepin vector. Alternatively, you may digitize this vector.
8. Specify the length of the stepin vector in the `Stepin Length` field.



9. Choose the `Stepout by Angle` option and enter the angle of the stepout vector.

This angle is measured relative to the cut direction vector. Thus an angle of 0 degrees means that the stepout vector is parallel to and in the same direction as the cut direction vector, whereas an angle of 180 degrees means that the stepout vector is parallel to but in the opoosite direction as the cut direction vector. A counter clockwise direction indicates a positive angle. The stepout vector is completely independent of the machine, turret, station and tool assembly configurations.

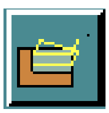
OR



10. Choose the Stepout by Loc option and enter a vector for the stepout vector. Alternatively, you may digitize this vector.

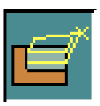
11. Specify the length of the stepout vector in the Stepout Length field.

Please note: If you enter a zero value for the length of the stepin or stepout vectors, CVNC does not generate the corresponding motion.

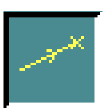


12. Choose the No Return to Start Point option to specify that you do not want the tool to return to the start point after each pass.

OR

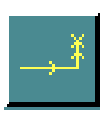


13. Choose the Return to Start Point option to specify that the tool should return to the start point after each pass. When you choose this option, the following additional options appear:



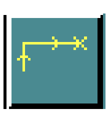
- Choose the Direct option to specify that the tool moves directly to the start point.

  OR



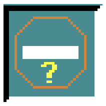
- Choose the Axial option to specify that the tool moves to the start point in an axial and radial motion, in that order.

  OR



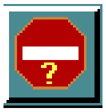
- Choose the Radial option to specify that the tool moves to the start point in a radial and axial motion, in that order.

Please note: If you select the Return to Start Point option the tool moves to the start point and then to the start of the stepin vector.



- Choose the Opstop Off option to specify that CVNC should not generate an OPSTOP after the tool has returned to the start point.

  OR



- Choose the Opstop On option to specify that CVNC should generate an OPSTOP after the tool has returned to the start point. This allows manual checking of the tool before the next pass.

14. Click Apply if you have finished entering all parameters for the task.

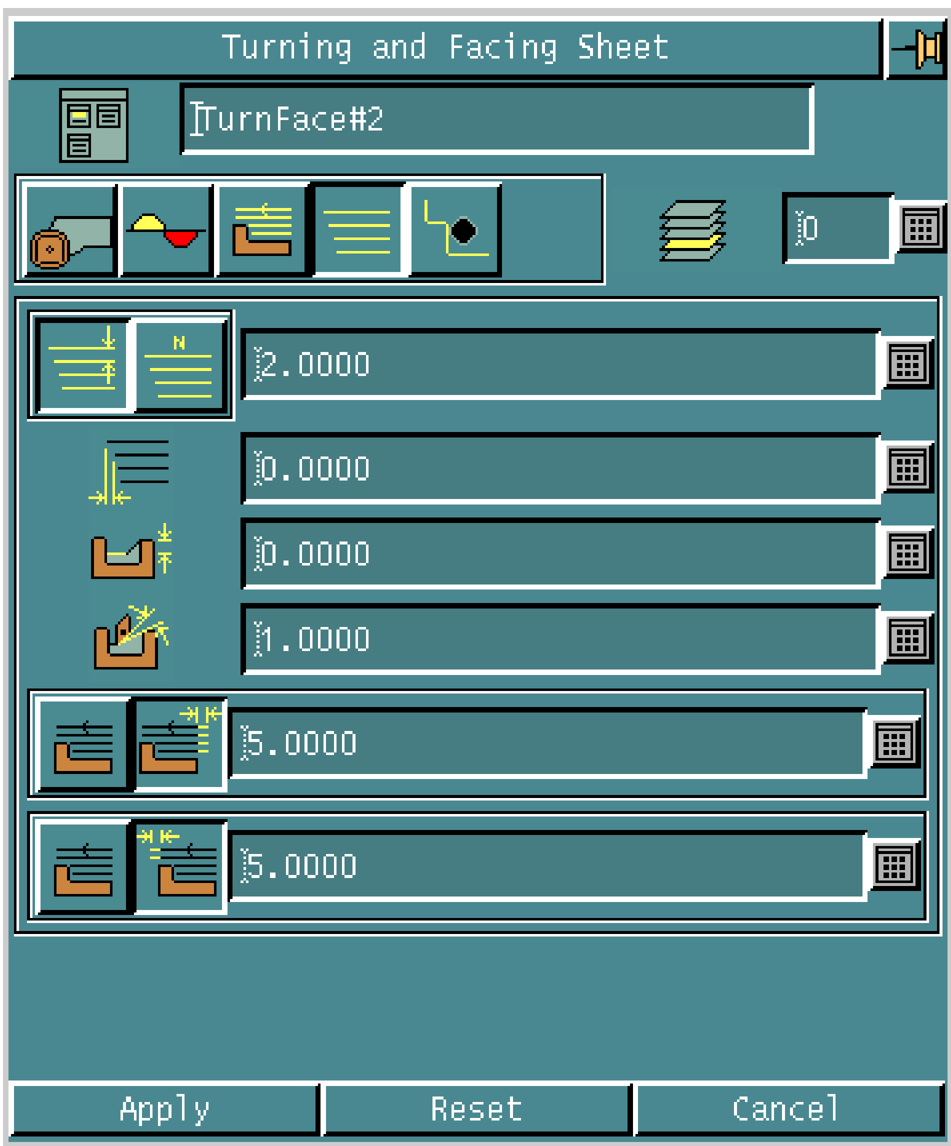
Specifying More Machining Parameters

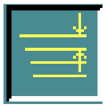
Use the MORE PARAMETERS option to specify more machining parameters for the current task.

Using this Option

1. Choose the More Parameters option from the turning and facing sheet. A set of more parameters appears on the sheet.

Figure 5-18 Turning and Facing Sheet for General Parameters





2. Choose the Maximum Depth option and enter the maximum depth of cuts. CVNC then calculates the number of cuts necessary to complete the process without exceeding the maximum depth of cuts.

   OR

3. Choose the Number of Cuts option and enter the actual number of cuts. In this case CVNC calculates the depth of cuts necessary to complete the process.

4. Specify the incremental end offsets in the Incremental End Offset field.

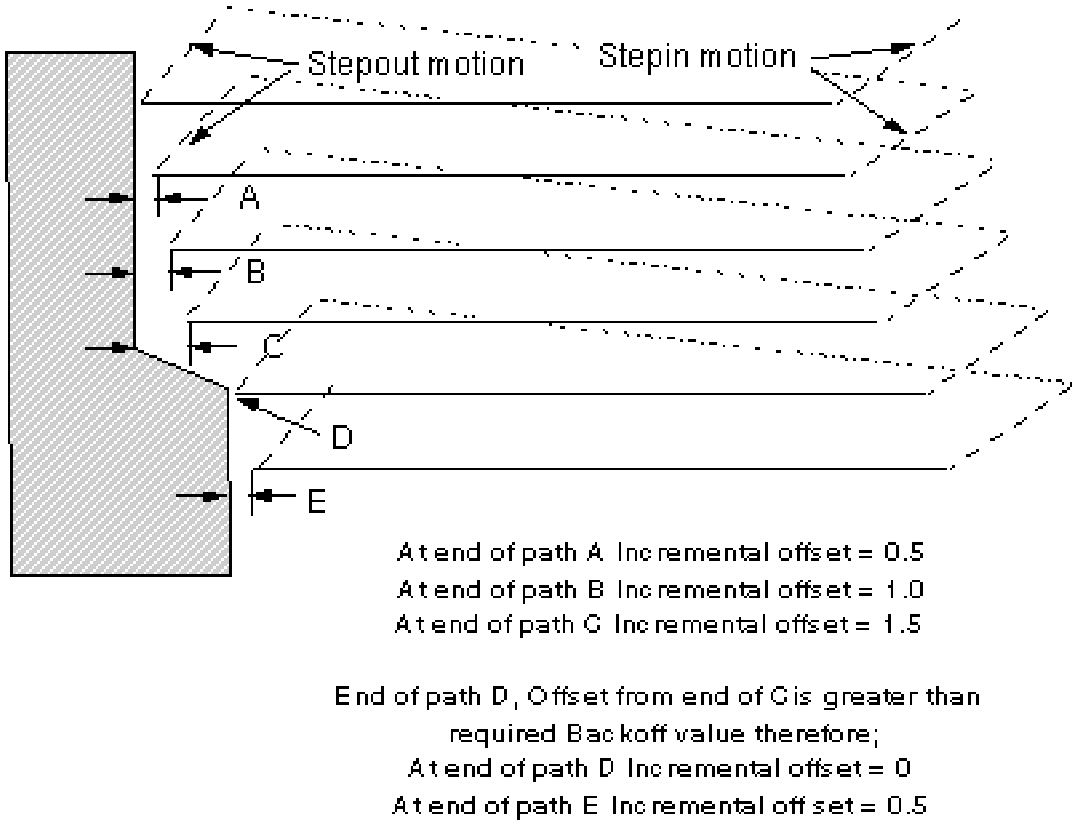
   Due to tool tip geometry and its orientation, it is often necessary to offset the end of each pass when machining up to a face. This avoids the tip from being buried into uncut material left by the previous pass. When you specify the incremental end offset, CVNC trims back the successive passes incrementally by this value.

   For example, if you specify a value of 0.5 mm for the offset, the following sequence takes place:

   - The first pass remains unchanged.
   - The second pass is trimmed back 0.5 mm.
   - The third pass is trimmed back 1.0 mm.
   - The fourth pass is trimmed back 1.5 mm and so on.

   If there is a natural break in the geometry where a cut terminates at a distance greater than or equal to the offset you specify from the end of the preceding cut, CVNC sets the offset to 0. Subsequent cuts are again offset using the value you have specified. See the following figure for details.

Figure 5-19 Incremental End Offsets



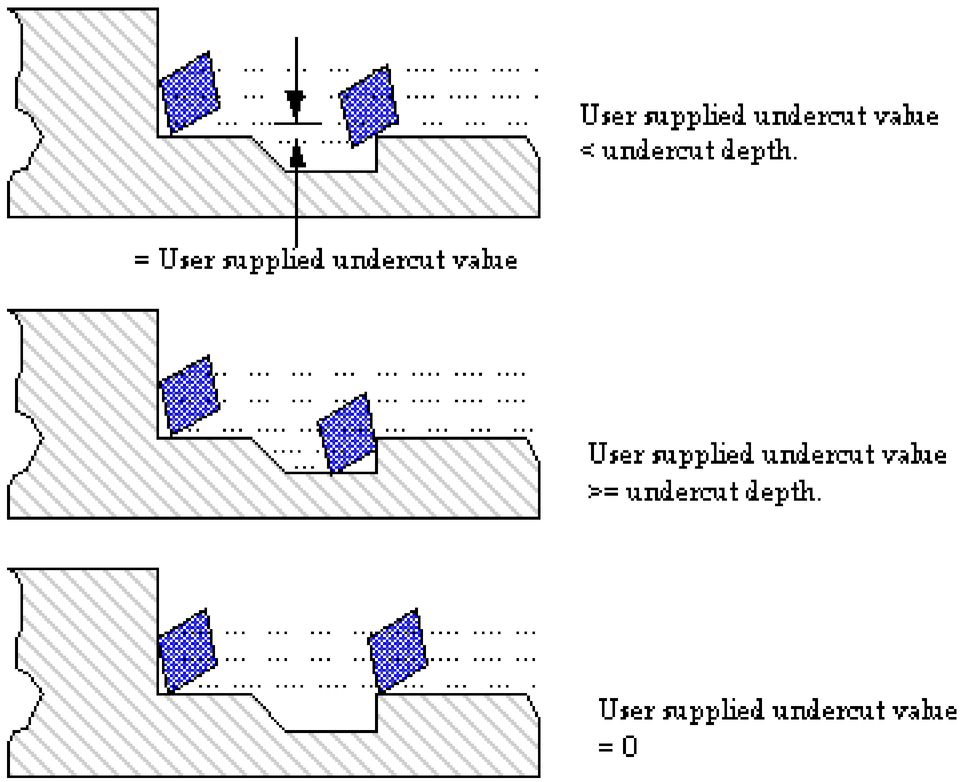
5. Specify the maximum machining depth for an undercut in the Undercut Depth field.

   The tool can cut into the undercut region to this value as the maximum. There are 3 cases that can occur while machining undercuts. They are as follows:

   - The value you specify is less than the depth of the undercut.
   - The value you specify is greater than or equal to the depth of the undercut.
   - You specify 0.

   The following figure shows how CVNC handles these cases.

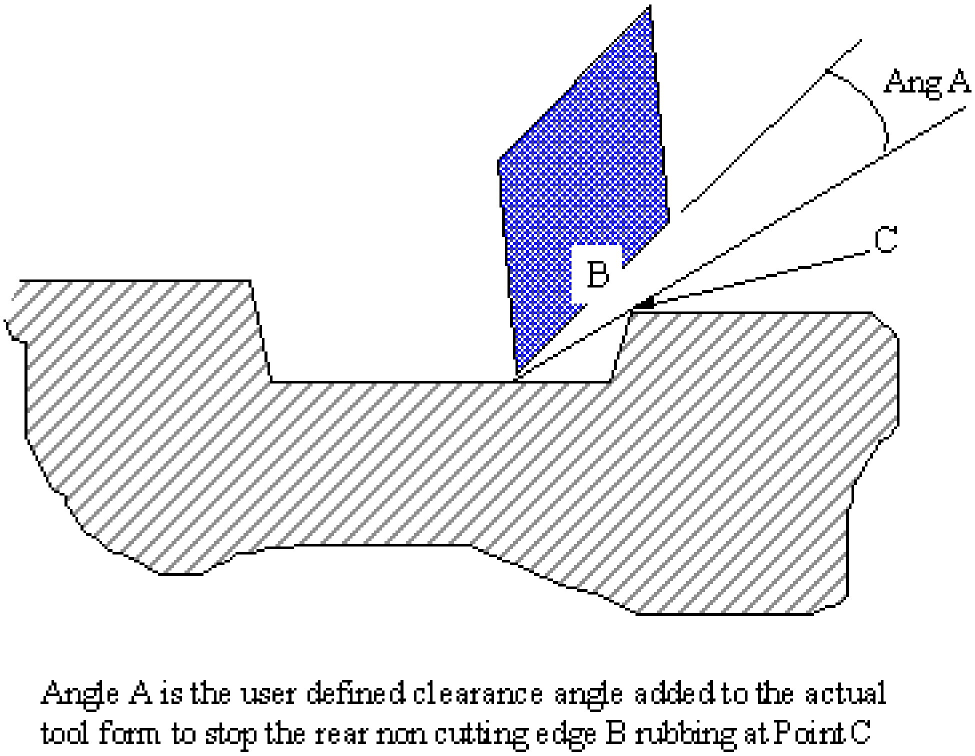
Figure 5-20 Undercuts with Depth Support

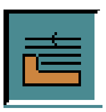


6. Specify the clearance angle in the Clearance Angle field.

   Sometimes the shape of the tool may be such that a non cutting edge may rub against the boundary of the undercut. To avoid this, you may specify a clearance angle to modify the tool path as shown in the following figure.

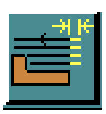
Figure 5-21 Clearance Angle





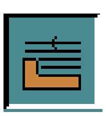
7. Choose the No Start Extension option to specify that you do not want a start extension.

   OR



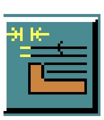
8. Choose the Start Extension option and enter the start extension in the field.

   This is required while machining cast or forged components where the tool must start clear of the material boundary. The start point is increased by this value for all the cuts. CVNC increases the length of the tool path in the direction of the drive vector.



9. Choose the No End Extension option to specify that you do not want an end extension.

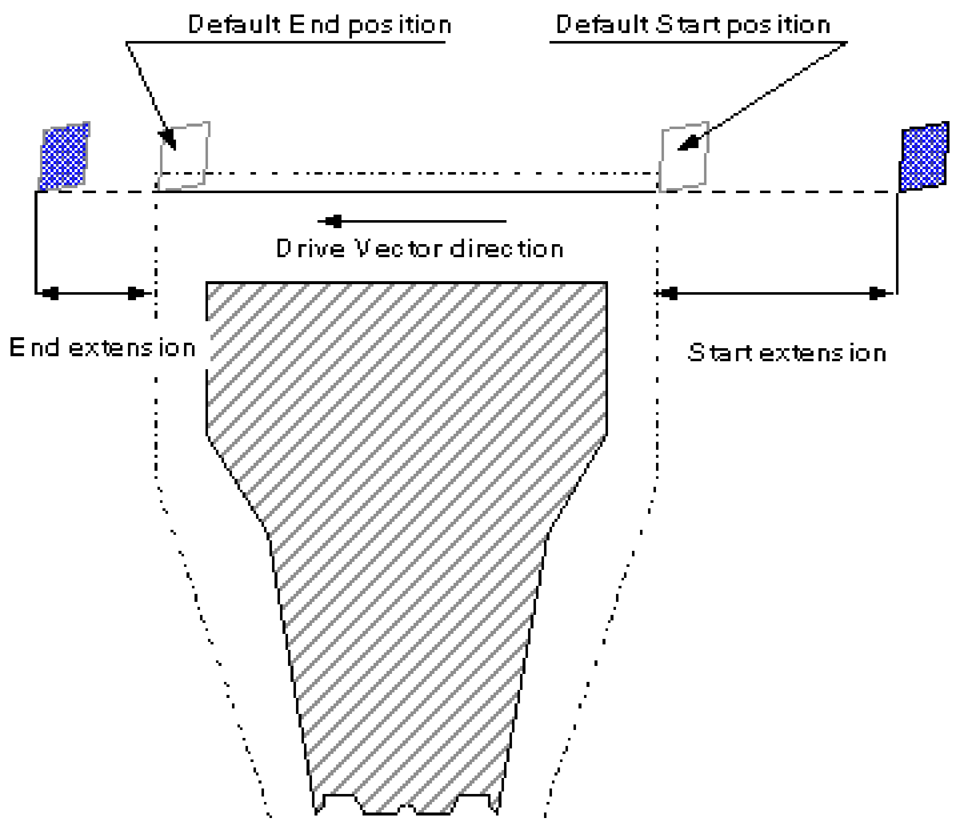
   OR

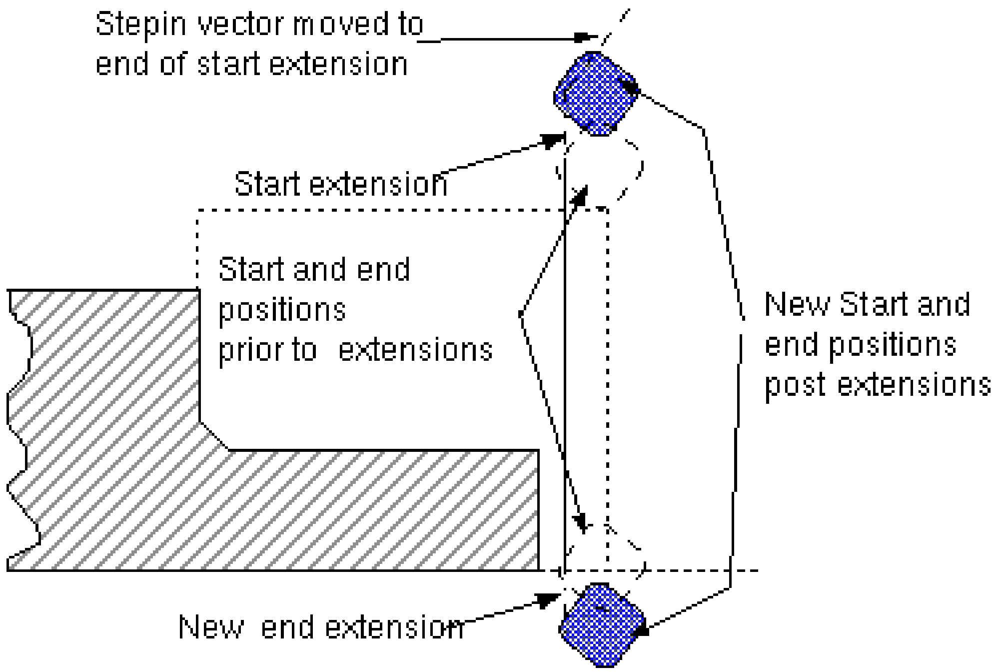


10. Choose the End Extension option and enter the end extension in the field.

This is required while machining cast or forged components where the tool must end clear of the material boundary. The end point is increased by this value for all the cuts. CVNC increases the length of the tool path in the direction of the drive vector.

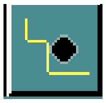
Figure 5-22 Facing Operation with Start and End Extensions





11. Click Apply if you have finished entering all parameters for the task.

Specifying Parameters for the Final Profile Pass

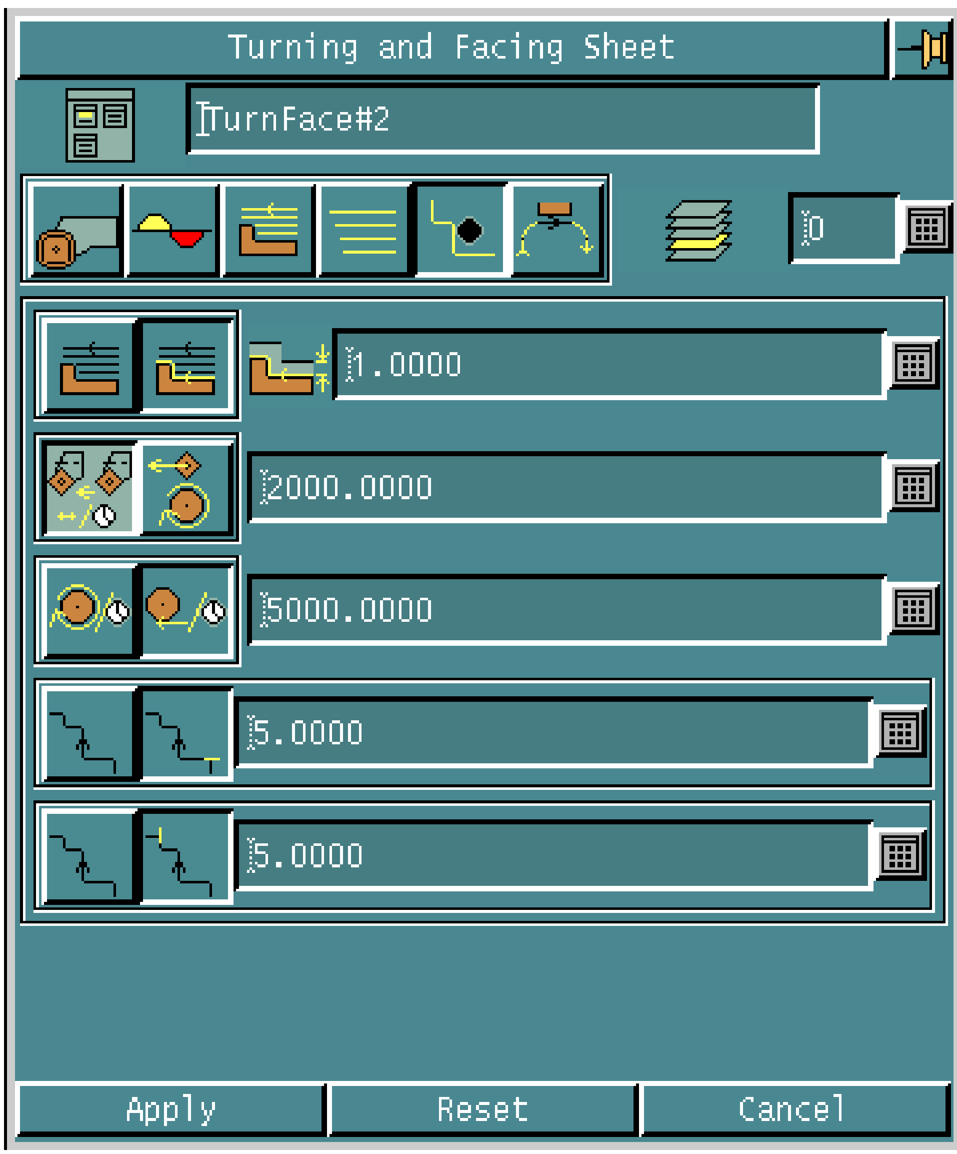


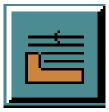
Use the PROFILE PARAMETERS option to specify parameters for the optional final profile pass.

Using this Option

1. Choose the Profile Parameters option from the turning and facing sheet. The set of parameters for the profile pass appears on the sheet.

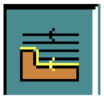
Figure 5-23 Turning and Facing Sheet for Profile Parameters



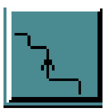


2. Choose the No Profile Pass option if you do not want a final profile pass. If you choose this option you will not see any parameters on the sheet.

OR

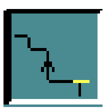


3. Choose the Profile Pass option to specify that you want a final profile pass. The parameters for the profile pass appear on the sheet as shown in the above figure.
4. Specify the stock offset for the current task in the Profile Allowance field.
5. Click and enter the feed rate for the profile pass. You can specify it in units of mm per min or mm per revolution.
6. Click and enter the spindle speed for the profile pass. You can specify it in revolutions per minute or surface meters per minute.

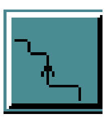


7. Choose the No Profile Start Extension option to specify that you do not want a start extension for the profile pass.

OR

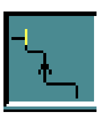


8. Choose the Profile Start Extension option and enter the start extension in the field.



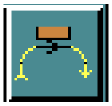
9. Choose the No Profile End Extension option to specify that you do not want a end extension.

OR



10. Choose the Profile End Extension option and enter the end extension in the field.
11. Click Apply if you have finished entering all parameters for the task.

Specifying Leadin/Leadout Parameters for the Final Profile Pass



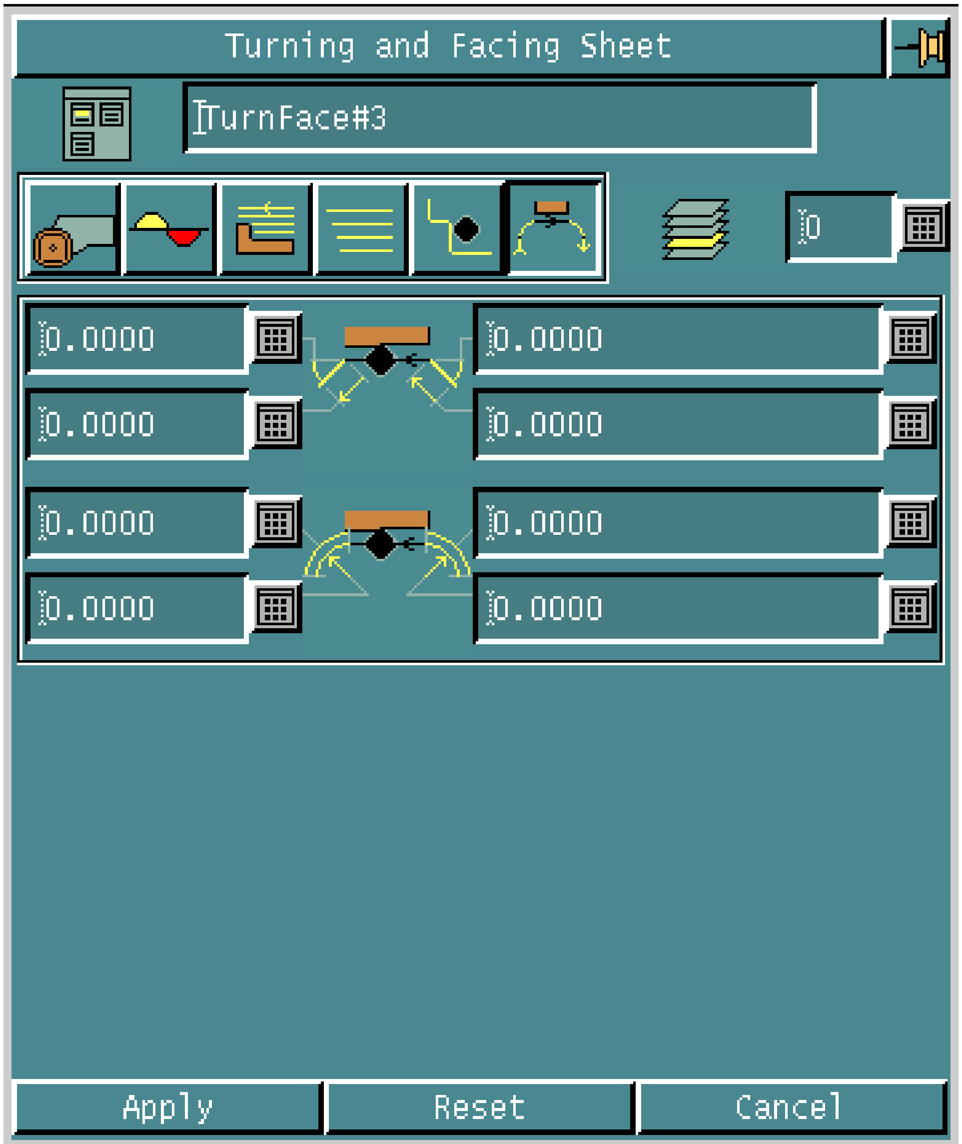
Choose the LEAD PARAMETERS option to specify the leadin and leadout parameters for the optional profile pass.

Please note: This option appears on the turning and facing sheet only if you choose the Profile Pass option as explained in “Specifying Parameters for the Final Profile Pass” on page 5-35.

Using this Option

1. Choose the Lead Parameters option from the turning and facing sheet. The set of leadin and leadout parameters appears on the sheet.

Figure 5-24 Turning and Facing Sheet for Leadin/Leadout Parameters



2. Specify the angle for the linear leadout in the Linear Leadout Angle field.
3. Specify the angle for the linear leadin in the Linear Leadin Angle field.
4. Specify the length of the linear leadout in the Linear Leadout Length field.
5. Specify the length of the linear leadin in the Linear Leadin Length field.
6. Specify the angle for the radial leadout in the Radial Leadout Angle field.

7. Specify the angle for the radial leadin in the Radial Leadin Angle field.
8. Specify the radius of the radial leadout in the Radial Leadout Radius field.
9. Specify the radius of the radial leadin in the Radial Leadin Radius field.
10. Click Apply if you have finished entering all parameters for the task.

Executing the Job

You have to execute the task after specifying parameters for it. After you have clicked Apply on the turning and facing sheet, use the following steps for executing your job:

1. Click Apply on the operation sheet when you finish specifying all the tasks for your operation.
2. If you have used synchronization sections, you can see the Sync Master option in the NCBuilder sheet.

Figure 5-25 NCBuilder Sheet to Choose the Master Turret

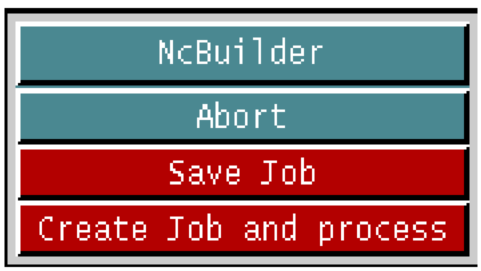
- Choose this option to select the master turret. The Synchronization Master Sheet appears.

Figure 5-26 Synchronization Master Sheet

- Choose the master turret for each synchronization section from the option list. For each synchronization section, CVNC uses feed rates and spindle speeds of the master turret.
- Click Apply.

3. Click Apply on the NCBuilder sheet after you finish specifying all the operations for your job. The following pulldown menu appears:

Figure 5-27 Pulldown Menu for Executing a Task



4. Click Create Job and process.

   You can now see the tool path and execution of your job.

Please note: If you have used synchronization sections, CVNC sorts the job control file (JCF) when you apply the job, such that there is minimum lead between turrets.

Chapter 6 Grooving Tasks

This chapter describes how you can use the NCBuilder Graphical User Interface (GUI) for grooving tasks on turned parts. You can create grooves of various shapes, at various locations on the turned parts. The following sections describe the grooving process in detail. You can also refer to "NGROOVE" on page A-2 or the online help file to see the syntax of the command issued by CVNC for grooving tasks.

- Overview of Grooves
- Accessing the Grooving Options
- Specifying the Grooving Parameters
- Specifying Tool Assembly Selection Parameters
- Specifying Tolerance Parameters
- Specifying Machining Parameters
- Specifying More Machining Parameters
- Specifying Parameters for the Final Profile Pass
- Executing the Job

Overview of Grooves

A groove is defined as a part of a feature between two entities called the start and end entities. You can select the start and end entities from a feature. All the entities between them (and excluding them) make up the groove.

You can machine grooves having various shapes and at various locations on the component using NCBuilder. The shapes and locations vary in complexity.

The following figures show examples of the different types of grooves:

Figure 6-1 Simple Groove

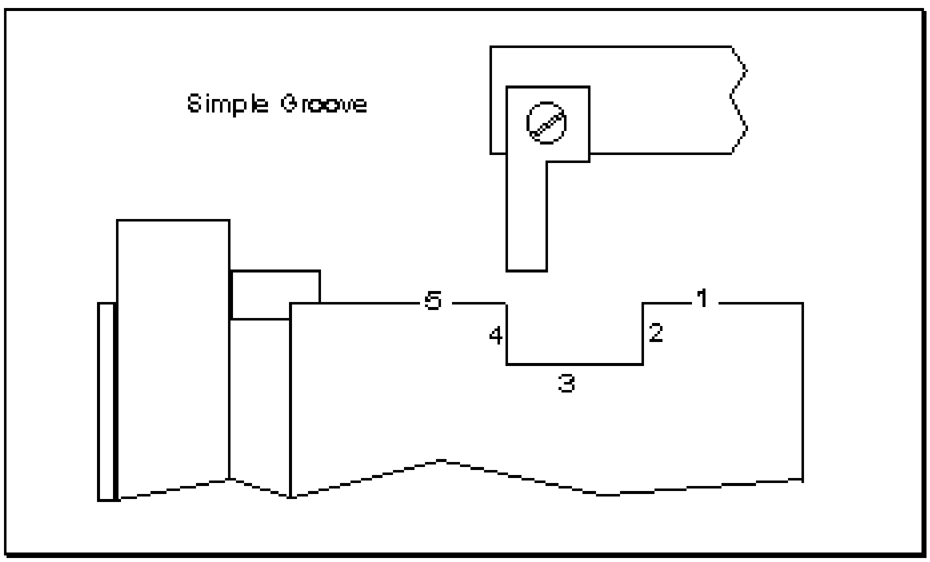


Figure 6-2 Medium Complexity Groove

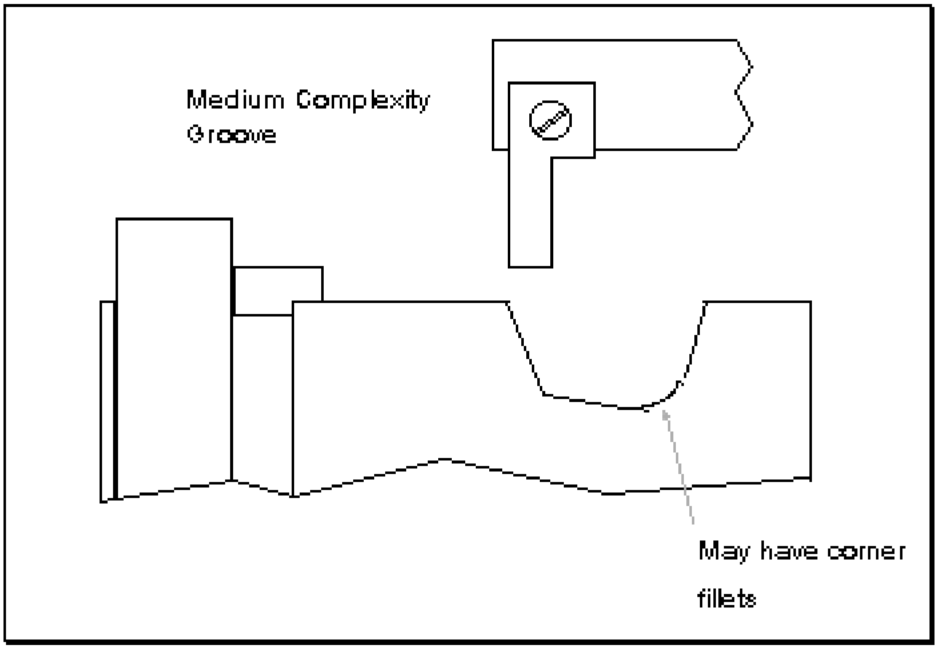


Figure 6-3 Complex Groove

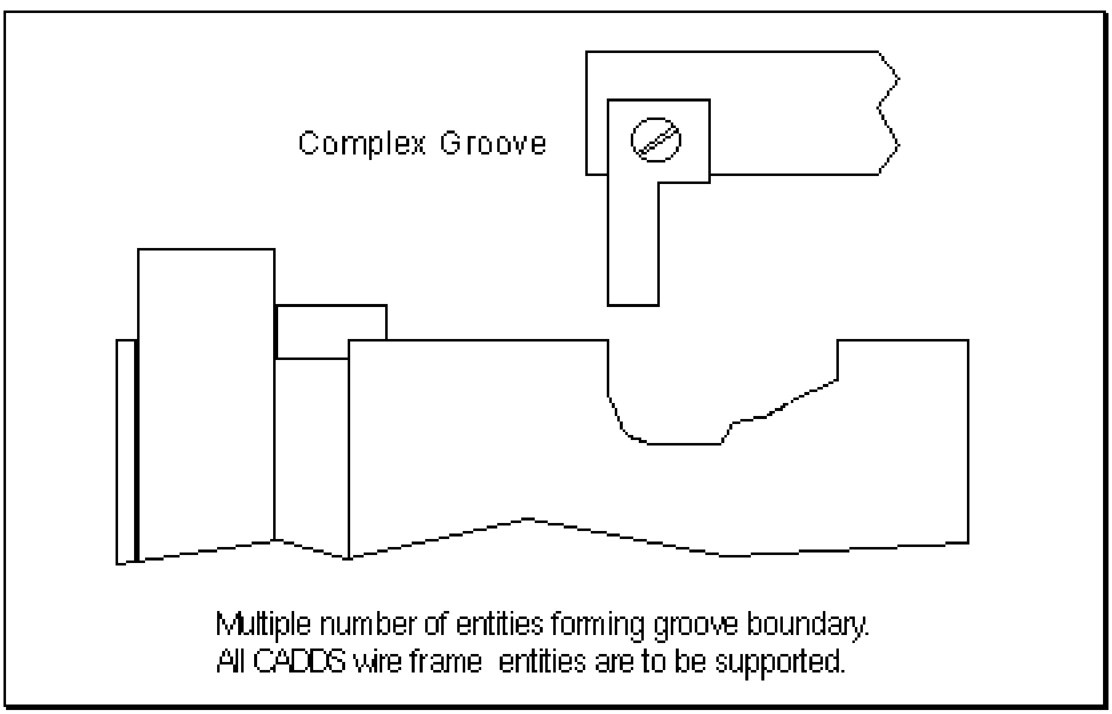


Figure 6-4 Groove with Different Start and End Diameters

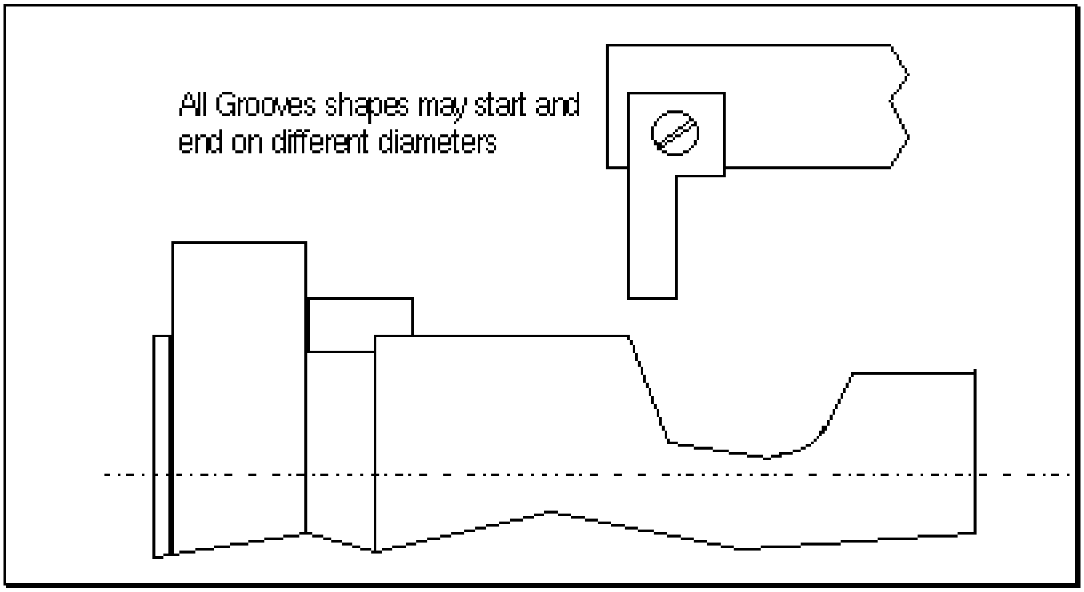
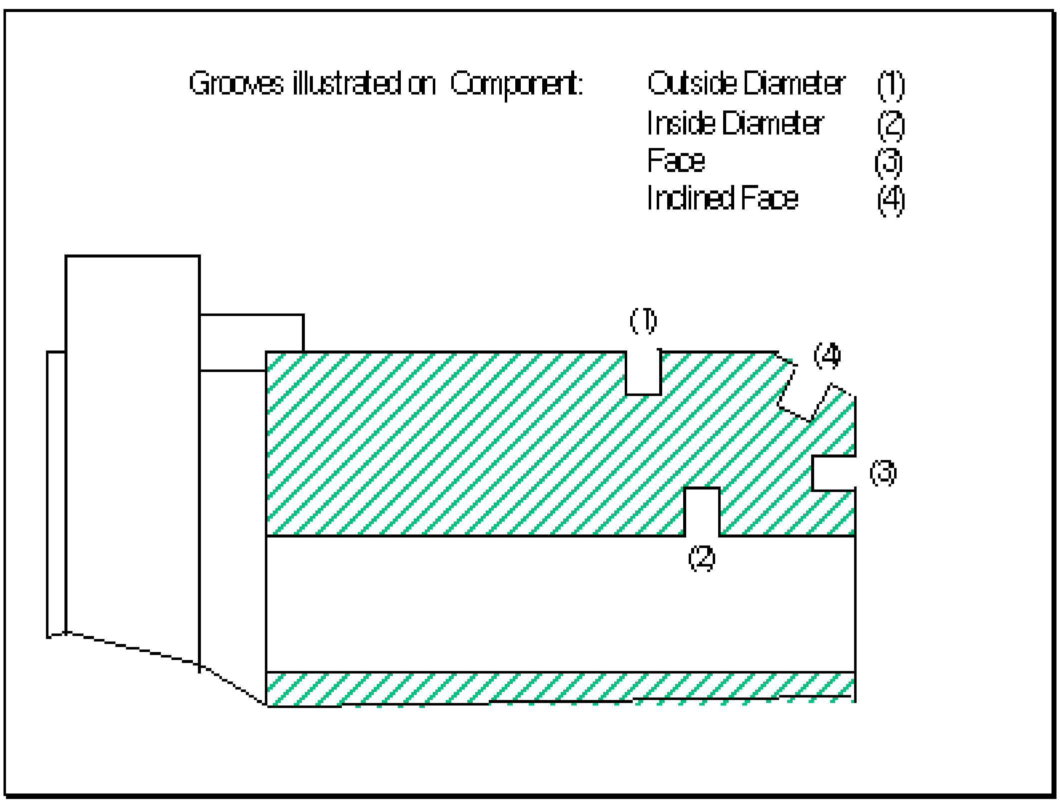


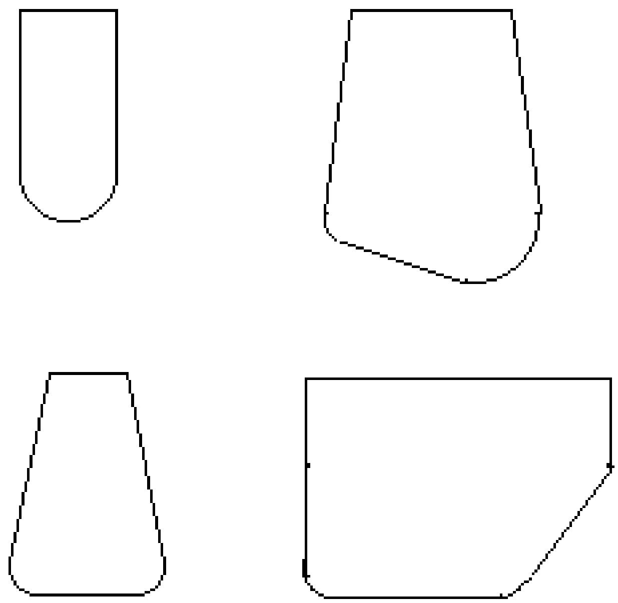
Figure 6-5 Grooves at Different Locations on the Component



Shapes of the Grooving Tool

You can use tools of different shapes while machining grooves. The following figure shows examples of the basic grooving tool shapes.

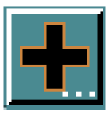
Figure 6-6 Shapes of Grooving Tools



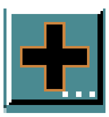
Accessing the Grooving Options

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.

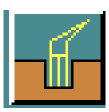
4. Choose the Job Block option from the NCBuilder task set. The NCBuilder property sheet appears.



5. Choose the Add Operation option from the NCBuilder property sheet. The Operation Sheet appears.

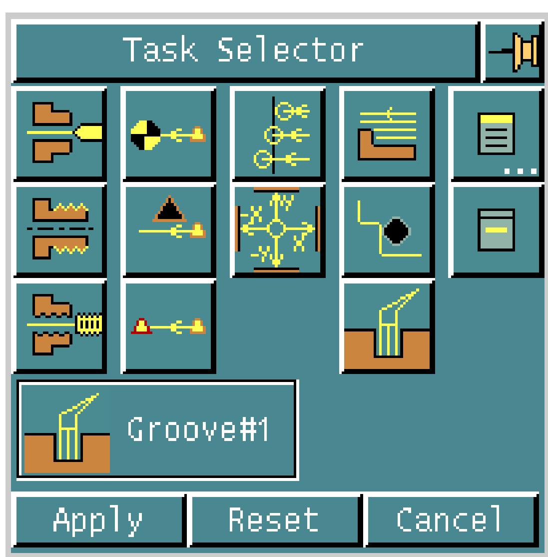


6. Choose the Add Task option from the operation sheet. The Task Selector palette appears.



7. Choose the Groove option from the Task Selector palette. The task name and its icon appear in the Task Selector palette as shown in the following figure. The sequence number of the task also appears. For example, if the task is the first grooving task in the operation, it has the number 1.

Figure 6-7 Task Selector Palette with Task Name and Icon



Please note: You should define the machine tool parameters and tooling parameters before specifying any task. See Chapter 2, “Machine Tool Definitions” for details.

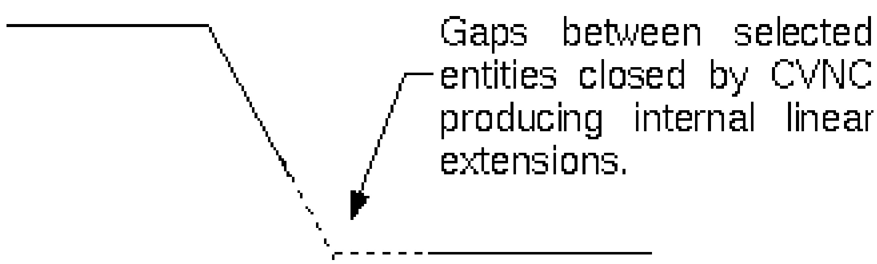
8. Click Apply.

   The task now appears in the operation sheet.

9. Specify the geometry as described in “Specifying Geometry” on page 5-6.

   You can specify up to 512 wireframe entities for the part boundary. They need to be in order. The direction in which you select these entities determines the default values of parameters such as the stepover direction; they will have the same direction. If there are gaps between the selected entities, which are greater than the current tolerance value, CVNC closes them automatically (see the figure below). CVNC also trims back overlapping geometry, if any, to form a contiguous part boundary.

Figure 6-8 Closing Gaps between Entities



   CVNC automatically generates the material boundary for the grooving process.

Please note: You can machine only one groove per task.

10. Specify the feed rates for the operation as described in “Specifying the Feed Rates for an Operation” on page 5-10.

11. Specify the spindle speed for the operation as described in “Specifying the Spindle Speed for an Operation” on page 5-13.

12. Specify the coolant setting for the operation as described in “Specifying the Coolant Setting for an Operation” on page 5-14.

Specifying the Grooving Parameters

Use the EDIT option on the operation sheet to specify parameters for the grooving process.

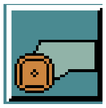
Using this Option

1. Highlight the task and choose the Edit option on the operation sheet. The Grooving Sheet appears.

Figure 6-9 Grooving Sheet

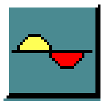
2. Specify the layer for creating the tool path in the Layer field.

Options on the Grooving Sheet



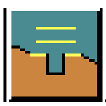
SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 6-10 for details.



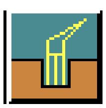
SPECIFYING TOLERANCE PARAMETERS
Displays a property sheet that enables you to specify tolerance and stock parameters for the current task.

See “Specifying Tolerance Parameters” on page 6-13 for details.



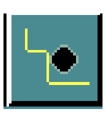
SPECIFYING MACHINING PARAMETERS
Displays a property sheet that enables you to specify machining parameters for the current task.

See “Specifying Machining Parameters” on page 6-15 for details.



SPECIFYING MORE MACHINING PARAMETERS
Displays a property sheet that enables you to specify more machining parameters for the current task.

See “Specifying More Machining Parameters” on page 6-19 for details.

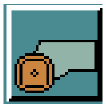


SPECIFYING PARAMETERS FOR THE FINAL PROFILE PASS
Displays a property sheet that enables you to specify parameters for the optional final profile pass.

See “Specifying Parameters for the Final Profile Pass” on page 6-23 for details.

Please note: For each of these options, a different set of parameters appears on the same grooving sheet.

Specifying Tool Assembly Selection Parameters



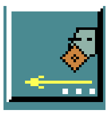
Use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

Using this Option

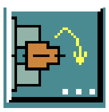
1. Choose the Tool Parameters option from the grooving sheet. The set of parameters for the tool assembly appears on the sheet.

Figure 6-10 Grooving Sheet for Tool Assembly Selection Parameters

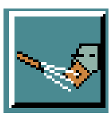
2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field. To do this,
   - Click the list button. The Tool Selector scroll list appears.
   - Choose a tool assembly from the list.
   - Click Accept.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.
5. Digitize a vector for the drive vector in the Drive Vector field. This vector specifies the direction for machining the groove.



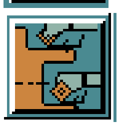
6. Choose the Feed Parameters option and enter the feed rates for the current task.



7. Choose the Spindle Speed Parameters option and enter the spindle speed parameters for the current task.



8. Choose the Coolant option and specify the coolant setting.



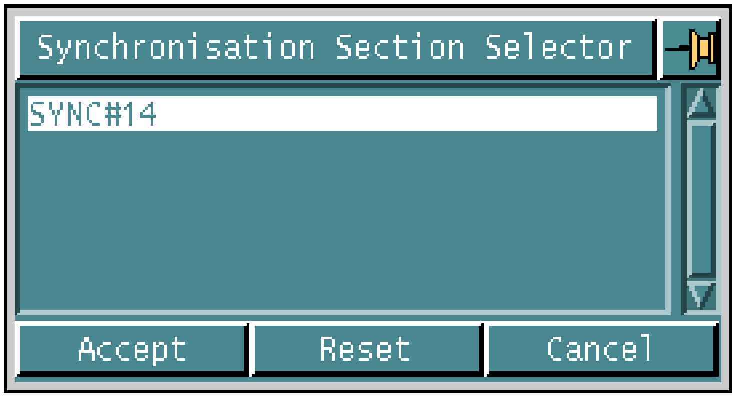
9. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

10. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 6-11 Synchronization Section Selector Scroll List

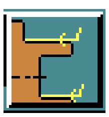


- Choose a synchronization section from the list.
- Click Accept.

OR

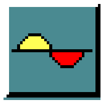
11. You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

OR



12. Choose the New Sync Section option to create a new synchronization section to add the task.
13. Click Apply if you have finished entering all parameters for the task.

Specifying Tolerance Parameters

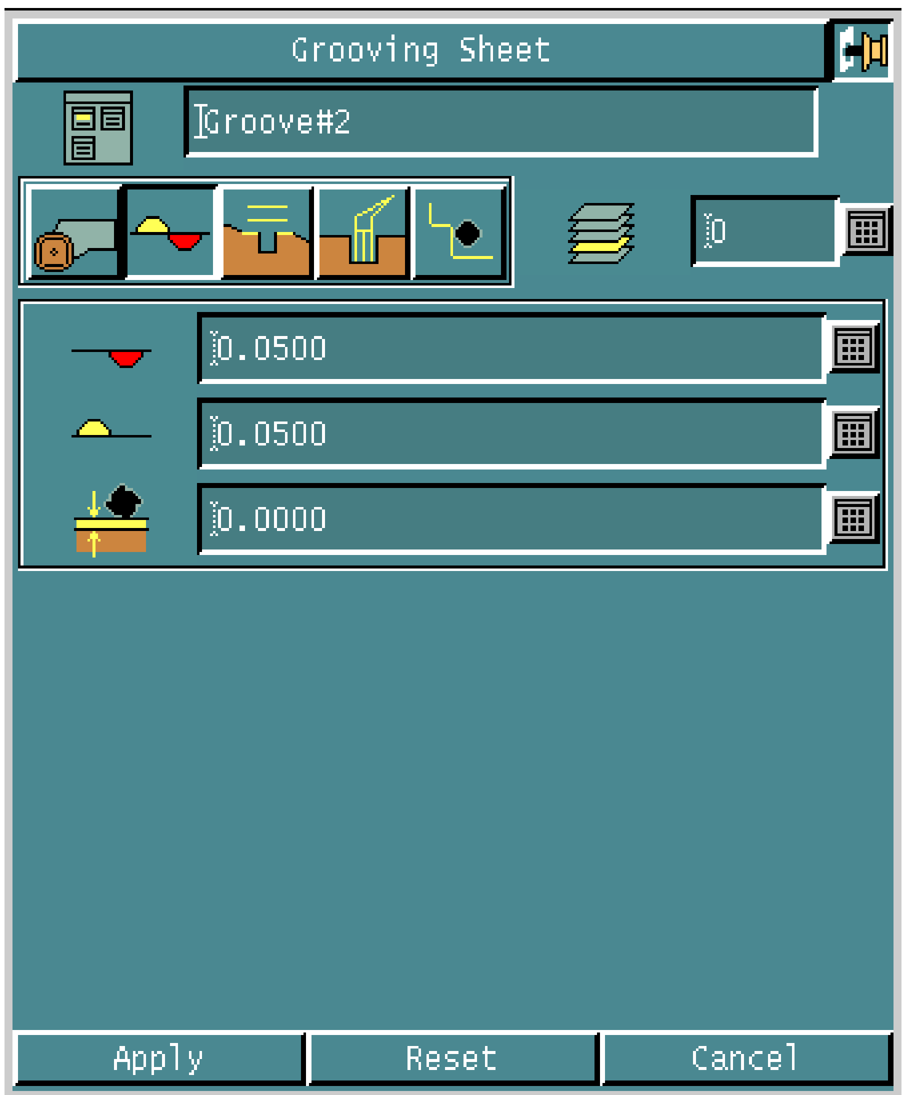


Use the TOLERANCE PARAMETERS option to specify tolerance and stock parameters for the current task.

Using this Option

1. Choose the Tolerance Parameters option from the grooving sheet. The set of parameters for tolerance appears on the sheet.

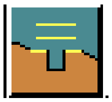
Figure 6-12 Grooving Sheet for Tolerance Parameters



2. Specify the inner tolerance for the current task in the Inner Tolerance field.

3. Specify the outer tolerance for the current task in the Outer Tolerance field.
4. Specify the stock offset for the current task in the Stock Offset field.
5. Click Apply if you have finished entering all parameters for the task.

Specifying Machining Parameters



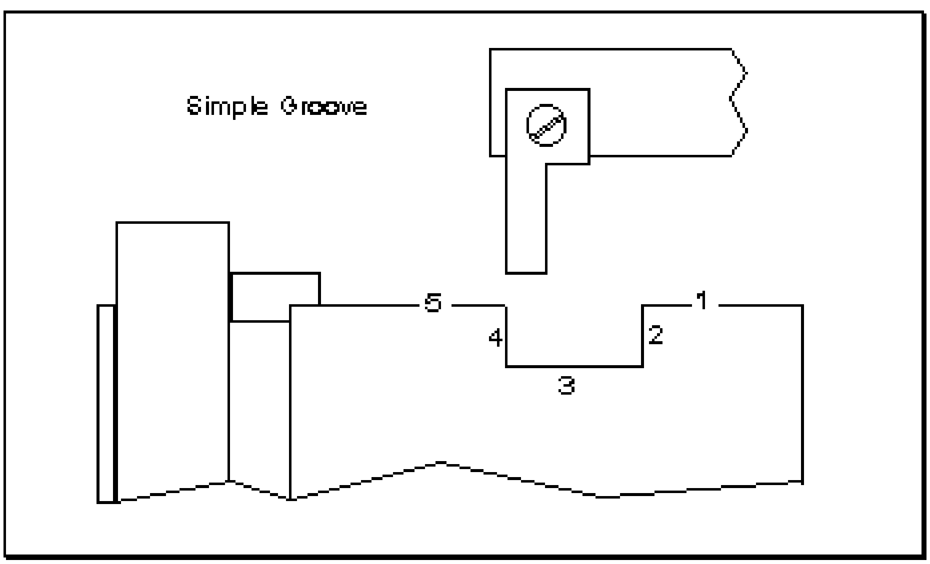
Use the MACHINING PARAMETERS option to specify machining parameters for the current task.

Start and End Entities

The start entity is the entity that immediately precedes the groove while the end entity is the entity that immediately follows the groove.

For grooving tasks you must specify start and end entities of the groove, in addition to the geometry. This is because the geometry you select from the Feature Selection sheet is the same for all tasks in the operation. Thus the geometry may contain a number of grooves. The start and entities isolate the current groove from the rest of the geometry. In the following figure, the start entity is labeled 1 and the end entity is labeled 5.

Figure 6-13 Start and End Entities

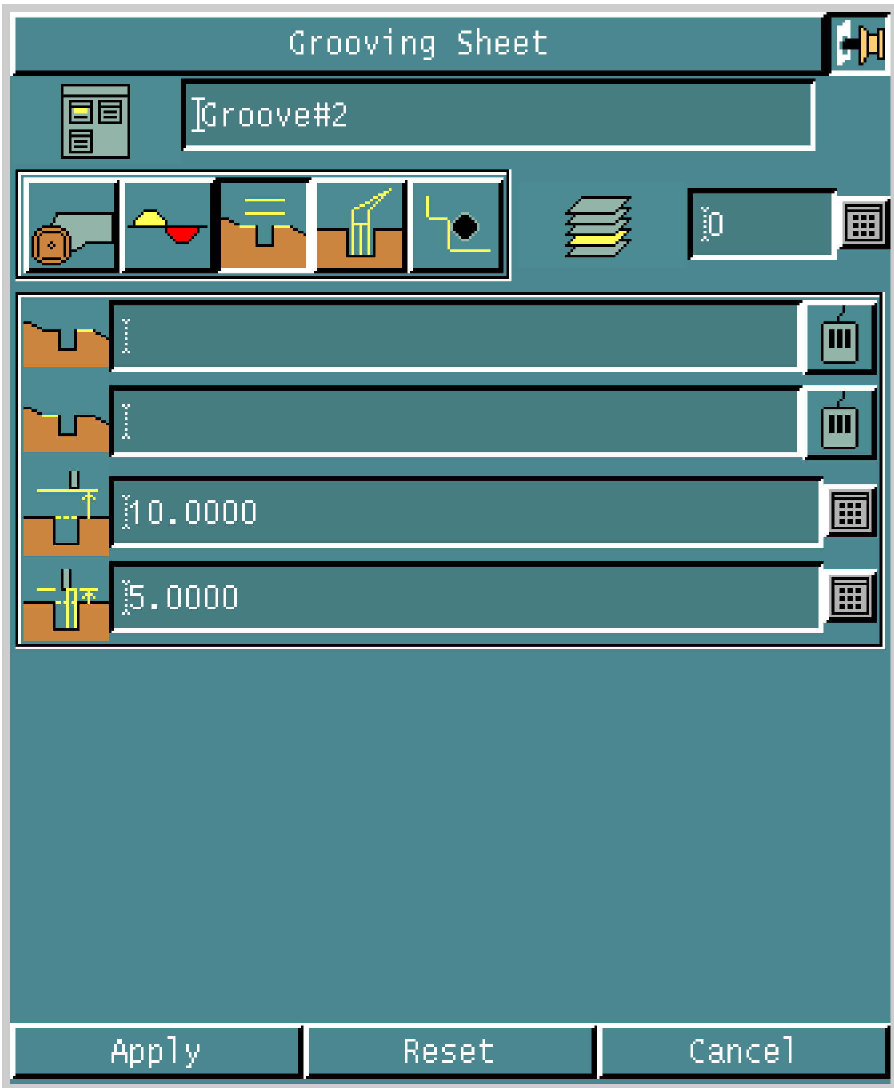


Thus, a groove can be defined as a set of entities obtained after removing the start and end entities in a feature.

Using this Option

1. Choose the Machining Parameters option from the grooving sheet. The set of machining parameters appears on the sheet.

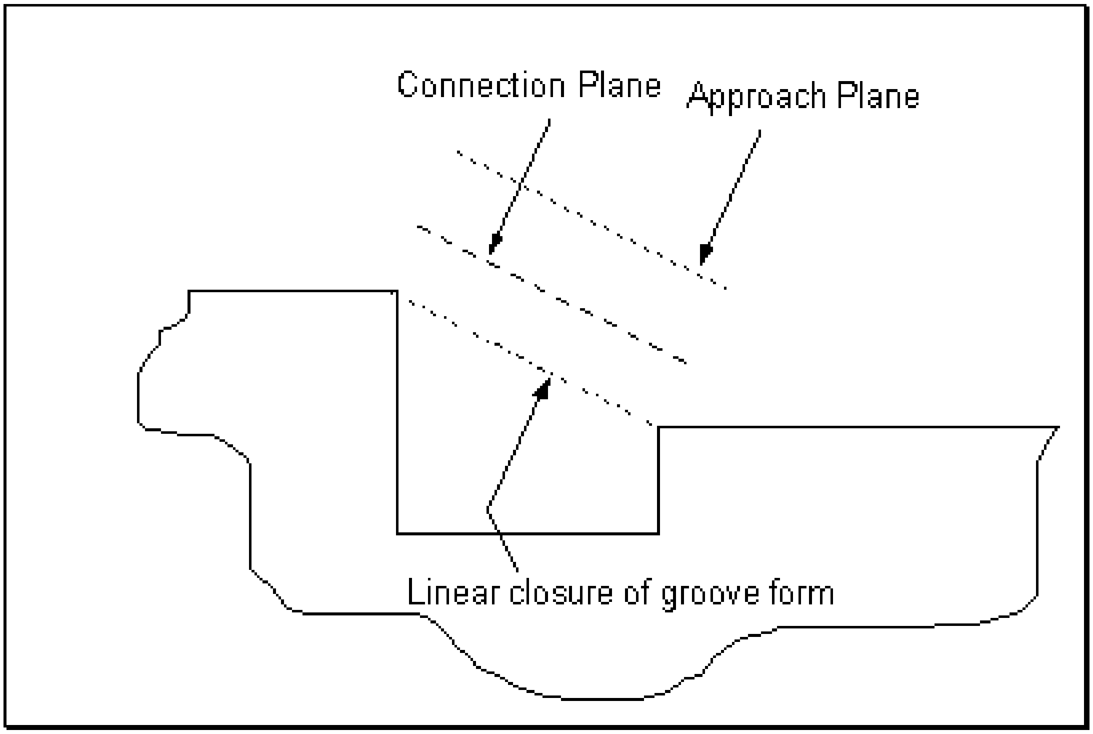
Figure 6-14 Grooving Sheet for Machining Parameters



2. Digitize the start entity for machining the groove.
3. Digitize the end entity for machining the groove.
4. Specify the offset for the approach plane in the Approach Plane Offset field.
5. Specify the offset for the connection plane in the Connection Plane Offset field. This is the plane along which the tool moves between multiple passes.

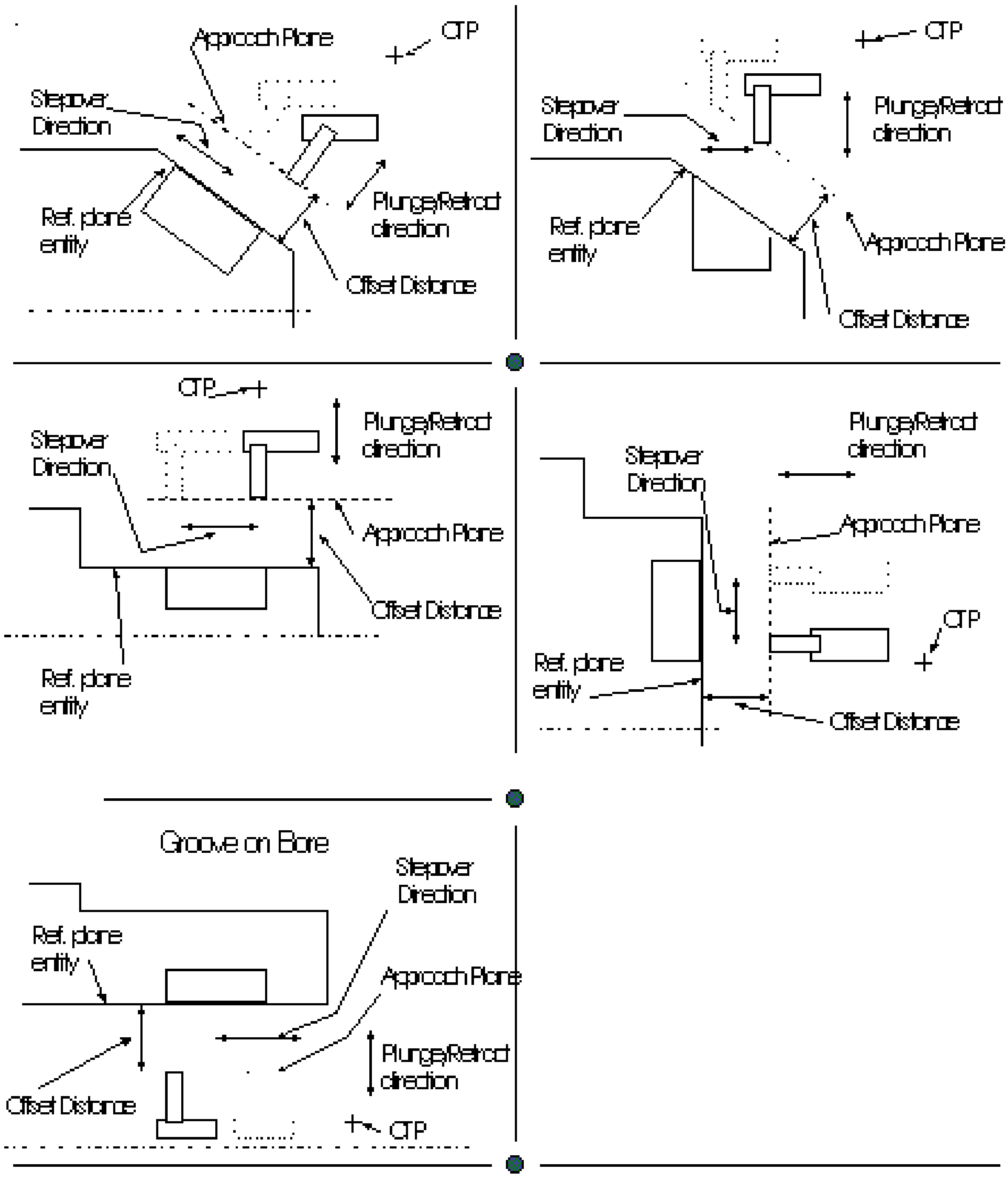
   The following figure shows the approach and connection planes:

Figure 6-15 Approach and Connection Planes



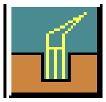
The following figure shows the relationship between different planes for various types of grooves:

Figure 6-16 Relationship between Planes for Various Grooves



6. Click Apply if you have finished entering all parameters for the task.

Specifying More Machining Parameters

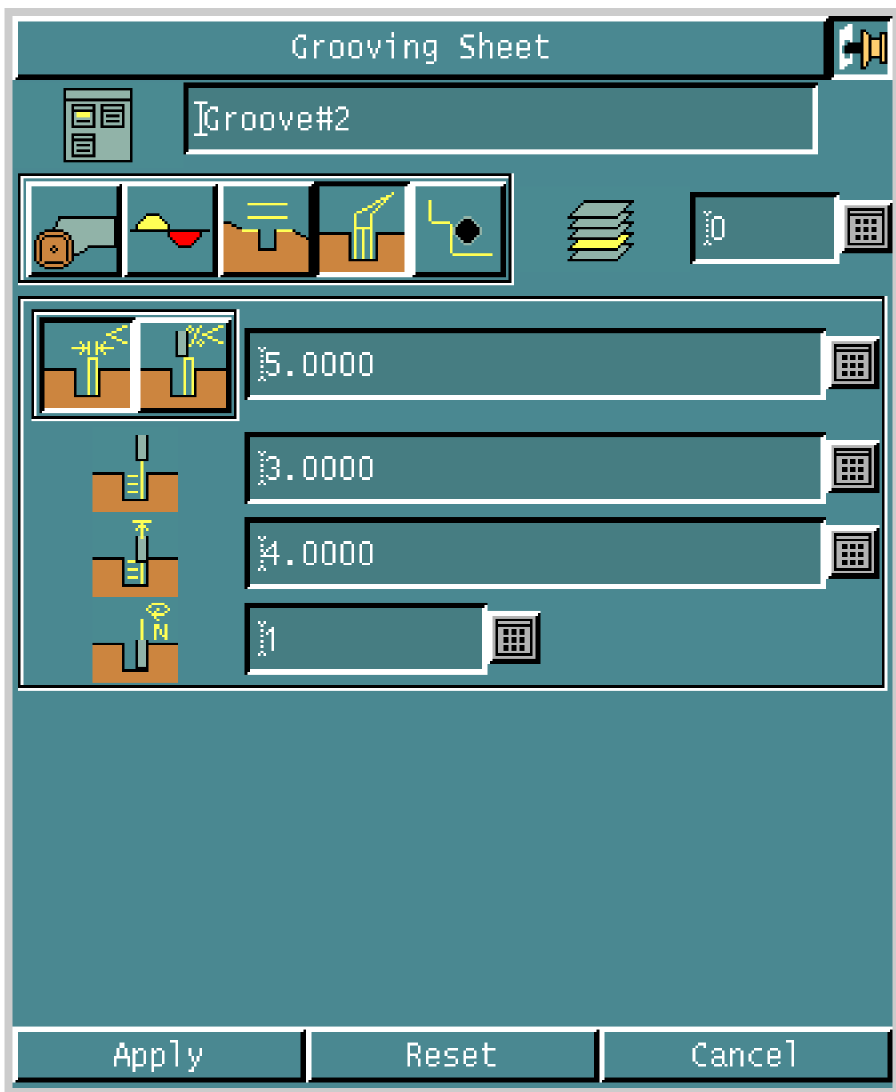


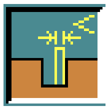
Use the MORE PARAMETERS option to specify more machining parameters for the current task.

Using this Option

1. Choose the More Parameters option from the grooving sheet. A set of more parameters appears on the sheet.

Figure 6-17 Grooving Sheet for General Parameters

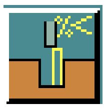




2. Choose the Max Step option and enter a value for the maximum stepover between the cuts.

Please note: If you enter a value greater than the current maximum tool width, CVNC uses the default value of the percentage of the tool width.

OR



3. Choose the Percent Tool Step option and specify a value for the maximum stepover in terms of a percentage of the maximum tool width. Enter this percentage in the field.
4. Specify the maximum peck depth in the Max Peck Depth field.
5. Specify the retract distance in the Retract Distance field.
6. Specify the number of revolutions for the dwell in the Dwell Rev field.
7. Click Apply if you have finished entering all parameters for the task.

Examples of Pecking Sequences for Various Grooves

Different types of grooves need different pecking sequences for machining. The following sections explain the various pecking sequences:

Centre Out for Width of Groove <= Toolwidth + Stock

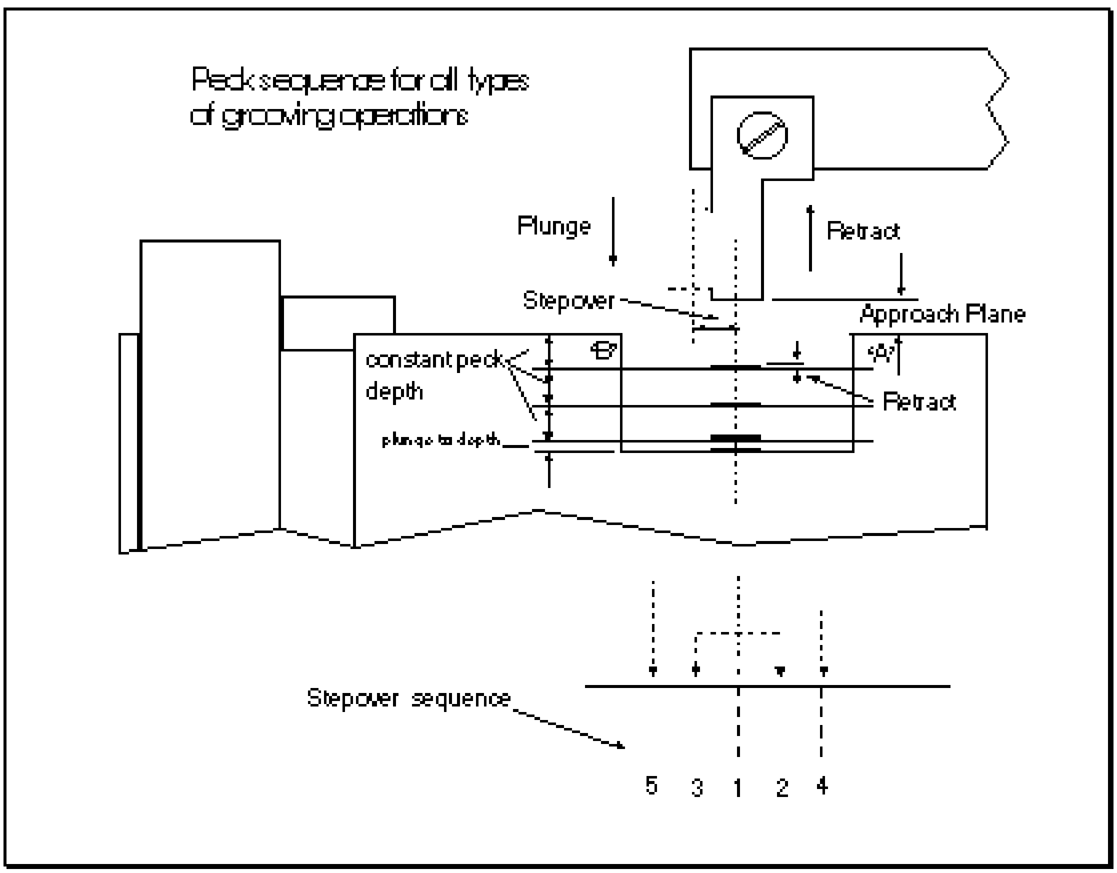
The pecking sequence for this type of groove is as follows:

1. Tool moves to the approach plane, such that the tool is centered on the center line of the groove.
2. Tool plunges into the groove up to the calculated peck depth, based on the maximum peck depth.
3. Tool retracts by the retract distance.
4. Tool plunges up to the calculated peck depth + retract distance.
5. Steps 3 and 4 are repeated till the material remaining at the base of the groove is less than the peck value or exactly equals the peck depth.
6. Tool plunges to the final cut depth. If you have specified a dwell, the tool dwells at this final position. While doing this, CVNC takes care of any stock values you have specified.
7. Tool retracts to the connection plane.

Centre Out for Width of Groove > Toolwidth + Stock

1. After the first cut, tool moves in the direction of A (please see the following figure) by the stepover value.
2. Repeat steps 2 to 7 from the previous section.
3. Tool moves in the direction of B using the stepover distance from the groove centre.
4. Repeat steps 2 to 7 from the previous section.
5. Repeat steps 1 to 4 till the full groove width is machined. During this, CVNC calculates the appropriate stepover. See the following figure for details.
6. Tool retracts to the approach plane.

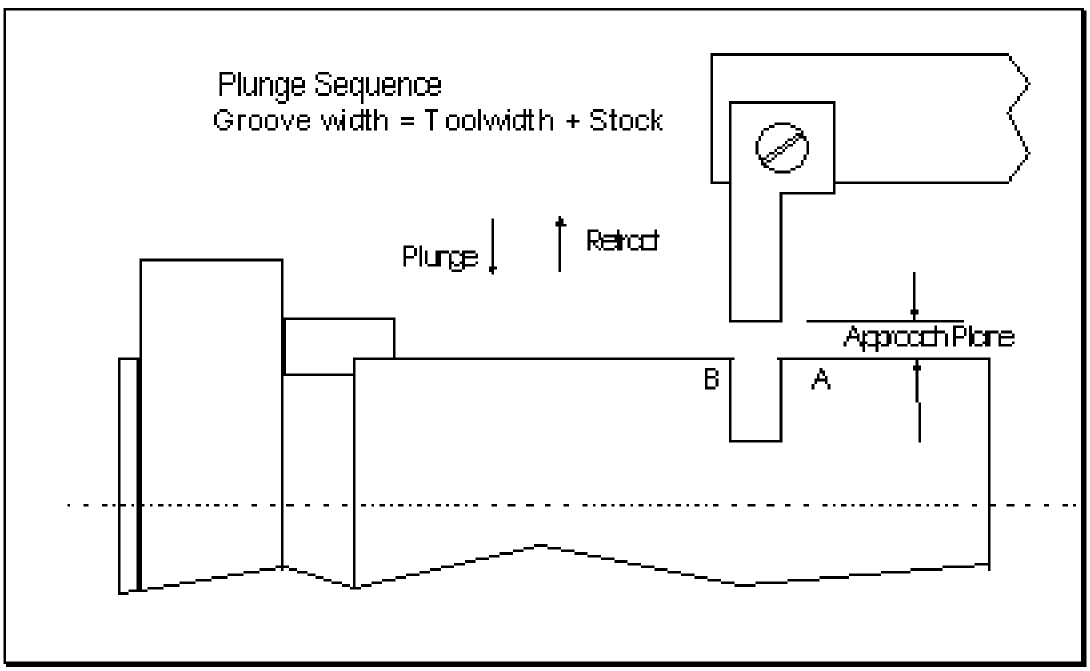
Figure 6-18 Peck Sequence for All Types of Grooving Operations



Single Plunge to Depth for Parallel Sided Grooves

1. Tool moves to approach plane with the primary tip positioned against the relevant boundary edge.
2. Tool plunges to the full depth of the groove.
3. Tool dwells at this position if you have specified a dwell.
4. Tool retracts to the approach plane.

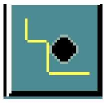
Figure 6-19 Plunge Sequence



Parallel Sided Grooves where Groove Width + Stock = Tool Width

The sequence is the same as “Centre Out for Width of Groove <= Toolwidth + Stock” on page 6-20.

Specifying Parameters for the Final Profile Pass

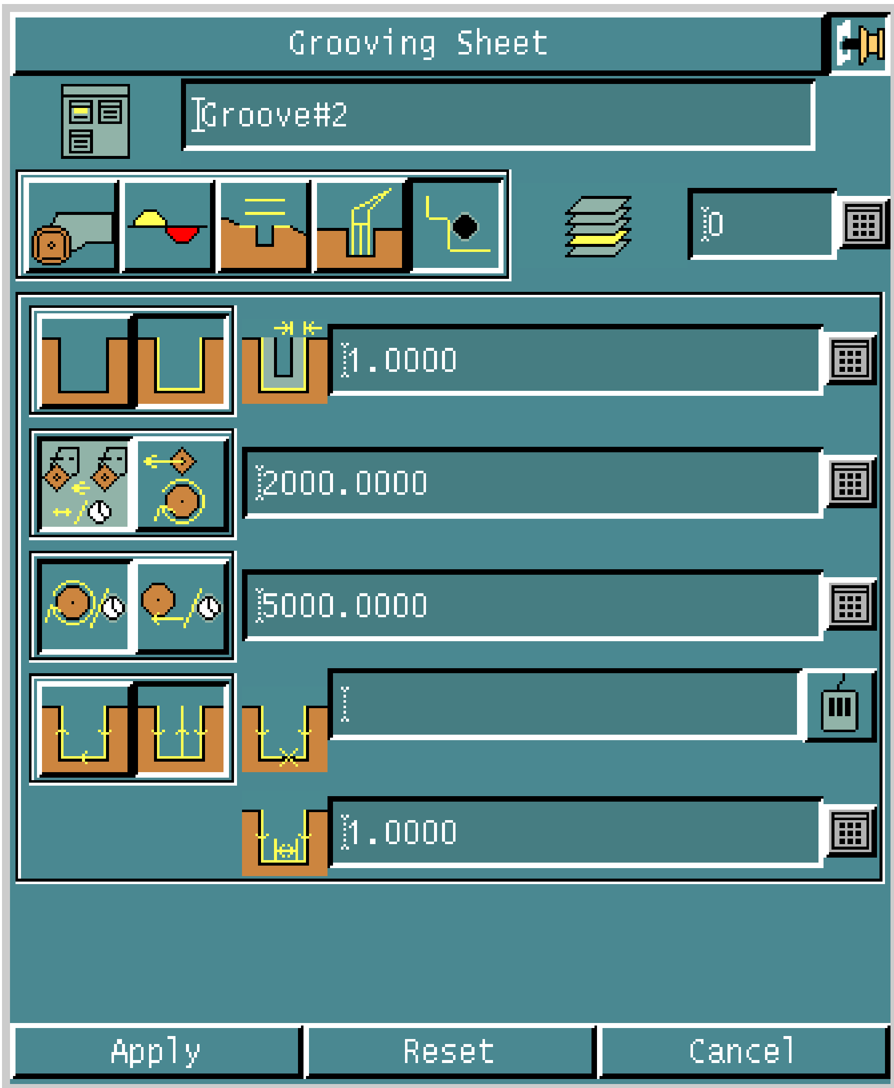


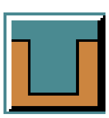
Use the PROFILE PARAMETERS option to specify parameters for the optional final profile pass.

Using this Option

1. Choose the Profile Parameters option from the grooving sheet. The set of parameters for the profile pass appears on the sheet.

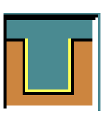
Figure 6-20 Grooving Sheet for Profile Parameters





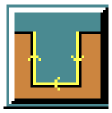
2. Choose the No Profile Pass option if you do not want a final profile pass. If you choose this option you will not see any parameters on the sheet.

OR



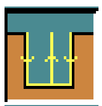
3. Choose the Profile Pass option to specify that you want a final profile pass. The parameters for the profile pass appear on the sheet as shown in the above figure.

4. Specify the stock for the profile pass in the Profile Allowance field.
5. Click and enter the feed rate for the profile pass. You can specify it in units of mm per min or mm per revolution.
6. Click and enter the spindle speed for the profile pass. You can specify it in revolutions per minute or surface meters per minute.



7. Choose the No Profile Split option if you want to profile the groove in a single pass. See Profiling a Groove in a Single Pass for details.

   OR



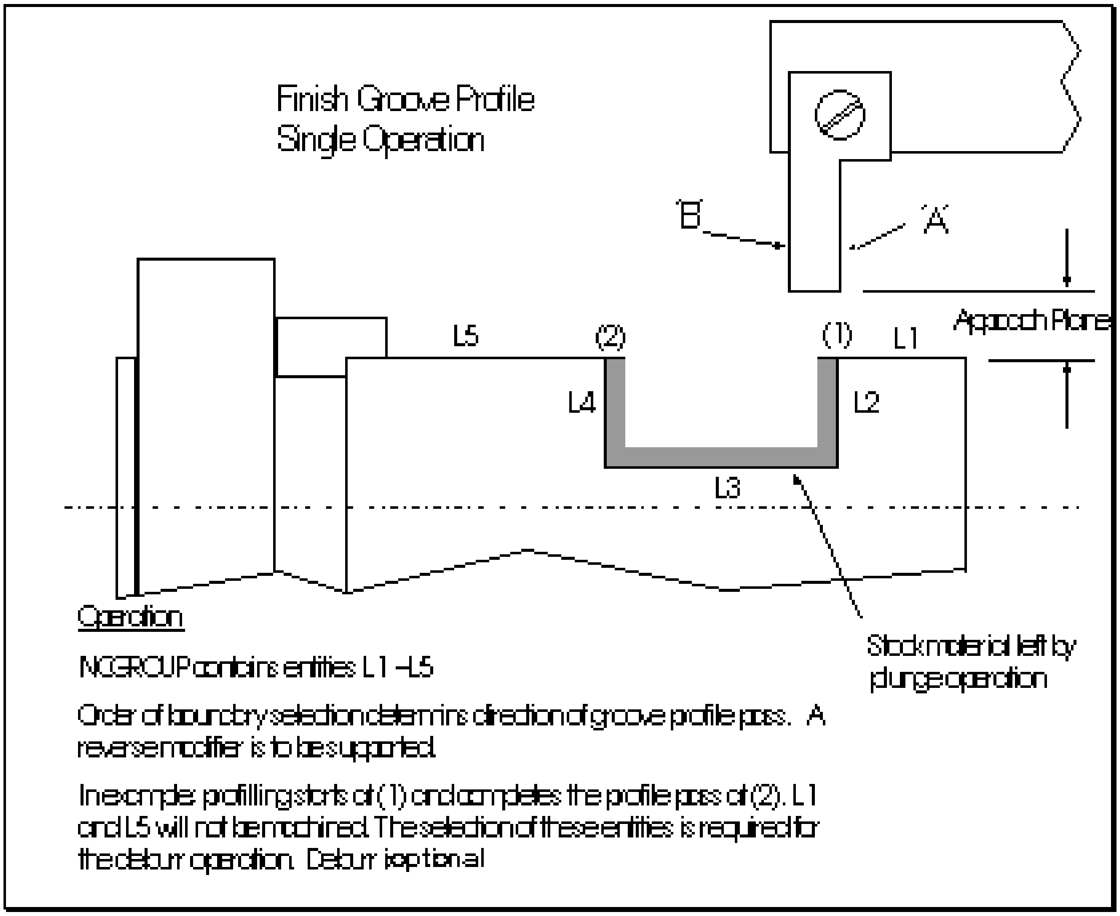
8. Choose the Profile Split option if you want to profile the groove in two passes. See Profiling a Groove in Two Passes for details. If you choose this option you have to specify the following:
   - Specify or digitize a split point to split the two profile passes.
   - Specify an overlap distance on either sides of the split point in the Overlap Distance field.
9. Click Apply if you have finished entering all parameters for the task.

Profiling a Groove in a Single Pass

The following sequence of operations takes place while profiling a groove in a single pass:

1. Tool moves to the approach plane and positions the active tip (A) as shown in Figure 6-21.
2. The complete groove is profiled.
3. Tool retracts to the approach plane.

Figure 6-21 Profiling a Groove in a Single Pass

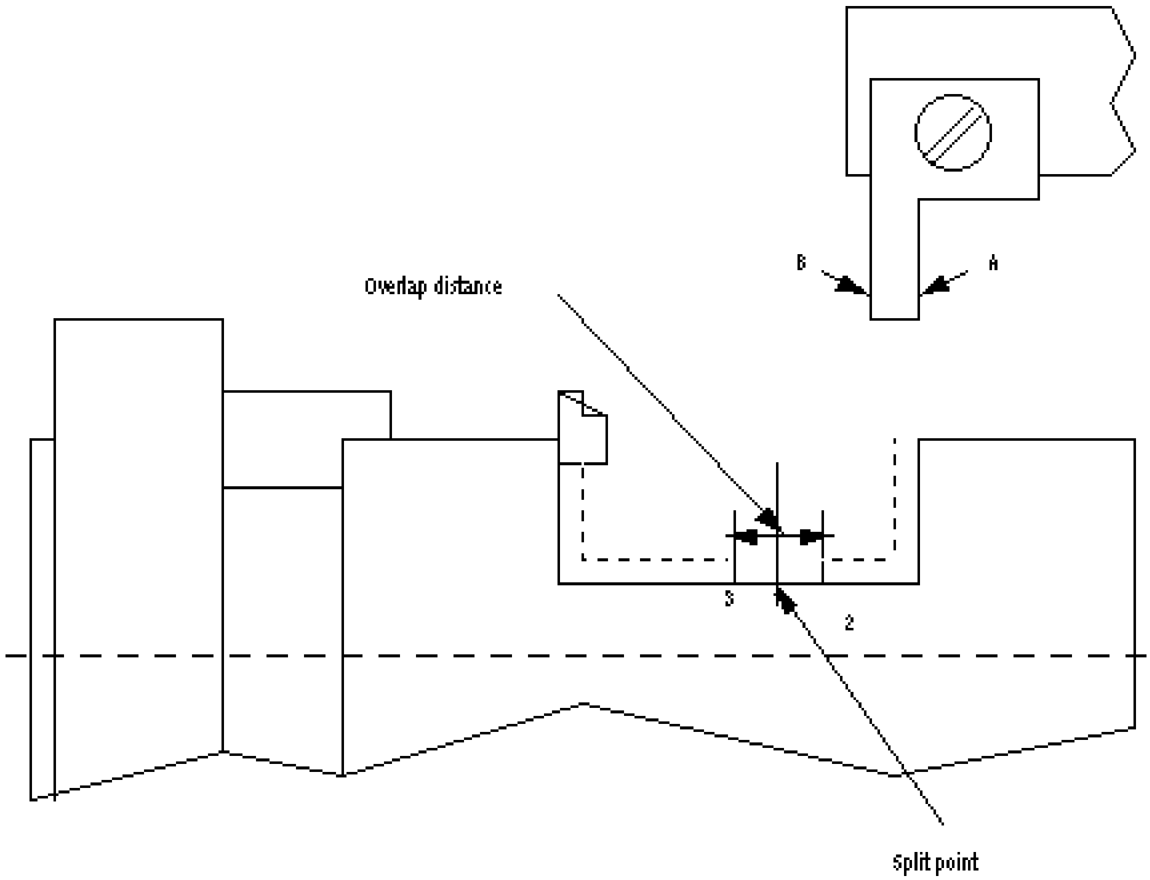


Profiling a Groove in Two Passes

The following sequence of operations takes place while profiling a groove in two passes:

1. Tool moves to the approach plane and positions the active tip (face A) as shown in Figure 6-22.
2. The groove is profiled up to point ‘3’ which is half the overlap distance from the split offset point ‘1’.
3. Tool retracts to the connection plane.
4. Tool moves along the connection plane until the tool tip (face B) is positioned to the opposing side of the groove boundary.
5. The groove is then profiled up to point ‘2’ which is half the overlap distance from the split offset point ‘1’.
6. Tool retracts to the approach plane.

Figure 6-22 Profiling a Groove in Two Passes



Executing the Job

You have to execute the task after specifying parameters for it. After you have clicked Apply on the grooving sheet, use the following steps for executing your job:

1. Click Apply on the operation sheet when you finish specifying all the tasks for your operation.
2. If you have used synchronization sections, you can see the Sync Master option in the NCBuilder sheet.

Figure 6-23 NCBuilder Sheet to Choose the Master Turret

SYNC MASTER OPTION

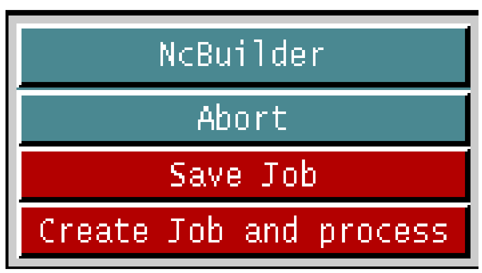
- Choose this option to select the master turret. The Synchronization Master Sheet appears.

Figure 6-24 Synchronization Master Sheet

- Choose the master turret for each synchronization section from the option list. For each synchronization section, CVNC uses feed rates and spindle speeds of the master turret.
- Click Apply.

3. Click Apply on the NCBuilder sheet after you finish specifying all the operations for your job. The following pulldown menu appears:

Figure 6-25 Pulldown Menu for Executing a Task



4. Click Create Job and process.

   You can now see the tool path and execution of your job.

Please note: If you have used synchronization sections, CVNC sorts the job control file (JCF) when you apply the job, such that there is minimum lead between turrets.

Chapter 7

Drilling Tasks

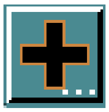
This chapter describes how you can use the NCBuilder Graphical User Interface (GUI) for drilling tasks on turned parts. You can use deep-hole drilling as well as peck or break chip drilling to process holes. The following sections describe the drilling process in detail. You can also refer to the *CVNC-T2 Command Reference* or the online help file to see the syntax of the DRILL command.

- Accessing the Drilling Options
- Specifying the Drilling Parameters
- Specifying Tool Assembly Selection Parameters
- Specifying Machining Parameters
- Specifying More Machining Parameters
- Executing the Job

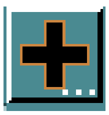
Accessing the Drilling Options

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.

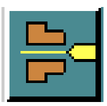
4. Choose the Job Block option from the NCBuilder task set. The NCBuilder property sheet appears.



5. Choose the Add Operation option from the NCBuilder property sheet. The Operation Sheet appears.

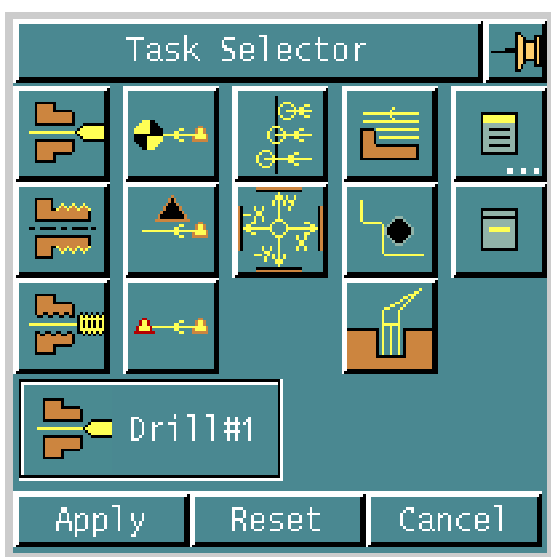


6. Choose the Add Task option from the operation sheet. The Task Selector palette appears.



7. Choose the Drill option from the Task Selector palette. The task name and its icon appear in the Task Selector palette as shown in the following figure. The sequence number of the task also appears. For example, if the task is the first drilling task in the operation, it has the number 1.

Figure 7-1 Task Selector Palette with Task Name and Icon



Please note: You should define the machine tool parameters and tooling parameters before specifying any task. See Chapter 2, “Machine Tool Definitions” for details.

8. Click Apply.

   The task now appears in the operation sheet.

9. Specify the geometry as described in “Specifying Geometry” on page 5-6.

Please note: This task does not require part geometry. However, you may specify geometry for the material boundary.

10. Specify the feed rates for the operation as described in “Specifying the Feed Rates for an Operation” on page 5-10.

11. Specify the spindle speed for the operation as described in “Specifying the Spindle Speed for an Operation” on page 5-13.

12. Specify the coolant setting for the operation as described in “Specifying the Coolant Setting for an Operation” on page 5-14.

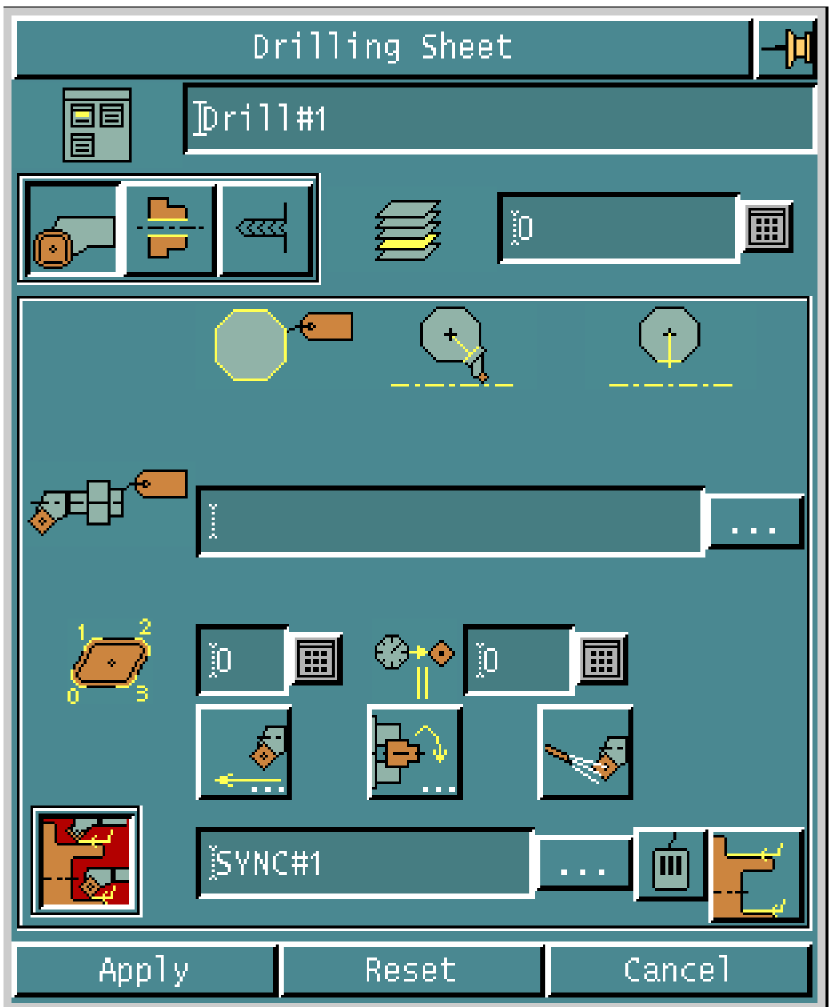
Specifying the Drilling Parameters

Use the EDIT option on the operation sheet to specify parameters for the drilling process.

Using this Option

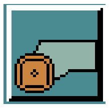
1. Highlight the task and choose the Edit option on the operation sheet. The Drilling Sheet appears.

Figure 7-2 Drilling Sheet



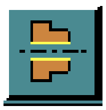
2. Specify the layer for creating the tool path in the Layer field.

Options on the Drilling Sheet



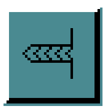
SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 7-6 for details.



SPECIFYING MACHINING PARAMETERS
Displays a property sheet that enables you to specify machining parameters for the current task.

See “Specifying Machining Parameters” on page 7-9 for details.

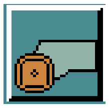


SPECIFYING MORE MACHINING PARAMETERS
Displays a property sheet that enables you to specify more machining parameters for the current task.

See “Specifying More Machining Parameters” on page 7-12 for details.

Please note: For each of these options, a different set of parameters appears on the same drilling sheet.

Specifying Tool Assembly Selection Parameters

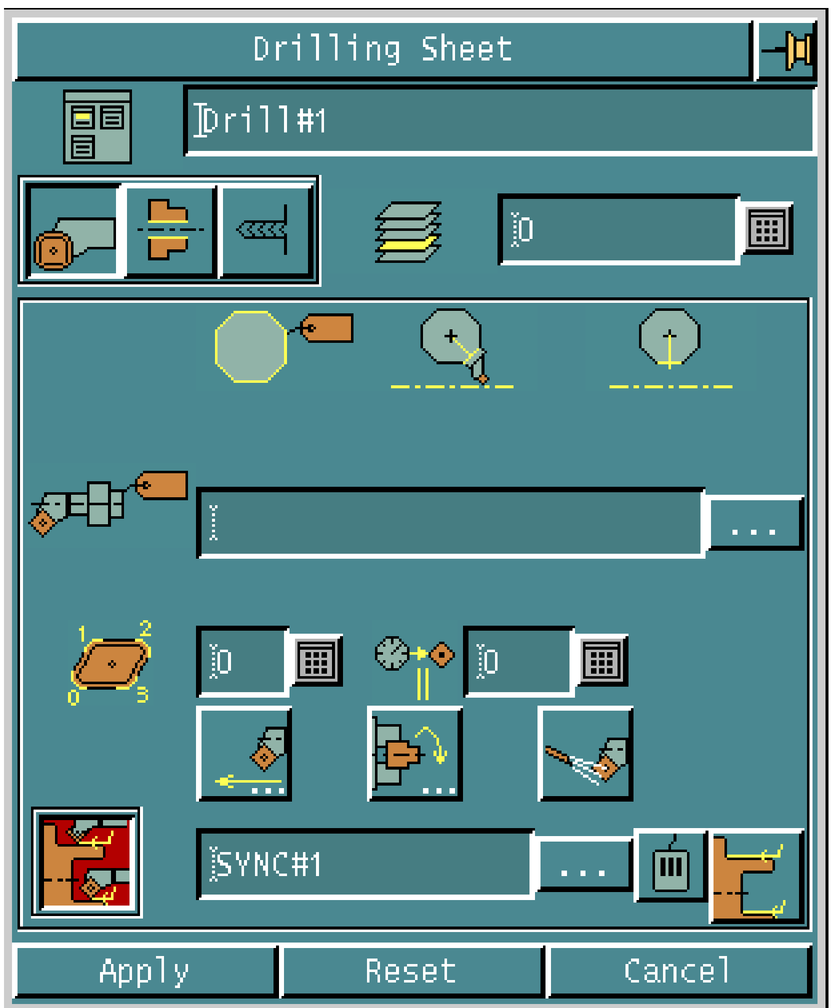


Use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

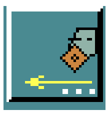
Using this Option

1. Choose the Tool Parameters option from the drilling sheet. The set of parameters for the tool assembly appears on the sheet.

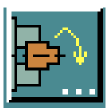
Figure 7-3 Drilling Sheet for Tool Assembly Selection Parameters



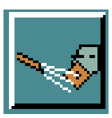
2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field. To do this,
   - Click the list button. The Tool Selector scroll list appears.
   - Choose a tool assembly from the list.
   - Click Accept.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



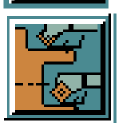
5. Choose the Feed Parameters option and enter the feed rates for the current task.



6. Choose the Spindle Speed Parameters option and enter the spindle speed parameters for the current task.



7. Choose the Coolant option and specify the coolant setting.



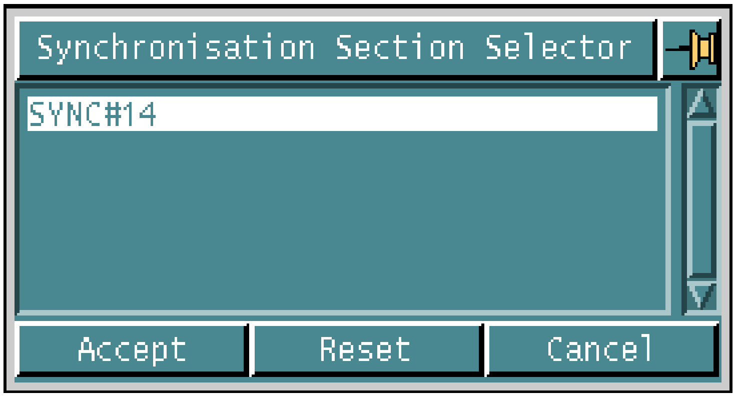
8. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

9. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 7-4 Synchronization Section Selector Scroll List

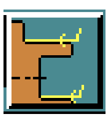


- Choose a synchronization section from the list.
- Click Accept.

OR

10. You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

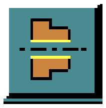
OR



11. Choose the New Sync Section option to create a new synchronization section to add the task.

12. Click Apply if you have finished entering all parameters for the task.

Specifying Machining Parameters

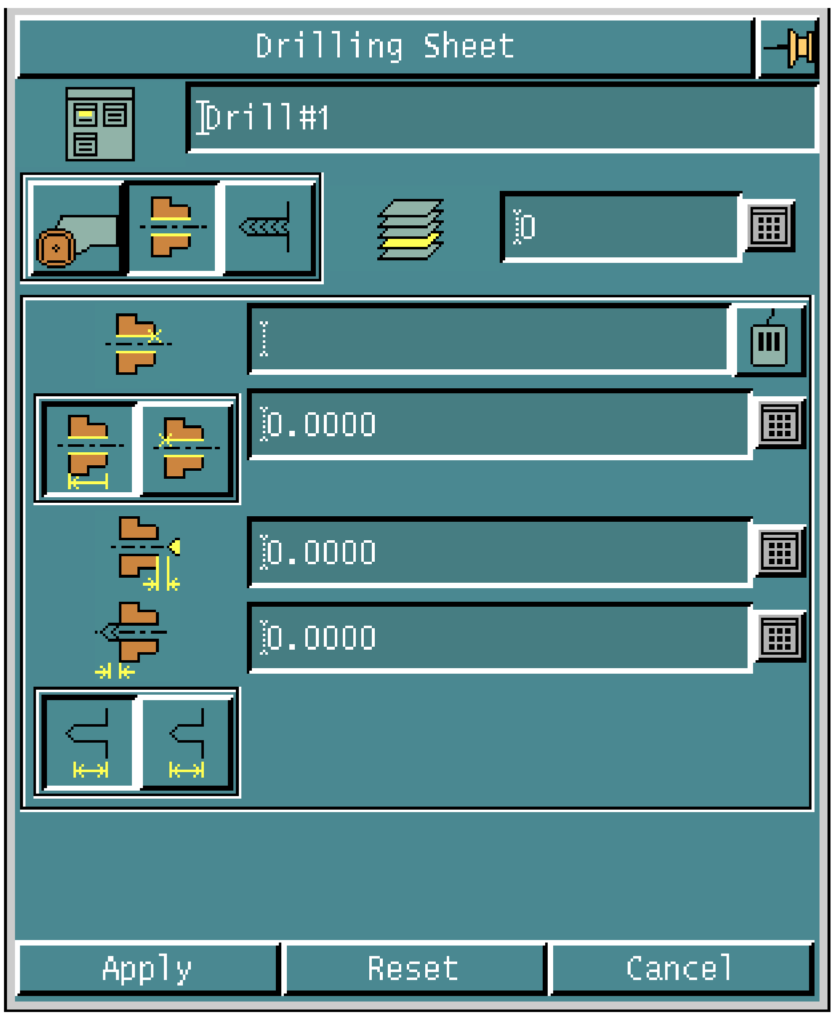


Use the MACHINING PARAMETERS option to specify machining parameters for the current task.

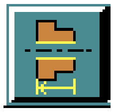
Using this Option

1. Choose the Machining Parameters option from the drilling sheet. The set of machining parameters appears on the sheet.

Figure 7-5 Drilling Sheet for Machining Parameters

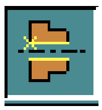


2. Digitize the start location for drilling the hole.



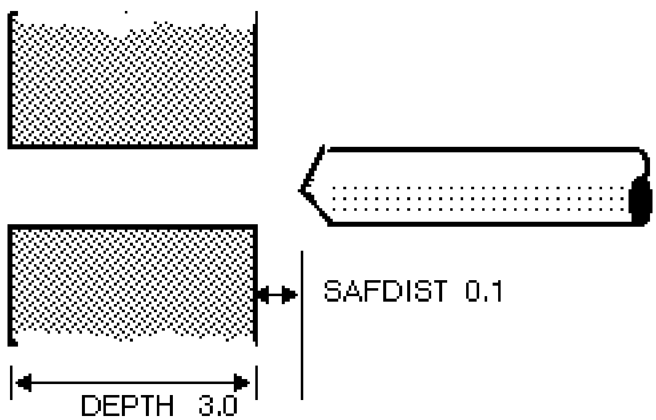
3. Choose the Depth Distance option and enter the depth of the hole in the Hole Depth field.

   OR



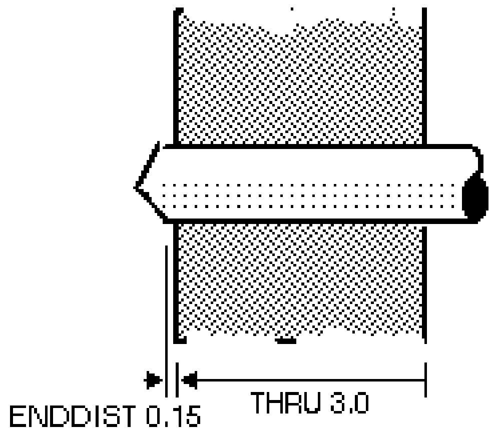
4. Choose the Depth Loc option and digitize a location for the depth of the hole.
5. Specify the value to be added to the start location of the hole for approaches and retracts in the Safe Distance field. See the following figure for details.

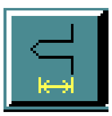
Figure 7-6 Safe Distance



6. Specify the value to be added to the depth of the hole in the End Distance field. See the following figure for details.

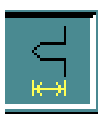
Figure 7-7 End Distance





7. Choose the `Angle` option if you want the angle of the tool tip to be considered while calculating the drill depth.

   OR

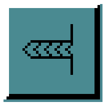


8. Choose the `No Angle` option if you do not want the angle of the tool tip to be considered while calculating the drill depth.

   OR

9. Click `Apply` if you have finished entering all parameters for the task.

Specifying More Machining Parameters

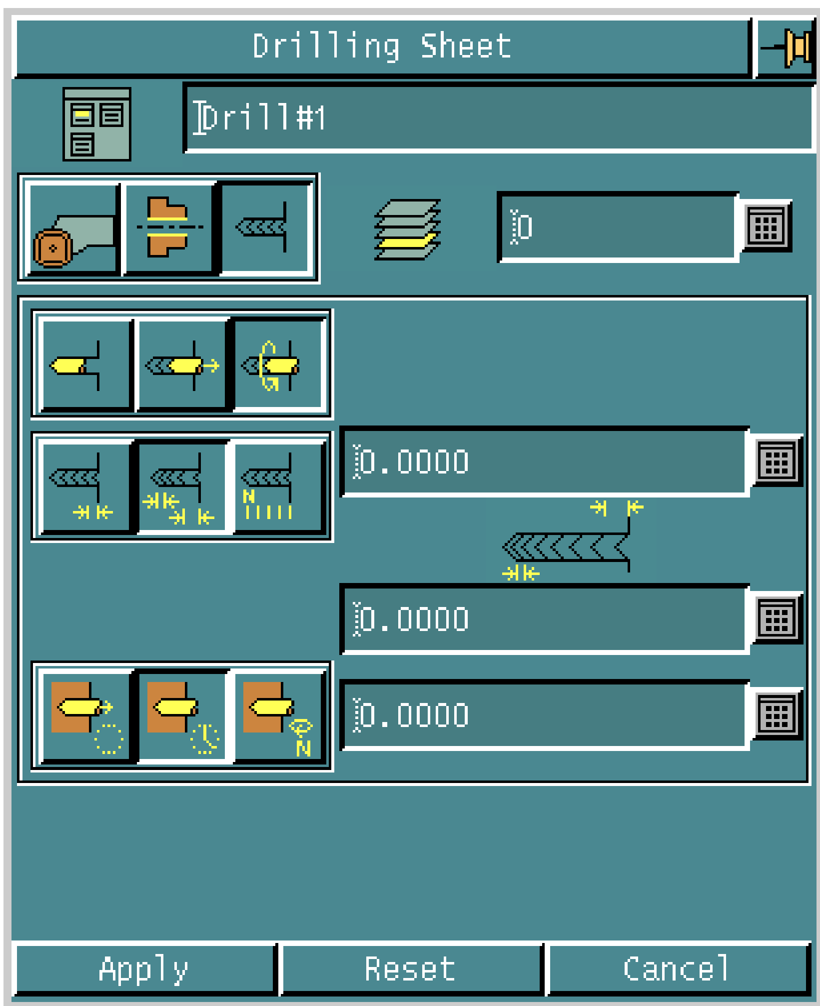


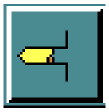
Use the MORE PARAMETERS option to specify more machining parameters for the current task.

Using this Option

1. Choose the More Parameters option from the drilling sheet. A set of more parameters appears on the sheet.

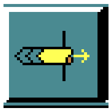
Figure 7-8 Drilling Sheet for General Parameters





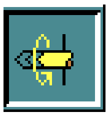
2. Choose the Default Drill option for direct drilling to the depth required.

   OR

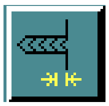


3. Choose the Peck Drilling option for peck (deep-hole) drilling.

   OR

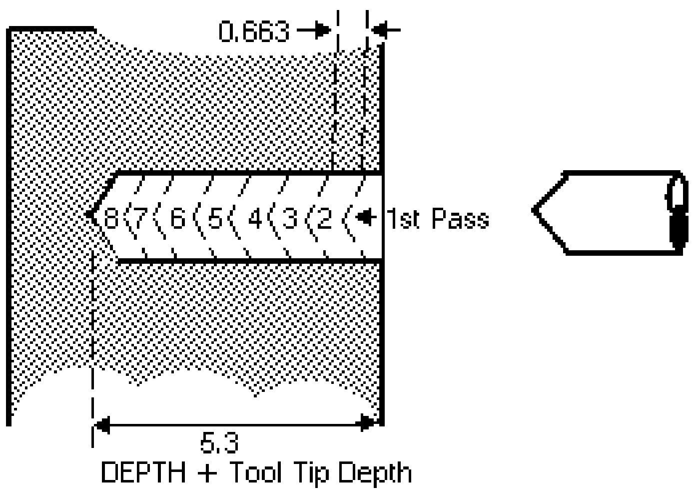


4. Choose the Break Drilling option for break-chip (pause) drilling.

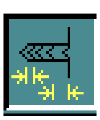


5. Choose the Maximum Depth option and enter the maximum depth of each cut. CVNC then calculates the number of passes at depths equal to or less than this maximum depth. This option is not required for the default drilling mode. See the following figure for details.

Figure 7-9 Maximum Depth of Cuts



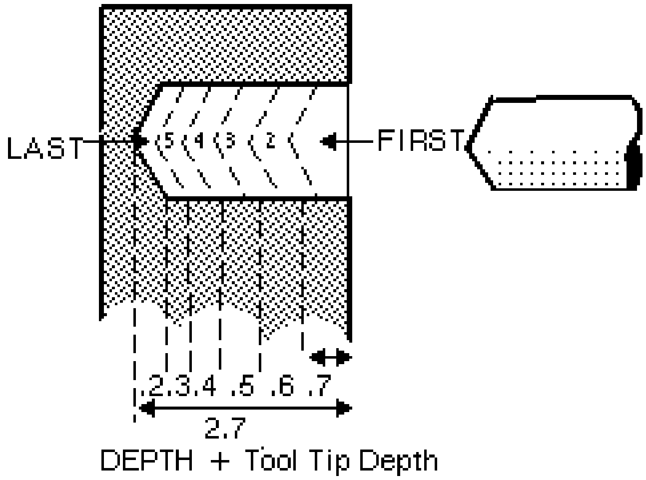
   OR



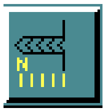
6. Choose the First And Last option and enter the depths of the first and last tool passes. CVNC then calculates linearly decreasing depths for each intermediate pass. This option is not required for the default drilling mode. See the following figure for details.

Please note: You should enter a greater depth for the first pass.

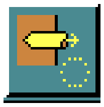
Figure 7-10 Depths of First and Last Cuts



OR

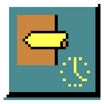


7. Choose the Number of Passes option and enter the number of passes. CVNC then calculates an equal cut depth for each pass. This option is not required for the default drilling mode.



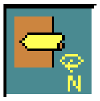
8. Choose the No Dwell option if you do not want the tool to dwell after it has reached the specified depth.

   OR



9. Choose the Dwell Time option and enter the dwell time in seconds.

   OR



10. Choose the Dwell Revs option and enter the number of revolutions for the dwell.
11. Click Apply if you have finished entering all parameters for the task.

Executing the Job

You have to execute the task after specifying parameters for it. After you have clicked Apply on the drilling sheet, use the following steps for executing your job:

1. Click Apply on the operation sheet when you finish specifying all the tasks for your operation.
2. If you have used synchronization sections, you can see the Sync Master option in the NCBuilder sheet.

Figure 7-11 NCBuilder Sheet to Choose the Master Turret

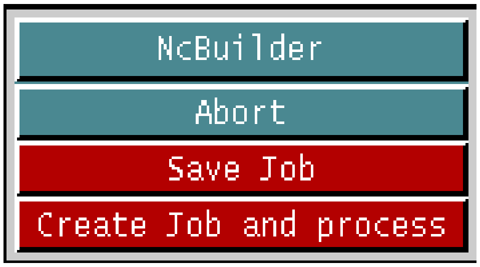
- Choose this option to select the master turret. The Synchronization Master Sheet appears.

Figure 7-12 Synchronization Master Sheet

- Choose the master turret for each synchronization section from the option list. For each synchronization section, CVNC uses feed rates and spindle speeds of the master turret.
- Click Apply.

3. Click Apply on the NCBuilder sheet after you finish specifying all the operations for your job. The following pulldown menu appears:

Figure 7-13 Pulldown Menu for Executing a Task



4. Click Create Job and process.

   You can now see the tool path and execution of your job.

Please note: If you have used synchronization sections, CVNC sorts the job control file (JCF) when you apply the job, such that there is minimum lead between turrets.

Chapter 8

Threading Tasks

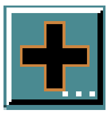
This chapter describes how you can use the NCBuilder Graphical User Interface (GUI) for threading tasks on turned parts. The following sections describe the threading process in detail. You can also refer to the *CVNC-T2 Command Reference* or the online help file to see the syntax of the THREAD command.

- Accessing the Threading Options
- Specifying the Threading Parameters
- Specifying Tool Assembly Selection Parameters
- Specifying Machining Parameters
- Specifying More Machining Parameters
- Specifying Further Machining Parameters
- Executing the Job

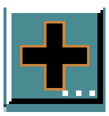
Accessing the Threading Options

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.

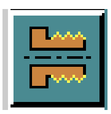
4. Choose the Job Block option from the NCBuilder task set. The NCBuilder property sheet appears.



5. Choose the Add Operation option from the NCBuilder property sheet. The Operation Sheet appears.

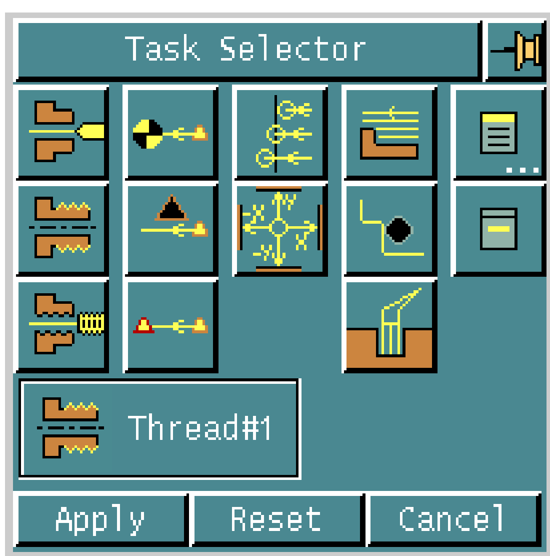


6. Choose the Add Task option from the operation sheet. The Task Selector palette appears.



7. Choose the Thread option from the Task Selector palette. The task name and its icon appear in the Task Selector palette as shown in the following figure. The sequence number of the task also appears. For example, if the task is the first threading task in the operation, it has the number 1.

Figure 8-1 Task Selector Palette with Task Name and Icon



Please note: You should define the machine tool parameters and tooling parameters before specifying any task. See Chapter 2, “Machine Tool Definitions” for details.

8. Click Apply.

   The task now appears in the operation sheet.

9. Specify the geometry as described in “Specifying Geometry” on page 5-6.

Please note: This task does not require part geometry. However, you may specify geometry for the material boundary.

10. Specify the feed rates for the operation as described in “Specifying the Feed Rates for an Operation” on page 5-10.
11. Specify the spindle speed for the operation as described in “Specifying the Spindle Speed for an Operation” on page 5-13.
12. Specify the coolant setting for the operation as described in “Specifying the Coolant Setting for an Operation” on page 5-14.

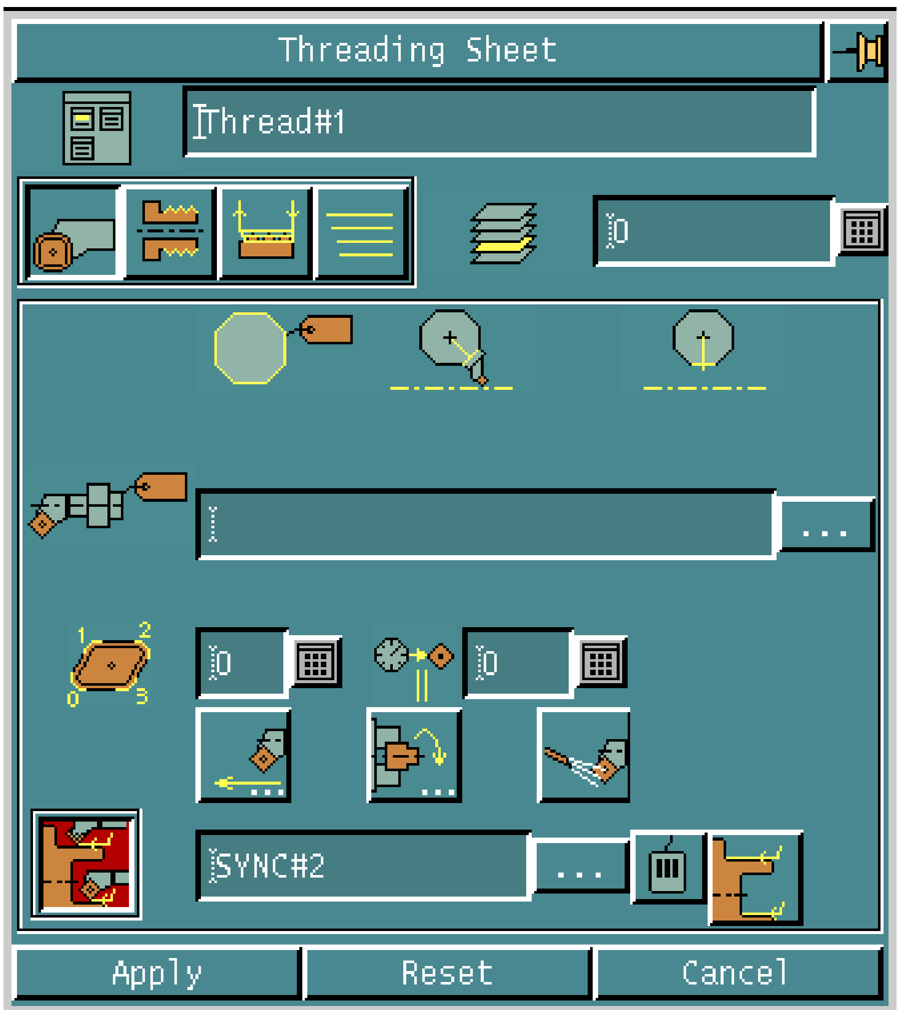
Specifying the Threading Parameters

Use the EDIT option on the operation sheet to specify parameters for the threading process.

Using this Option

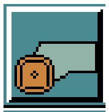
1. Highlight the task and choose the Edit option on the operation sheet. The Threading Sheet appears.

Figure 8-2 Threading Sheet



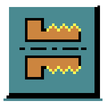
2. Specify the layer for creating the tool path in the Layer field.

Options on the Threading Sheet



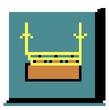
SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 8-6 for details.



SPECIFYING MACHINING PARAMETERS
Displays a property sheet that enables you to specify machining parameters for the current task.

See “Specifying Machining Parameters” on page 8-9 for details.



SPECIFYING MORE MACHINING PARAMETERS
Displays a property sheet that enables you to specify more machining parameters for the current task.

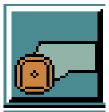
See “Specifying More Machining Parameters” on page 8-12 for details.

SPECIFYING FURTHER MACHINING PARAMETERS
Displays a property sheet that enables you to specify further machining parameters for the current task.

See “Specifying Further Machining Parameters” on page 8-15 for details.

Please note: For each of these options, a different set of parameters appears on the same threading sheet.

Specifying Tool Assembly Selection Parameters

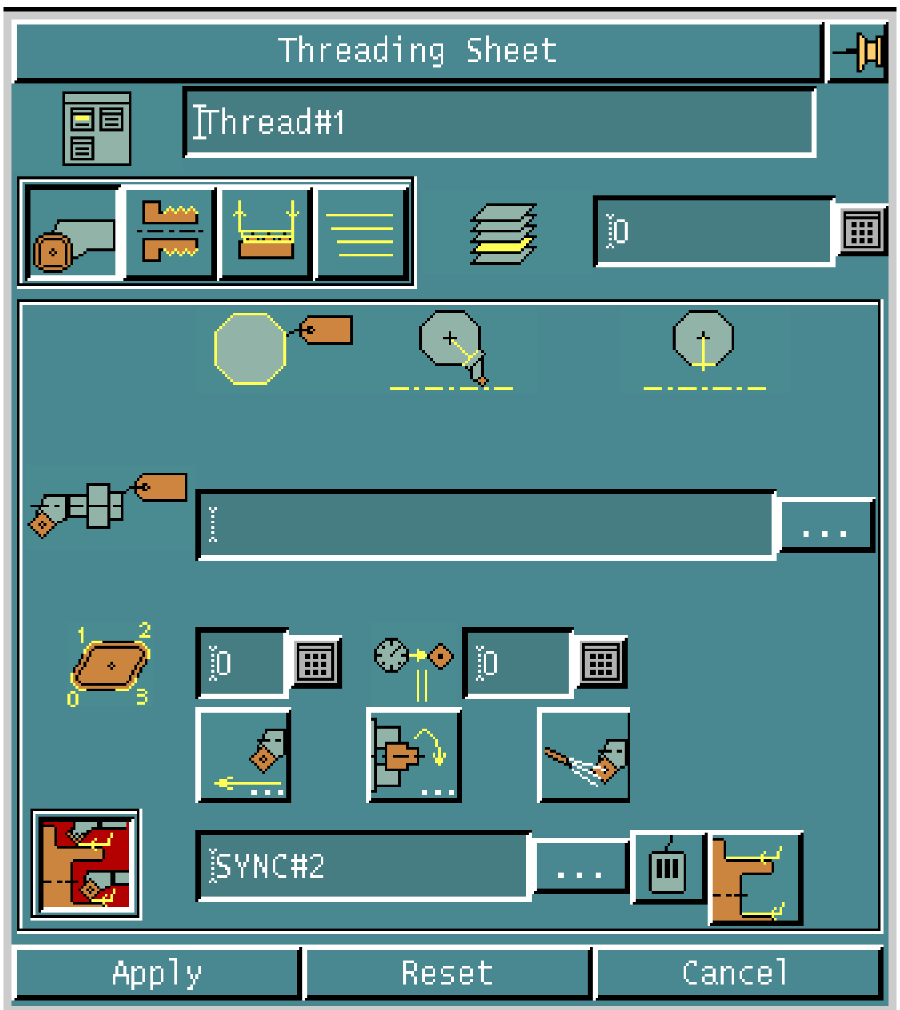


Use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

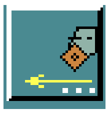
Using this Option

1. Choose the Tool Parameters option from the threading sheet. The set of parameters for the tool assembly appears on the sheet.

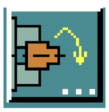
Figure 8-3 Threading Sheet for Tool Assembly Selection Parameters



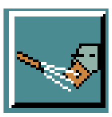
2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field. To do this,
   - Click the list button. The Tool Selector scroll list appears.
   - Choose a tool assembly from the list.
   - Click Accept.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



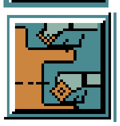
5. Choose the Feed Parameters option and enter the feed rates for the current task.



6. Choose the Spindle Speed Parameters option and enter the spindle speed parameters for the current task.



7. Choose the Coolant option and specify the coolant setting.



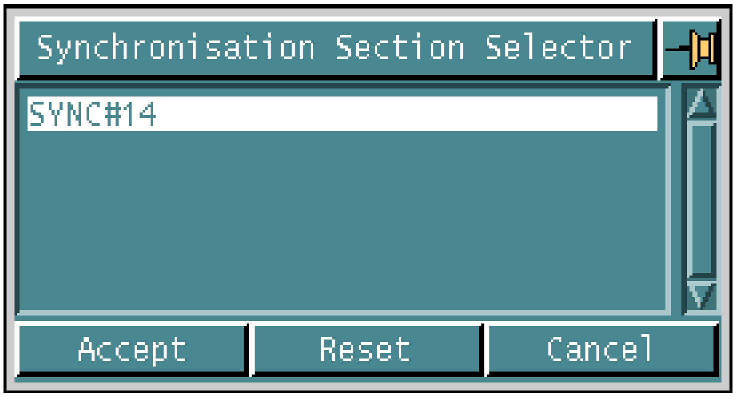
8. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

9. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 8-4 Synchronization Section Selector Scroll List

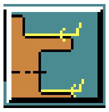


- Choose a synchronization section from the list.
- Click Accept.

OR

10. You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

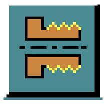
OR



11. Choose the New Sync Section option to create a new synchronization section to add the task.

12. Click Apply if you have finished entering all parameters for the task.

Specifying Machining Parameters

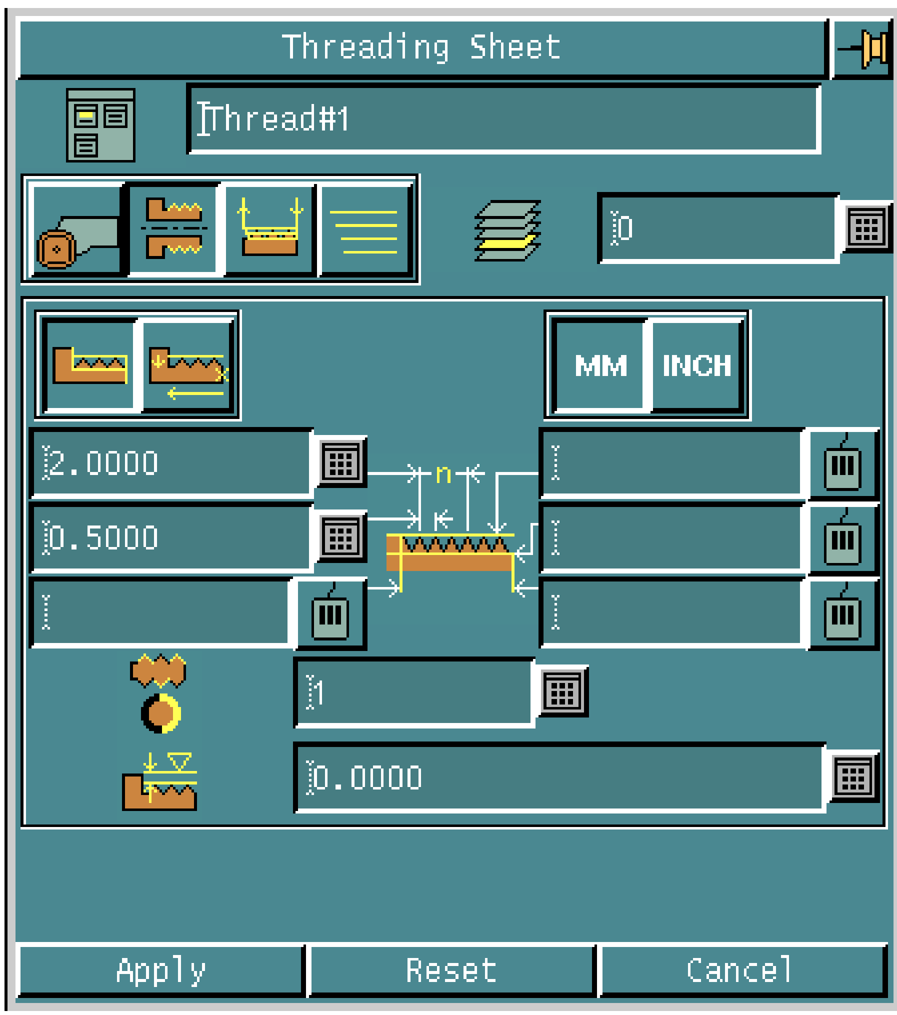


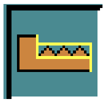
Use the MACHINING PARAMETERS option to specify machining parameters for the current task.

Using this Option

1. Choose the Machining Parameters option from the threading sheet. The set of machining parameters appears on the sheet.

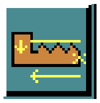
Figure 8-5 Threading Sheet for Machining Parameters





2. Choose the Thread Data by Entity option to enter the data for threading using entities.

   OR



3. Choose the Thread Data by Loc option to enter the data for threading using digitized locations.

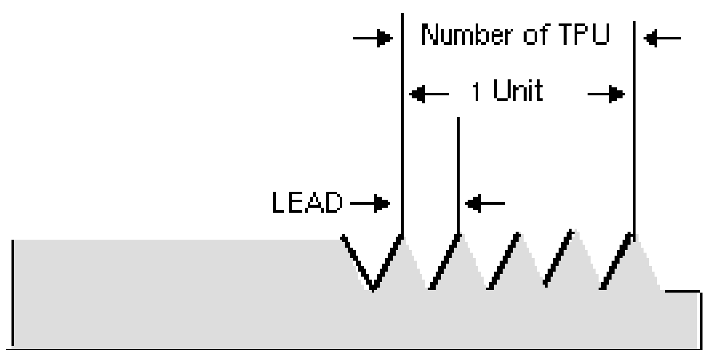
4. Choose the Mm option to specify the units in millimeters.

   OR

5. Choose the Inch option to specify the units as inches.
6. Specify the number of threads per unit in the Threads per Unit field.
7. Digitize the material entity.
8. Specify the distance between corresponding points on consecutive threads in the Lead field. This value is related to the Threads per Unit value and each changes when the other is changed.

   See the following figure for the relationship between the Lead and Threads per Unit values.

Figure 8-6 Threads per Unit and Lead

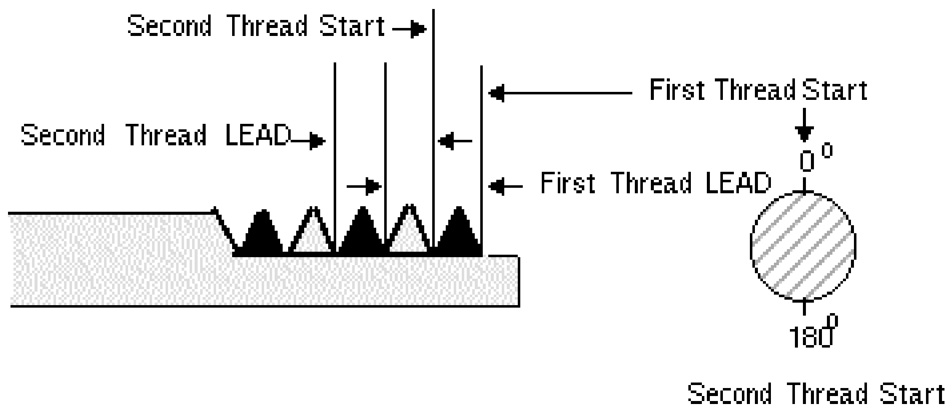


9. If you have chosen the Thread Data by Entity option, specify the following:
   - Digitize the root entity.
   - Digitize the check entity.
   - Digitize the start entity.

   OR

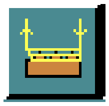
10. If you have chosen the Thread Data by Loc option, specify the following:
    - Specify the depth of the thread in the Thread Depth field.
    - Specify the length of the thread in the Thread Length field.
    - Digitize the starting location of the thread.
11. Specify the number of thread starts in the Thread Starts field. See the following figure for details of a two-start thread.

Figure 8-7 Two-start Thread



12. Specify the safe distance in the Safe Distance field.
13. Click Apply if you have finished entering all parameters for the task.

Specifying More Machining Parameters

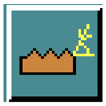


Use the MORE PARAMETERS option to specify more machining parameters for the current task.

Using this Option

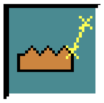
1. Choose the More Parameters option from the threading sheet. A set of more parameters appears on the sheet.

Figure 8-8 Threading Sheet for General Parameters

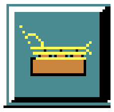


2. Choose the `Stepin by Angle` option and enter the angle for the stepin vector, in the field.

   OR

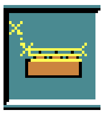


3. Choose the `Stepin by Loc` option and digitize the stepin vector.

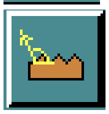


4. Choose the `Leadout by Angle` option and enter the angle for the leadout vector, in the field.

   OR

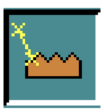


5. Choose the `Leadout by Loc` option and digitize the leadout vector.



6. Choose the `Stepout by Angle` option and enter the angle for the stepout vector, in the field.

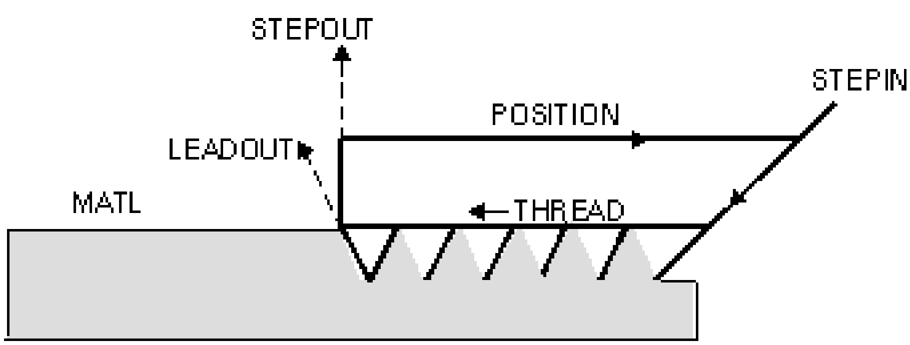
   OR

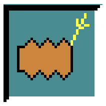


7. Choose the `Stepout by Loc` option and digitize the stepout vector.

   Please see the following figure for details of the stepin, stepout and leadout vectors.

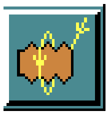
Figure 8-9 Threading Vectors



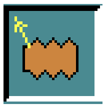


8. Choose the Threadin Off option if you do not want a threaded transition into the material.

OR

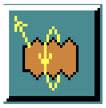


9. Choose the Threadin On option for a smooth threaded transition into the material while positioning to the depth of each pass. This also prevents the tool from dwelling at the start of the thread.

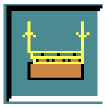


10. Choose the Threadout Off option if you do not want a threaded transition out of the material.

OR

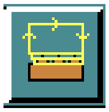


11. Choose the Threadout On option for a smooth threaded transition out of the material. This also prevents the tool from dwelling at the end of the thread.



12. Choose the Restart Off option if you do not want the tool to return to the starting point at the end of the threading cycle or between repeats of the last pass.

OR



13. Choose the Restart On option if you want the tool to return to the starting point at the end of the threading cycle or between repeats of the last pass.

14. Click Apply if you have finished entering all parameters for the task.

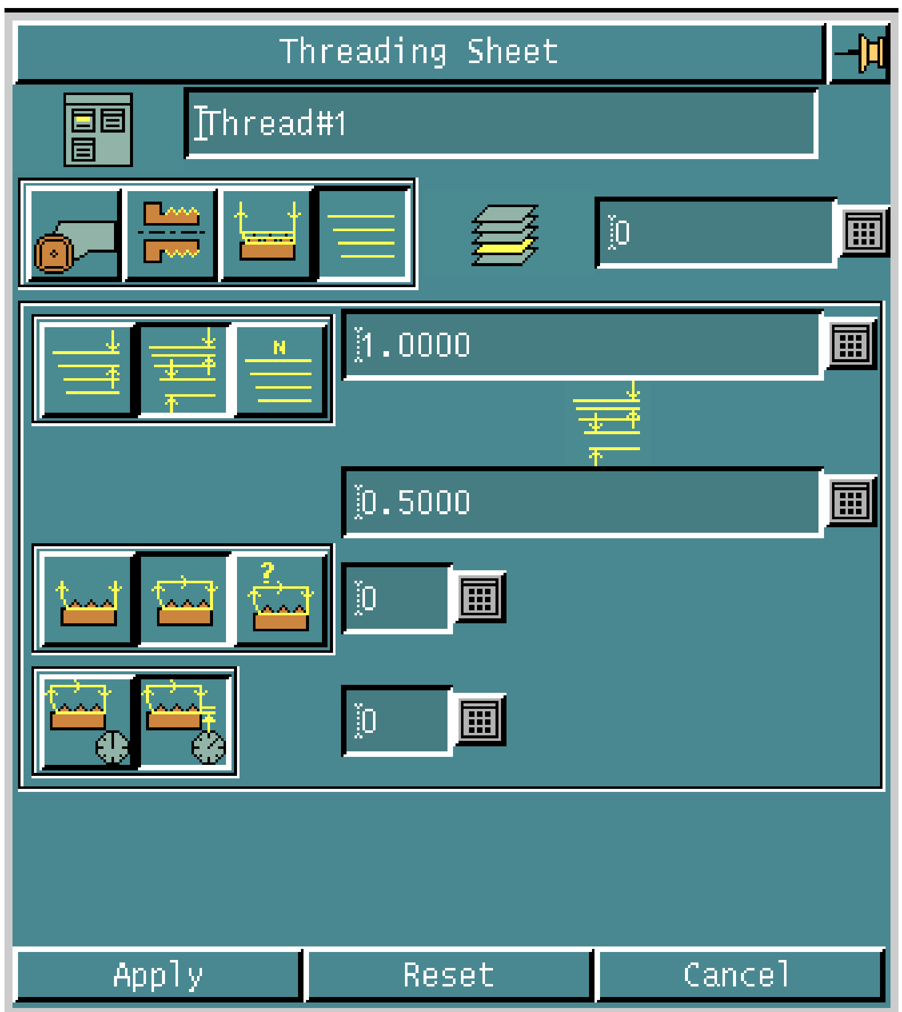
Specifying Further Machining Parameters

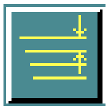
Use the FURTHER PARAMETERS option to specify further machining parameters for the current task.

Using this Option

1. Choose the Further Parameters option from the threading sheet. A set of further parameters appears on the sheet.

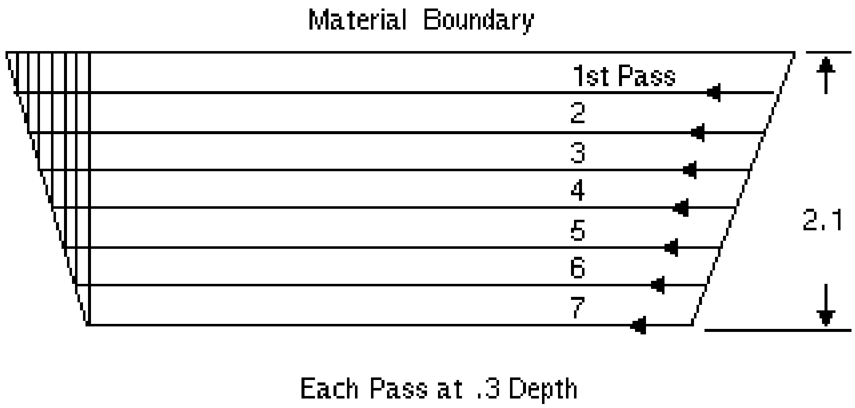
Figure 8-10 Threading Sheet for Standard Parameters



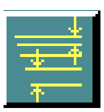


2. Choose the Maximum Depth option and enter the maximum depth of each pass. CVNC then calculates the number of passes at depths equal to or less than this maximum depth. See the following figure for details.

Figure 8-11 Maximum Depth of Each Pass

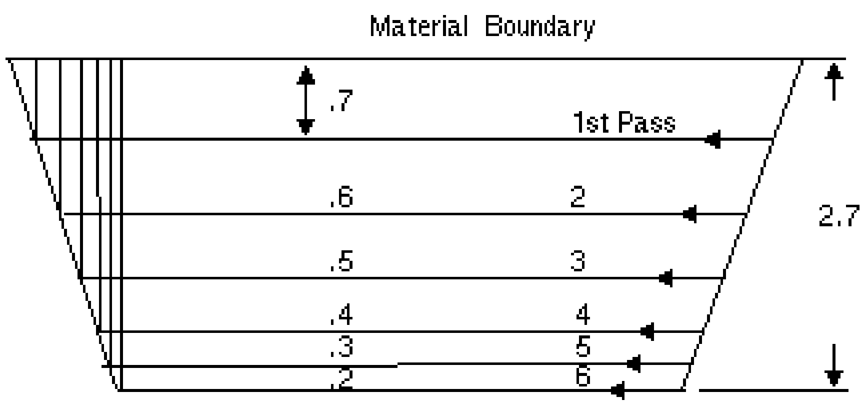


OR



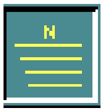
3. Choose the First And Last option and enter the depths of the first and last tool passes. CVNC then calculates linearly decreasing depths for each intermediate pass. See the following figure for details.

Figure 8-12 Depth of First and Last Cuts



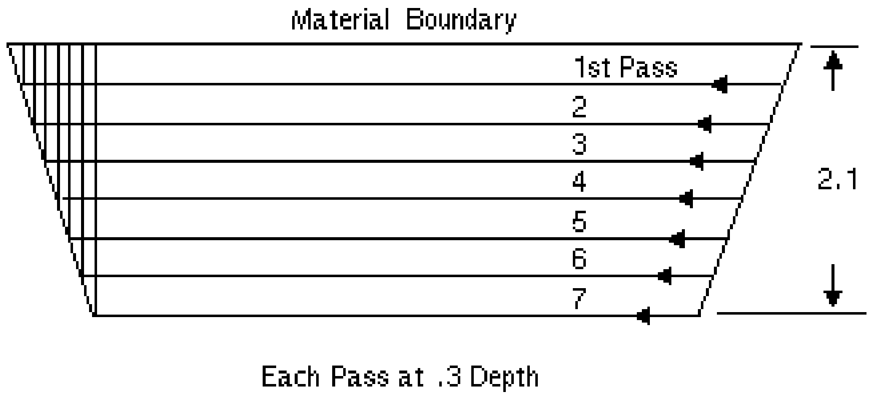
Please note: You should enter a greater depth for the first pass.

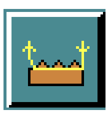
OR



4. Choose the Number of Passes option and enter the number of passes. CVNC then calculates an equal cut depth for each pass. See the following figure for details.

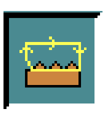
Figure 8-13 Number of Passes



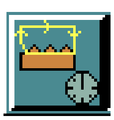


5. Choose the No Repeat option if you do not want to repeat the last pass.

   OR

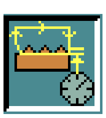


6. Choose the Repeat Last option if you want to repeat the last pass and enter the number of repetitions.



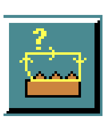
- Choose the Repeat Offset Off option if you do not want to invoke a new axis offset register during repeats of the last pass.

  OR



- Choose the Repeat Offset On option to invoke a new axis offset register during repeats of the last pass and enter the offset.

  OR



7. Choose the Opskip On option to output control data for skipping all or remaining last passes and enter the number in the field.

- Choose the Repeat Offset Off option if you do not want to invoke a new axis offset register during repeats of the last pass.

  OR

- Choose the Repeat Offset On option to invoke a new axis offset register during repeats of the last pass and enter the offset.

8. Click Apply if you have finished entering all parameters for the task.

Executing the Job

You have to execute the task after specifying parameters for it. After you have clicked Apply on the threading sheet, use the following steps for executing your job:

1. Click Apply on the operation sheet when you finish specifying all the tasks for your operation.
2. If you have used synchronization sections, you can see the Sync Master option in the NCBuilder sheet.

Figure 8-14 NCBuilder Sheet to Choose the Master Turret

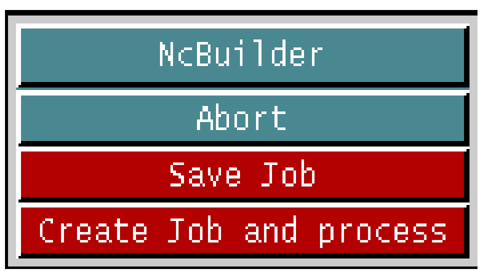
- Choose this option to select the master turret. The Synchronization Master Sheet appears.

Figure 8-15 Synchronization Master Sheet

- Choose the master turret for each synchronization section from the option list. For each synchronization section, CVNC uses feed rates and spindle speeds of the master turret.
- Click Apply.

3. Click Apply on the NCBuilder sheet after you finish specifying all the operations for your job. The following pulldown menu appears:

Figure 8-16 Pulldown Menu for Executing a Task



4. Click Create Job and process.

   You can now see the tool path and execution of your job.

Chapter 9

Tapping Tasks

This chapter describes how you can use the NCBuilder Graphical User Interface (GUI) for tapping tasks on turned parts. You can also refer to the *CVNC-T2 Command Reference* or the online help file to see the syntax of the TAP command.

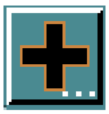
- Accessing the Tapping Options
- Specifying the Tapping Parameters
- Specifying Tool Assembly Selection Parameters
- Specifying Machining Parameters
- Executing the Job

Please note: Please refer to the *CVNC-T2 Command Reference* or the online help for the syntax of the TAP command.

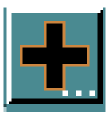
Accessing the Tapping Options

1. Choose the Manufacture option from the CADDS Environment Access menu.
2. Enter a new or existing Job Control File (JCF) in the turning mode. The Setup Turn task set appears.
3. Choose the NCBuilder option from the Task Set Access menu. The NCBuilder task set appears.

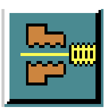
4. Choose the Job Block option from the NCBuilder task set. The NCBuilder property sheet appears.



5. Choose the Add Operation option from the NCBuilder property sheet. The Operation Sheet appears.

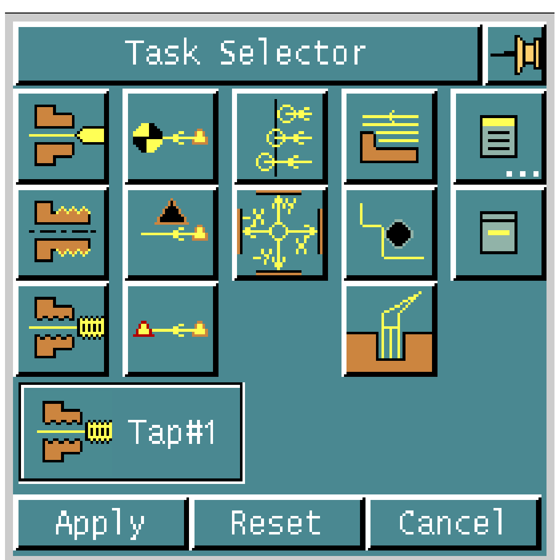


6. Choose the Add Task option from the operation sheet. The Task Selector palette appears.



7. Choose the Tap option from the Task Selector palette. The task name and its icon appear in the Task Selector palette as shown in the following figure. The sequence number of the task also appears. For example, if the task is the first tapping task in the operation, it has the number 1.

Figure 9-1 Task Selector Palette with Task Name and Icon



Please note: You should define the machine tool parameters and tooling parameters before specifying any task. See Chapter 2, “Machine Tool Definitions” for details.

8. Click Apply.

   The task now appears in the operation sheet.

9. Specify the geometry as described in “Specifying Geometry” on page 5-6.

Please note: This task does not require part geometry. However, you may specify geometry for the material boundary.

10. Specify the feed rates for the operation as described in “Specifying the Feed Rates for an Operation” on page 5-10.

11. Specify the spindle speed for the operation as described in “Specifying the Spindle Speed for an Operation” on page 5-13.

12. Specify the coolant setting for the operation as described in “Specifying the Coolant Setting for an Operation” on page 5-14.

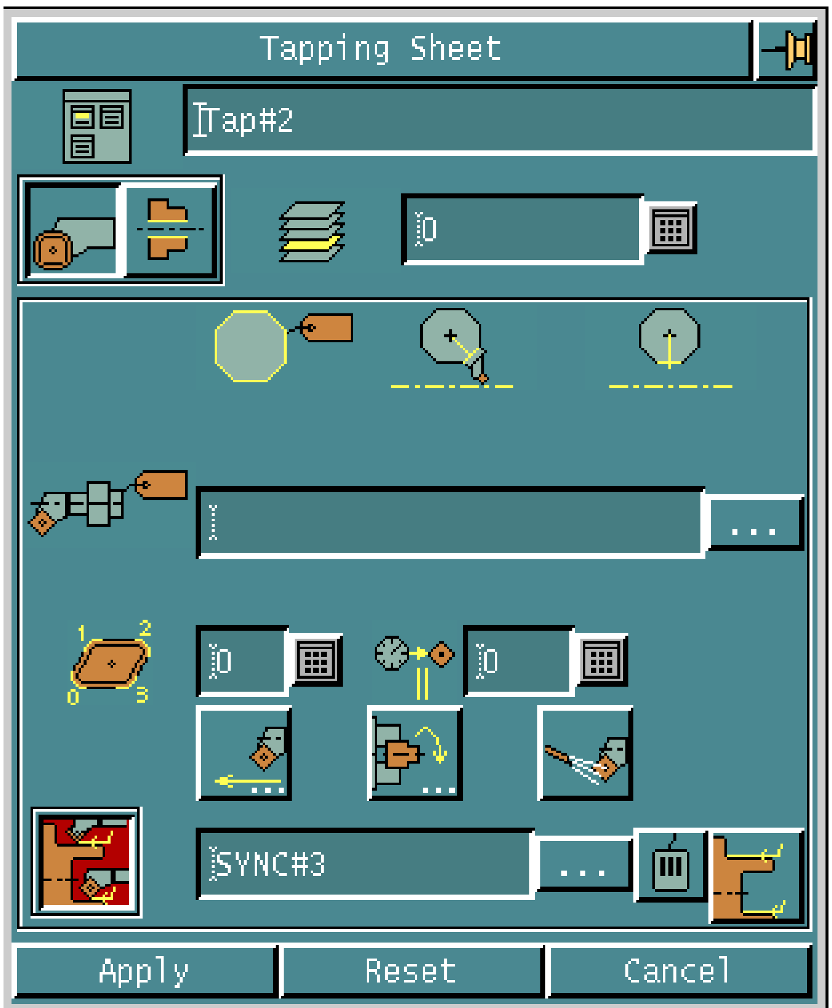
Specifying the Tapping Parameters

Use the EDIT option on the operation sheet to specify parameters for the tapping process.

Using this Option

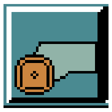
1. Highlight the task and choose the Edit option on the operation sheet. The Tapping Sheet appears.

Figure 9-2 Tapping Sheet



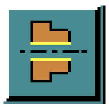
2. Specify the layer for creating the tool path in the Layer field.

Options on the Tapping Sheet



SPECIFYING TOOL ASSEMBLY SELECTION PARAMETERS
Displays a property sheet that enables you to specify tool assembly selection parameters for the current task.

See “Specifying Tool Assembly Selection Parameters” on page 9-6 for details.

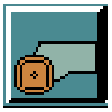


SPECIFYING MACHINING PARAMETERS
Displays a property sheet that enables you to specify machining parameters for the current task.

See “Specifying Machining Parameters” on page 9-9 for details.

Please note: For each of these options, a different set of parameters appears on the same tapping sheet.

Specifying Tool Assembly Selection Parameters

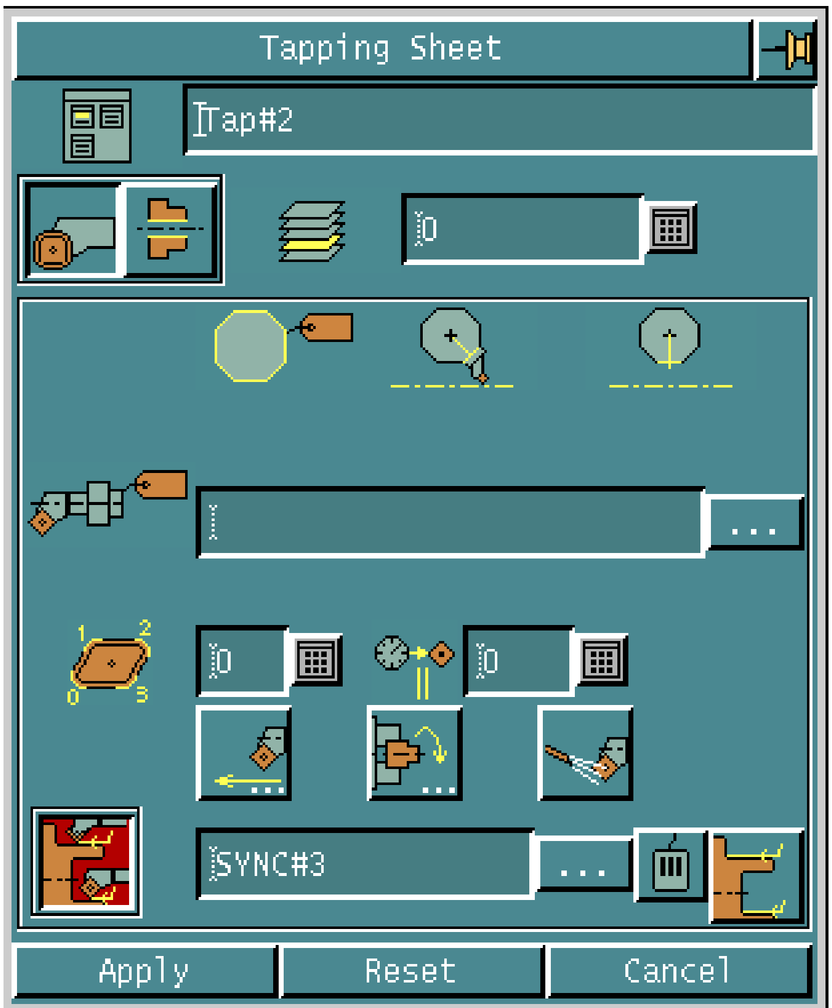


Use the TOOL PARAMETERS option to specify tool assembly selection parameters for the current task.

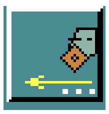
Using this Option

1. Choose the Tool Parameters option from the tapping sheet. The set of parameters for the tool assembly appears on the sheet.

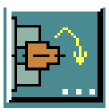
Figure 9-3 Tapping Sheet for Tool Assembly Selection Parameters



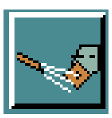
2. Specify the name of the tool assembly for the current task in the Tool Assembly Name field. To do this,
   - Click the list button. The Tool Selector scroll list appears.
   - Choose a tool assembly from the list.
   - Click Accept.
3. Specify the active tip on the insert for the tool assembly in the Active Tip field.
4. Specify the tip offset register for the active tip on the insert for the tool assembly in the Tip Offset Register field.



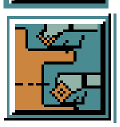
5. Choose the Feed Parameters option and enter the feed rates for the current task.



6. Choose the Spindle Speed Parameters option and enter the spindle speed parameters for the current task.



7. Choose the Coolant option and specify the coolant setting.



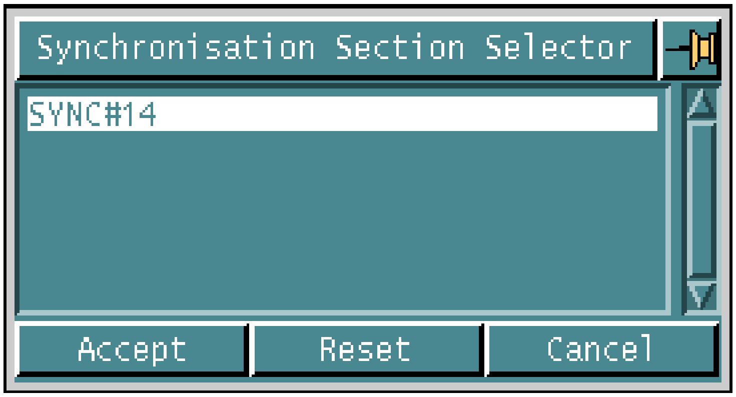
8. Choose the Sync option if you want to add the current task to the most recent synchronization section.

   A synchronization section is a part of a job during which two cutting tools in different turrets, having a two axis turning motion are synchronized to obtain four axis turning motion.

Please note: If no synchronization section exists, CVNC creates the current task in the first synchronization section. If you do not choose the Sync option the task is not synchronized and normal two axis turning motion is performed.

9. If you want to change the synchronization section, click the list button and select a particular synchronization section to apply to the current task. The Synchronization Section Selector scroll list appears.

Figure 9-4 Synchronization Section Selector Scroll List

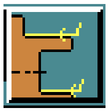


- Choose a synchronization section from the list.
- Click Accept.

  OR

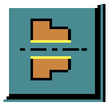
10. You can also digitize a task which has already been executed in a synchronization section. This can happen during the interactive mode of NCBuilder. The current task is then added to the same synchronization section as that of the digitized task.

    OR



11. Choose the New Sync Section option to create a new synchronization section to add the task.
12. Click Apply if you have finished entering all parameters for the task.

Specifying Machining Parameters

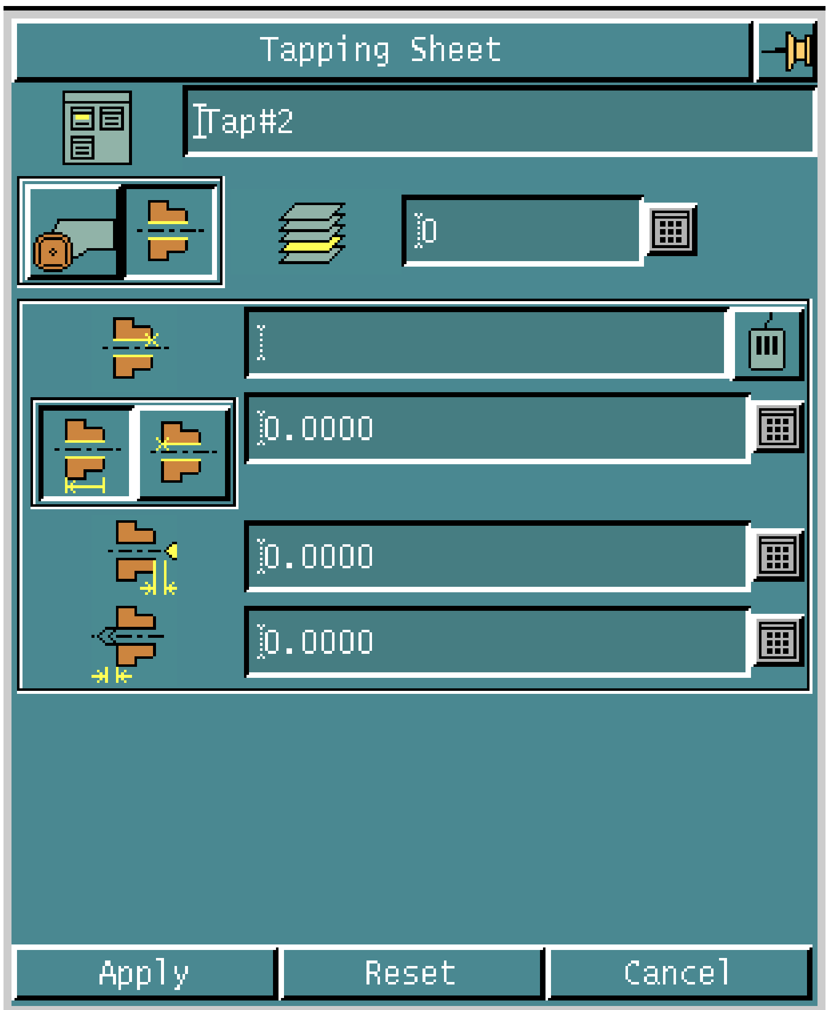


Use the MACHINING PARAMETERS option to specify machining parameters for the current task.

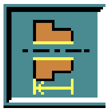
Using this Option

1. Choose the Machining Parameters option from the tapping sheet. The set of machining parameters appears on the sheet.

Figure 9-5 Tapping Sheet for Machining Parameters

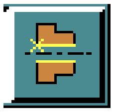


2. Digitize the start location for tapping the hole.



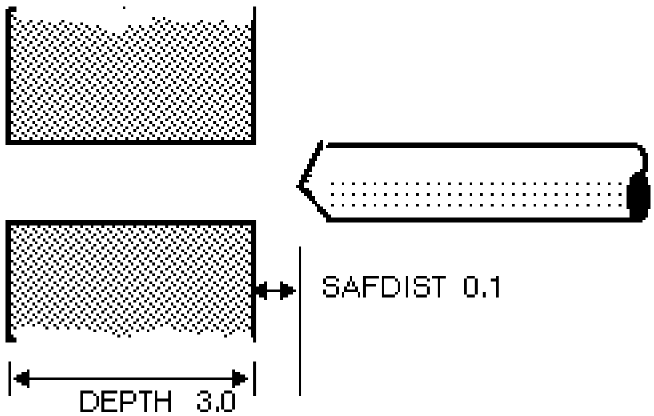
3. Choose the Depth Distance option and enter the depth of the hole in the Hole Depth field.

   OR



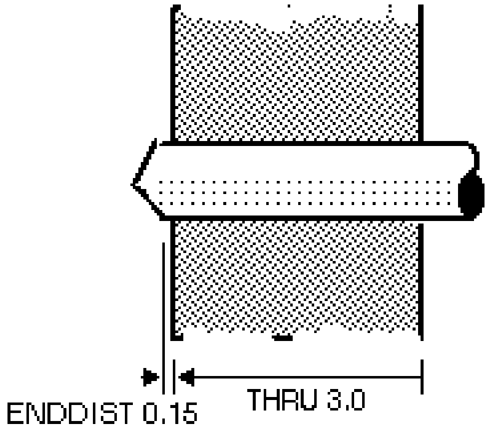
4. Choose the Depth Loc option and digitize a location for the depth of the hole.
5. Specify the value to be added to the start location of the hole for approaches and retracts in the Safe Distance field. See the following figure for details.

Figure 9-6 Safe Distance



6. Specify the value to be added to the depth of the hole in the End Distance field. See the following figure for details.

Figure 9-7 End Distance



7. Click Apply if you have finished entering all parameters for the task.

Executing the Job

You have to execute the task after specifying parameters for it. After you have clicked Apply on the tapping sheet, use the following steps for executing your job:

1. Click Apply on the operation sheet when you finish specifying all the tasks for your operation.
2. If you have used synchronization sections, you can see the Sync Master option in the NCBuilder sheet.

Figure 9-8 NCBuilder Sheet to Choose the Master Turret

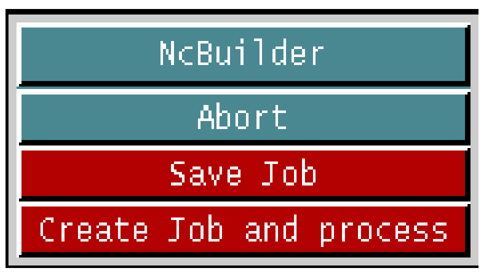
- Choose this option to select the master turret. The Synchronization Master Sheet appears.

Figure 9-9 Synchronization Master Sheet

- Choose the master turret for each synchronization section from the option list. For each synchronization section, CVNC uses feed rates and spindle speeds of the master turret.
- Click Apply.

3. Click Apply on the NCBuilder sheet after you finish specifying all the operations for your job. The following pulldown menu appears:

Figure 9-10 Pulldown Menu for Executing a Task



4. Click Create Job and process.

   You can now see the tool path and execution of your job.

Appendix A

Commands

This appendix describes the commands that CVNC issues when you choose options from the NCBuilder Turning GUI. The following sections describe these commands and their modifiers.

- NGROOVE
- TURNFACE

Please note: The other commands that NCBuilder issues for turning, like DRILL, TAP, THREAD are the same as in CVNC turning. Please refer to the *CVNC-T2 Command Reference* for these commands.

NGROOVE

This command generates a grooving tool path in NCBuilder Turning. You can configure the parameters of the command from the NCBuilder grooving sheet. NCBuilder issues it when you click Apply on the NCBuilder sheet.

Syntax

NGROOVE : ncgroup ; STARTLIN : ent ; CHECKLIN : ent

Modifiers

ncgroup: Specifies the part boundary.

STARTLIN ent: Specifies the starting entity of the groove.

CHECKLIN ent: Specifies the last entity of the groove.

TURNFACE

This command generates a turning and facing tool path in NCBuilder Turning. You can configure the parameters of the command from the NCBuilder turning and facing sheet. NCBuilder issues it when you click Apply on the NCBuilder sheet.

Syntax

TURNFACE : ncgroup1 ; MODEL : ncgroup2

Modifiers

ncgroup1: Specifies up to 16 ncgroups to define the part boundary for turning.

MODEL: Specifies that the ncgroups that follow define the material boundary for turning. This modifier is optional.

ncgroup2: Specifies up to 16 ncgroups to define the material boundary for turning. You can also specify a boundary generated by previous turning commands.

Index

A

C

D

E

F

G

I

J

L

M

N

O

P

R

S

T

U

V

X