



TRAINING KIT

Synergy

Turning Applications

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Turning Applications

Training Manual

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Introductions:

Welcome to Weber System's Synergy Cad Cam Turning Software. Synergy Lathe combines the world's most powerful cad cam automation tool with a specialized interface aimed at the turning professional.

About this Guide

The Synergy Lathe handbook contains detailed information about Weber Systems' Synergy Lathe tools and commands. We have designed this handbook to be used as an every day reference tool in your work with Synergy Lathe.

This guide assumes you have a working knowledge of your computer and it's operating conventions, including how to use a mouse and standard menus and commands.

Conventions Used in this Handbook

Mouse Notation

Click	Press and release the primary mouse button. For most people, this will be the left-button.
Right-Click	Press and release the secondary mouse button. This is usually the button furthest to the right on the top of the mouse.
Middle-Click	Press and release the middle mouse button.
Double-Click	Some what rapidly, press and release the mouse button twice.
Drag	Move the mouse as you press and hold the mouse button.

Text Notation

<Space>	(Or any other keystroke...) Press the key indicated between the <> symbols. In this case, you would press the spacebar. Any instructions that require notation of non-printing keystrokes will be set off by <> symbols. For example: <Esc>, <Enter>, or <Backspace>
<Shift> + <R>	(Or any other keystroke combination...) Sometimes it is necessary to press two keys at once. This will be represented by two keystrokes joined by a plus (+) sign. This notation means that you should press the <Shift> and <R> keys at the same time.
<Shift> , <R>	(Or any other keystroke combination...) A comma will separate keystrokes that should be pressed in sequence. This notation means that you should press <Shift>, release it, and then press <R>.
<Shift>+Click	Press and hold the Shift (or any other keystroke) while you press and release the mouse button.
→	The → symbol indicates a chain of menus that you will navigate by clicking. For example, you begin shutting the system down by clicking File → Exit from the main menu. This notation means that your should click the word File on the main menu and then click Exit from the file drop down menu.

Menu Notation

Drop Down	A drop down menu is the kind of menu that appears underneath an area after you click it. The majority of Synergy's menus are drop down menus.
Pop Up Menu	A pop up menu is just like a drop down menu, except that it appears when you click in a certain area or in a certain way. For example, right clicking on an icon on the Windows NT desktop will bring up a pop up menu.

Synergy's File Structure

Synergy stores all of its necessary files in regimented directory structure.

ARCHIVE

This is the default location for the Synergy's storage Archive.

WEBDEF

This directory contains interface files that work in conjunction with the guidef files.

UTIL

Tools and materials data and plotter configuration files go in this directory.

USER DATA

This directory stores miscellaneous information and auxiliary data.

TMP

This is the temporary directory used by the system.

POST_FORMATS

Common post processors are stored in this directory.

HTTP

This directory contains all of the Netscape-help executables and data.

GUIDEF

This directory contains customizable Graphic User Interface (GUI) definition scripts.

ETC

Output files that get copied to the bin directory are stored in this directory.

DATABASE

The Database directory contains strategies and messages used by the system.

CONTRIB

This directory acts as a Depository for customized ascrpts; it is for unsupported material that isn't updated.

BITMAPS

This directory contains bitmaps for the interfaces. The pictures and backgrounds that you see on the buttons go here.

BIN

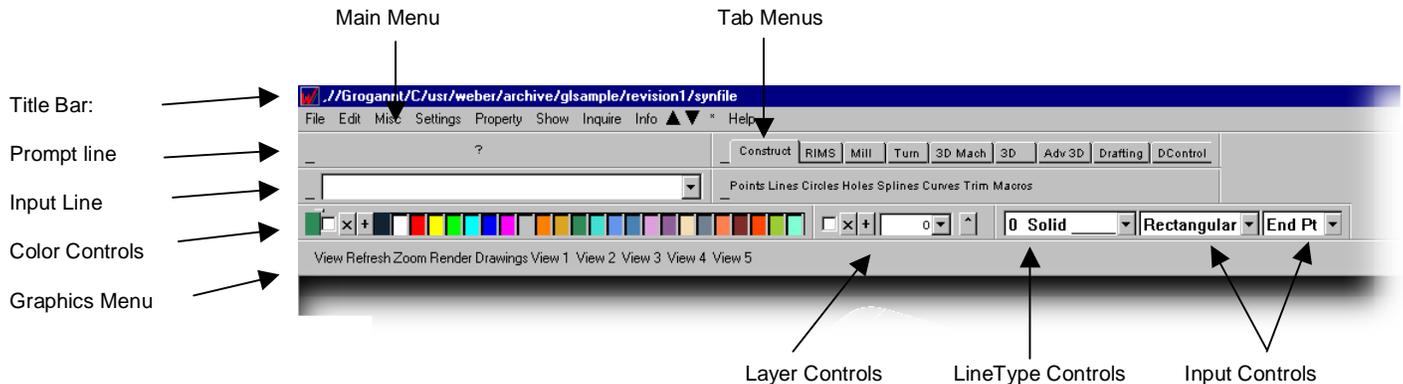
The BIN directory contains binary executable files.

ASCRPT

The Synergy script library base for the system is stored in this directory.

Introduction to the Interface

Throughout Synergy's documentation, specific terminology refers to specific areas of the screen. These areas are illustrated in the image below.



Release 12 Interface Changes:

Raising the Tab menus

The Tab Control Menus (**D**, **V** and **M**) have been moved to the **▲** menu. Tab menus can be raised and lowered by selecting the appropriate item from the **▲** menu.

- To raise the D tabs, click **▲** → Design
- To raise the V tabs, click **▲** → Variational
- To raise the M tabs, click **▲** → Misc

Undocking menus

Menus other than the main and graphics menus can be “undocked” from their default locations. Clicking and dragging a menu on any non-button area will undock the menu and you can move it around the screen. Menus will automatically snap to the edges of the screen, or you can choose to have the menus “float” freely.

Where is the Icon Bay?

The Icon bay is not raised by default. To raise the ICON bay, open the AMO Mill, AMO Lathe, or Lathe Tools interface from the **▲** menu.

The “except” check box

The “except” button on the layer and color control (formerly marked with a “-”) has been replaced with a check box. When the box is checked, the except mode is active.

Color Change Buttons

The **E**, **B**, **C** and **I** buttons have been integrated into a pop up menu accessed by clicking on the area on the far left of the color controls that indicates the active color.

The File Menu

Main Menu: File → New

Synergy will ask you to confirm that you want to start a new job. Enter "Y" to start a new file, enter "N" to abort procedure. Then you will have the opportunity to pick the CAD settings that you are going to use in the next file. CAD settings are predefined system variable settings that are used to determine things like measurements, scales, and other variables.

Main Menu: File → Open

This command can open an entire Synergy file, or just the file's geometry or TurboPost. Since Synergy can only work on one file at a time, any thing that you are currently working on that you have not saved will be lost.

- Enter [E] to open the entire Synergy File
- Enter [G] to open just the file's geometry
- Enter [T] to open only the file's TurboPost information.
- Enter [N] to cancel the open procedure.
- Enter [C] to open the NC Tape

Main Menu: File → Merge → Current Position

This command reads a file and places it at its original location. Enter [U] to undo the file placement. Enter [M] to finalize the merge

Main Menu: File → Merge → Reposition in 2D

This command reads a file and places it in several locations in 2D Enter [U] to undo the file placement.
Enter [P] to finalize the merge

Main Menu: File → Merge → Reposition in 3D

This command reads a file and places it in several locations in 3D Enter [U] to undo the file placement.
Enter [P] to finalize the merge

Main Menu: File → Save

This command saves a file into the current location. You are encouraged to use our Archive system rather than this method of file storage.

- Enter [E] to save the entire Synergy file.
- Enter [G] to save only the drafting geometry of the file.

Main Menu: File → Save As

This command saves a file at the current location, and lets you enter a name and description for the file. You are encouraged to use our Archive system rather than this method of file storage.

- Enter [E] to save the entire Synergy file.
- Enter [G] to save only the drafting geometry of the file.

Main Menu: File → Copy File

This command will make a duplicate of a file.

Main Menu: File → Delete File

This command will delete a file.

Main Menu: File → Exit

The Edit Menu

Main Menu: Edit → Undo / Redo

Synergy 12 has the ability to undo and redo **many, but not all** actions.

You can tell when an action can be undone by checking the undo button *after* you have performed the action. If the last action may be undone, the undo button is active. If there are no actions in the undo buffer or Synergy is in the middle of performing an action, the undo button appears faded. Actions that may not be undone will clear the undo buffer and the undo button will be blank.

Limitations -- Functions that can be undone

- Most geometric constructions both 2d and 3d
- Most entity trimming functions
- Most rotate, invert, move, copy, scale, and project functions
- Most drafting entities and geometric tolerance feature boxes
- Most delete functions, except for deleting all of a certain type of entity (for example: Delete all surfaces), or deleting variational records or objects
- Most editing of dimension entities can be undone. Exception: Edits which involve all dimensions cannot be undone
- Creation of cutter paths for lathe, 2d, 3d and multi-axis machining
- Modification of contours such as corner radii, lead-ins, chamfers
- Placing a hole feature in VDM, however, redefining the hole feature can not be undone

Functions that cannot be undone

- Edits which involve all dimensions cannot be undone
- Deleting all of a certain type of entity (for example: delete all surfaces)
- Deleting variational records
- Modifying variational records, changing variables or free parameters.
- Auto-linking a contour
- Manually linking a contour
- Unlinking an entity, dimension, group or contour
- Saving and archiving
- Embedding a file or execution of a thread in VDM
- Creation or modifying a variable value
- Assignment of tools, work zones, info packets and item properties
- Creating, modifying or deleting a boundary
- Creating, modifying or deleting an area fill
- Creating or deleting a VDM Object
- Plotting, zooming, viewing, and other graphical functions
- Changing of system settings

- Recalculation of 2d or 3d cutterpaths in a non-vdm environment
- Modification of 3d cutterpath entities
- Executing a strategy in VDM
- Executing all in VDM
- Executing an object in VDM
- Delete last record in VDM
- Reading a hole feature in VDM, however, placing the hole feature can be undone.

Main Menu: Edit → Delete

Use the commands listed under this menu to remove items from your geometry. Delete can remove a single entity, multiple entities, all the entities, entities in a box. It can also delete items by property (line type, color, layers, etc...).

Main Menu: Edit → Delete → Property

You will find commands under this menu for deleting properties, such as tools, operations, material and workzone data.

Delete an area fill by selecting Main Menu: Edit → Delete → Property → Area Fill

Main Menu: Edit → Delete → Group

This sub-menu contains commands for deleting groups, and things that synergy considers groups. This includes: Contours, Names, Plot Groups, Tool Images, and Dimensions. This menu also contains the commands for unlinking linked contours.

Main Menu: Edit → Delete → Types

This submenus contains the special delete commands that you need to use if you are working with types.

Main Menu: Edit → Geometry

This menu contains commands that you will use when you want to make changes to existing geometry. You will find commands for making changes to Dimensions, End Points, control points, weights, whole depths, diameters. You will also find the commands for stretching geometry under this menu.

Main Menu: Edit → Tool Library

When working with, creating, or changing your tool library, you'll need the commands located in this submenu.

Main Menu: Edit → Types

When working wit Types, you'll need the commands located in this submenu.

Main Menu: Edit → File Info

This command gives you access to the file header. You are limited to 4096 characters in the file header.

Main Menu: Edit → Last Scripts

This command will give you a list of the scripts that have been run recently. You can pick any script in the list to repeat it.

Main Menu: Edit → Script

This is a way to edit a script from within Synergy.

Main Menu: Edit → Interface

These commands are used for creating custom interfaces, called Utility Kits. See the section on utility kits for more details.

The Misc Menu

Main Menu: Misc → Re-Start

Use this command when you want to begin again. This will also get you out of an error message.

Main Menu: Misc → Leave Script

Use this command when you want to exit a script without completing it.

Main Menu: Misc → Redo Last Operation

This command should really be named “Repeat” Last Operation. This command repeats the last operation performed.

Main Menu: Misc → Calculator, Clock & Term Window

These commands give you access to software that was not written by Weber Systems. Calculator launches your systems default calculator. Clock brings up you systems Clock pallet. Term Window launches a Korn Shell.

Main Menu: Misc → NC Docs

This command will launch your system’s default web browser and display the current NC docs for the job you are working on. The web browser will display the current job’s CNC output, command file, Tool list, Time analysis and cost analysis.

The Settings Menu

Main Menu: Settings → Input

These are the methods by which you place information on the screen. You can either enter the absolute Cartesian value of the locations you are entering, (this is called absolute xyz input), or you place the data by picking the location on the screen. (This is called dynamic input.)

Main Menu: Settings → Cad Settings

The purpose of the Cad Settings info packet is to set various default parameters in the system which were previously hard coded into the setconstants or userconstants scripts. The Cad settings packet is selected when a New File is specified.

Main Menu: Settings → Constants

This file sets many of the systems named variables. These include, default values used to create 2D and 3D geometry and manufacturing, and storage directories for tool and material libraries. This file is located in /usr/weber/ascrpt/ and is called **setconstants**. If any customizations are done to this file it must be moved to /usr/weber/contrib/ascrpt, so that it is not over written by an update. The Cad Settings over write the constants.

Main Menu: Settings → Defaults

This script allows system default values to be set. These values are temporary for the synergy This script allows system default values to be set. These values are temporary for the synergy session that you are in. When you exit Synergy and come back in the values will go back to the defaults.

Main Menu: Settings → Variables

Allows the value of named variables to be changed. These values are temporary for the synergy session that you are in. When you exit Synergy and come back in the values will go back to the defaults.

Main Menu: Settings → User Views

This allows the user to capture an area of the screen and create a view out of that. The user views can be accessed from the graphics menu area under **View 1, View 2, View 3, View 4, View 5.**

Main Menu: Settings → Inverse Pens

Inverse pens are used for moving objects on the screen without erasing the graphics that they pass over. There are 5 inverse pen values 0,1,2,3,4.

Main Menu: Settings → Storage Mode

This script will change the mode for "how to save files." You can save the entire job, just the Geometry, or just the turbopost information, (command_file and tapefile).

Main Menu: Settings → View Resolution

This script will set the resolution for viewing geometry.

Main Menu: Settings → Plot Resolution

This script allows system default values to be set. These values are temporary for the synergy session that you are in. When you exit Synergy and come back in the values will go back to the defaults.

The Property Menu

Main Menu: Property → Layer

The layer options allow the user to separate parts of a job by placing sections of the job under a different number. The user can layer by an Entity, Contour, Box, Entity Type, Workplane, or Dimension. The also be accessed from the Color Control Area in Synergy.

Main Menu: Property → Color

Allows the user to change the current color or change the color of entities in a job. These can also be accessed from the Color Control Area in Synergy.

Main Menu: Property → Line Type

Allows the user to change the current line type or change the line types of entities in your job. These can also be accessed from the Line Type Control area in Synergy.

Main Menu: Property → Tool

Allows the user to assign a tool to either and entity, a contour or by using the box method.

Main Menu: Property → Work Zones

Allows the user to change a work zone or assign geometry to a work zone by different methods. A work zone is a work area on a part. There are several examples where you might use work zones: Tombstone work, parts that have many faces or sides, or fixture offsets.

Main Menu: Property → Name

Allows a name to be assigned to an entity.

Main Menu: Property → Hide

Allows you to hide or unhide different entities of your job.

The Show Menu

Main Menu: Show → All Data

Will show all geometry on the screen.

Main Menu: Show → Color

Will show all entities of a specified color.

Main Menu: Show → Layer

Will show all entities of a specified layer.

Main Menu: Show → Line Type

Will show all entities of a specified line type.

Main Menu: Show → Group

This script allows a group that was specifically made as a plot group to be plotted. The group is associated with specific layers and colors.

Main Menu: Show → Tool

This script allows geometry associated with a particular tool to be plotted on the screen.

Main Menu: Show → Work Zone

This script allows geometry associated with a particular work zone to be plotted on the screen.

Main Menu: Show → Operation

This allows a chosen operation or hole feature to be plotted on the screen.

ON-OFF SWITCHES

Main Menu: Show → Hidden

This will turn **on-off** the plotting of hidden geometry.

Main Menu: Show → Dimensions

This will turn **on-off** the plotting of all dimensions.

Main Menu: Show → Area Fill

This will turn **on-off** the plotting of area fills that are done under Drafting->Hatching.

Main Menu: Show → Tables

This will turn **on-off** the plotting of tables that are created in drafting.

Main Menu: Show → Surfaces

This will turn **on-off** the plotting of the surfaces that were created in 3D.

Main Menu: Show → 3D Cutter Paths

This will turn **on-off** the plotting of 3D Cutter Paths.

Main Menu: Show → 3D Cutter Path Arrows

This will turn **on-off** the plotting of 3D Cutter Path Arrows.

Main Menu: Show → Trim Curves

This will turn **on-off** the plotting of all trim curves.

Main Menu: Show → Ruler

This will turn **on-off** the ruler or drawing grid. The hotkey for this is **Ctrl R**.

The Inquire Menu

Main Menu: Inquire → Highlight

This will show what is grouped or autolinked by turning everything in the group blue. It will also show arrows that indicate the direction of the autolink.

Main Menu: Inquire → Group Highlight

This is the same as Highlight but the user will choose the name of the group to be highlighted out of a list.

Main Menu: Inquire → General (work space)

This will show the information pertaining to a particular element relative to the standard relative to the current workplane.

Main Menu: Inquire → General (real space)

This will show the information pertaining to a particular element relative to the standard work plane.

Main Menu: Inquire → Distance

This script measures the perpendicular distance between a point and an entity.

Main Menu: Inquire → Angle Two Lines

This script measures the angle between two lines.

Main Menu: Inquire → Contour

This script will interrogate a contour and bring up the relevant information in the CONTOUR INFO BOX for viewing.

Main Menu: Inquire → Volume

This script calculates the volume, surface area, and centroid of a part.

Main Menu: Inquire → Part Limits

This script calculates the maximum and minimum limits of a 3D part.

Main Menu: Inquire → End Point

This script interrogates an element and will display the coordinates of the end point.

Main Menu: Inquire → Pick Point

This script interrogates an element and will display the coordinates of the pick point in the PICK POINT info box for viewing.

Main Menu: Inquire → 3D Fillet Radius

This script finds the radius of a circle contained in a 3d fillet surface.

Main Menu: Inquire → Data Size

This script will place the database information in a file box for viewing.

Main Menu: Inquire → Scallop Calc

This script will calculate the Scallop height.

Main Menu: Inquire → List

This script will list the specified entities.

The Info Menu

Main Menu: Info

This is a tear off menu that will show general information. It can also be used for stepping through the CNC code one block at a time.

Working with the Graphics Menu

Synergy offers precision control over your viewing area using the menus contained in the graphics menu.

Graphics Menu: View

All of the options in the view menu are used for controlling the angle from which Synergy displays your graphics. The following is a list of options:

- Top
- Bottom
- Front
- Back
- Right Side
- Left Side
- Isometric

- Isometric 2
- Isometric 3
- Isometric 4
- H (horizontal) mill view
- Lathe Iso

Rotate

This brings up the **Rotation Angle** dialog box. Use the slider to rotate the view about any line. Choose the axis of rotation by clicking on the axis described in the prompt line.

If you wanted to rotate the view about the Y axis:

1. Choose **view menu**: View → Rotate. The prompt line will read: MOVE SLIDER Xaxis, Yaxis, Zaxis, Turn90, Reverse?
2. Click Yaxis in the prompt line.
3. Slide the **rotation Angle** slider.
4. When you are satisfied, Click the **exit** button.

Exit

You must manually exit the Rotation Angle Dialog box when you are rotating your views.

Axis

The axis button lets you pick the line you want Synergy to use as a reference for the rotation. Synergy will not rotate about a point or circle you must pick a line.

Zero

The zero button sets the rotation angle for the axis being rotated about to zero.

Refresh

Fit

Fit automatically scales the your geometry to make sure it appears in the graphics window.

Redraw

The redraw button will recreate the geometry in the graphics window. Use this feature to clean up artifacts that might appear on your screen without resizing your view.

Refine

Use refine when you want to ensure that your screen geometry is an accurate representation of what is really being created. This is especially useful when you are considerably zoomed in on your geometry.

Resize All

If you are working with multiple graphics windows, resize all will perform fit across all of them.

Refine All

If you are working with multiple graphics windows, refine all will perform refine across all of them.

View On and View Off

Once you have created drawing views in your geometry, you can turn those views on or off by using these commands.

Zoom

Zooming is self-explanatory. Zooming in on your geometry allows you to view the fine details of your geometry up close. Zooming back lets you view all your geometry from a distance to get an overall view of the part.

Synergy offers several methods of zooming.

Graphics Menu: Zoom → Two pt. Zoom <f3>

Two point zoom lets you zoom in on your geometry using a dynamic zoom box. The dynamic zoom box grows and shrinks from its center.

To zoom up on geometry

1. Select Graphics Menu: **zoom → two pt. Zoom**
2. Click on the approximate center of the area you are going to zoom in on.
3. then move your mouse away from that point. Notice how the zoom box expands from the center point automatically.
4. Click again when you have enveloped the area you wish to zoom in on with the zoom box.

Graphics Menu: Zoom → Previous <f4> and Graphics Menu: Zoom → Next <f5>

Zoom Previous and Zoom next let you switch back and fourth between previous zooms. Each time you zoom, Synergy remembers where and what you zoomed. **Zoom→ next** and **zoom → previous** let you quickly flip back and forth between zooms.

Graphics Menu: Zoom → half <f11> and Graphics Menu: Zoom → Double <alt> + <f11>

Zoom→ half automatically reduces the scaling factor by two. **Zoom→ half** zooms back.

Zoom→ double automatically increases the scaling factor by two. **Zoom→double** zooms in.

Graphics Menu: Zoom → Unzoom

Unzoom resets the graphics window to the last fit performed. It is the same as clicking **Refresh→ fit**, except that Synergy will not check to make sure that all the geometry is contained in the graphics window.

Graphics Menu: Zoom → Pan <f6>

Panning lets you move the view around in the graphics window.

Render

Synergy's rendering abilities have been enhanced with the addition of Open GL manipulation and viewing techniques. On the surface, these changes are transparent when using Synergy's View: Render menu commands, however, OpenGL makes working with and visualizing surface and wireframe geometry considerably easier.

View: Render → Refresh

Puts Synergy into an OpenGL mode where you can work with your geometry as shaded surfaces.

View: Render → Refresh / Wire

Puts Synergy into an OpenGL mode where you can work with your geometry as shaded surfaces and visible wire frames

View: Render → Wireframe

Puts Synergy into an Open GL mode where you can work with your geometry as OpenGL wire frames

View: Render → Rotate

Use this command to put Synergy into a Viewing Mode after you have rendered your geometry. The viewing mode is for rotating, zooming and rendering geometry only. You will not be able to create or make changes to geometry when you are in the viewing mode. Press <esc> to leave the viewing mode.

While in the viewing mode, use your mouse to manipulate the geometry:

- Left Click and hold while you drag your mouse to tilt your geometry
- Middle Click to Pan your geometry
- Right Click to zoom your geometry

Panning and Zooming are mode based, and will require a normal (left) click to exit from panning or zooming mode. Tilting is not mode based, and therefore tilting does not require a second click to exit from tilting mode. Simply click and hold the mouse button while dragging to tilt your geometry.

View: Render → Render Off

Turns off Synergy's OpenGL functions.

View: Render → Remove Triangles

This is a command that is used to remove superfluous rendering information from your computer's memory.

Drawings

View: Drawings → On and View: Drawings → Off

This is a toggle menu for turning the display of drawings on and off.

What are drawings?

Synergy Drawings are methods of displaying data in the form of a drafting drawing without creating separate geometry for each view. Drawings are only for viewing. It is not possible to work with the geometry represented. You will need to specify the length and width of the drawing. The lower left corner of the drawing is always placed at zero. A negative length or width may be necessary for drawings imported from IGES.

Once you have specified the drawing's length and width, each view is placed and clipped on the drawing relative to the lower left corner.

Once specified, the drawings may be deleted, moved, and rotated. The clip points and views may be changed.

To plot a drawing to scale, the length and width of the drawing must match the physical dimensions available on the plotter.

Synergy's drawings differ from IGES drawings.

User Views

The remaining buttons on the Graphics Menu are the user views. These buttons are marked View 1, View 2, View 3...

They are used to quickly switch between predefined custom view settings.

Input Options

Absolute (XYZ) Input

Synergy has two main ways of asking you where you want to place information on the screen. You either enter the absolute Cartesian value of the locations you are entering, (this is call absolute (xyz) input), or you place the data by picking the location on the screen (this is called dynamic input).

Which method of input you use is controlled using the input controls located on synergy's main interface. (See: Introduction to the Interface.)

XYZ INPUT OPTIONS

Rectangular [R]

This option creates a point at the X,Y,Z coordinates you enter.

You have three more options when in rectangular mode, which you can also control using the input controls.

Center return

A point is created at the coordinates of the arc center

Midpoint return

A point is created at the coordinates of the midpoint of a line.

End point return

A point is created at the end point nearest the point picked.

Free Point [f]

This option creates a point at the present cursor location

Intersect [I]

This option creates a point at the intersection of two elements.

Nearest [N]

This option will create a point nearest to the cursor on the entity

Polar [p]

This option creates points that are defined in polar coordinates (radius and angle).

Spherical [s]**X & Angle [xa]**

This option creates a point at an absolute X location and an angle. The angle is referenced from the last point

Y & Angle [ya]

This option creates a point at an absolute Y location and an angle. The angle is referenced from the last point.

X & Hypotenuse [xh]

This option creates a point at a known X coordinate and a specified distance from the last point described. If the specified distance does not intersect the X coordinate, no point is created.

Y & Hypotenuse [yh]

This option creates a point at a known Y coordinate and a specified distance from the last point described
NOTE: If the specified distance does not intersect the Y coordinate, no point is created

Hypotenuse: Angle [ha]

This option creates a point at a specified distance and angle from the last point described

X & Entity [xe]

This option creates a point on an entity at a known X location

Y & Entity [ye]

This option creates a point on an entity at a known Y location

Angle: Element: [ae]

This option creates a point at a specified angle intersecting an entity.

Lathe Point [lp]

This option creates a point using Z-axis and diameters.

Selecting Input Options Automatically

Position the active point with your cursor. Clicking a location on the viewing window selects a point according to the following rules:

1. If the cursor is within the pick distance of an existing entity, it will pick the entity in the same method as an ordinary XYZ input.
2. If no entity is within pick distance it will select a free point. The free point is always on your existing work plane and has a Z coordinate in the work plane of zero.
3. If your cursor is on a ruler, the program will select the closest ruler point.

Be careful not to place the cursor close to an existing entity, because the program will select the entity (rule #1) before it selects a free point (rule #2.)

Absolute Input Keyboard Shortcuts

To manually select the active point, use the following keys:

- <R> Enters normal XYZ input. All normal functions are active.
- <N> Enters Nearest Mode.
- <I> Enters Intersect Mode.
- <LP> Enters Lathe Input Mode.
- <E> Exits the dynamic mode and leaves script.
- Backup. Destination determined by the script.
- <X> Restart script. Goes to beginning of script. (Sometimes B and X will have the same function.)

Dynamic Input

Dynamic input allows the entity to change in reference to the location of the cursor.

While most commands require only one point to define the entity, some commands require two points. For example, commands that require picking two points include: Line Two points, Circle Rad Two Points, Axis Cylinder and Line Plane.

Dynamic Input Keyboard Shortcuts

Not all of these keys are active for all entities. For example, some entities, (like torus) may have degrees of freedom not addressed by these keys. Some entities will not use some of the keys. (There is no reason to adjust the radius of a rectangle.)

You can experiment with the keys to determine what keys are active on which occasions:

- [+/-] changes the sign of an angle.
- [6] increases the starting angle.
- [4] decreases the starting angle.
- [8] increases the angular extent.
- [2] decreases the angular extent.
- [9] increases the a second angle.
- [7] decreases the a second angle.
- [3] increases the a third angle.
- [1] decreases the a third angle.
- [>] increases the radius/length.
- [<] decreases the radius/length.
- []] increases the second radius/length.
- [[] decreases the second radius/length.
- [/] increases the third radius/length.
- [*:] decreases the third radius/length.

SYNERGY ARCHIVAL PROJECT SYSTEM

The synergy archival project system is a method of organizing the storage of synergy projects. It uses the standard hierarchical file system that is present on the NT operating system.

Each project which is archived contains all the files which can be associated with that project. In addition each time a given project is archived a new revision of that project is created.

The archival system provides utilities for managing the project archive, for retrieving archived projects, and for browsing the components of archived projects.

DIRECTORY STRUCTURE:

The default start of the project archive is in

```
/usr/weber/archive/
```

This location is specified in a system variable called

```
"archive dir"
```

Changing this variable in userconstants will relocate the start of the project archive.

A user may have multiple Synergy archives. Access to the different archives may be obtained by modifying the archive directory variable.

In the archive may reside either simple projects or directories of projects.

SIMPLE PROJECTS:

A simple project is a directory having a valid file name which contains all the revisions of that project.

Each revision contains a text file called info.txt which contains information about that project, revision notes and a list of the component files of that project. The information contained in info.txt is patterned after the template file located in /usr/weber/database/properties/Projects.

The component files can be the following:

```
synfile: The synergy file stored in synergy format.
command_file.txt: The synergy turbo-post command file.
tapefile.txt: The synergy turbo-post CNC file.
notepad.*.txt: The turbo-post notepad files in .txt format.
notepad.*.html: The turbo-post notepad files in .html format.
*.hpgl: The .hpgl plotter files.
*.gif: Any .gif images associated.
*.jpg: Any .jpg images associated.
*.html: Any .html files associated.
*.htm: Any .htm files associated.
*.doc: Any Microsoft word .doc files associated.
*.xls: Any Microsoft Excel .xls files associated.
```

The choice of components to be archived is specified by the system variable:

```
"archive file types"
```

This is set by default in setconstants and may be set in userconstants or changed on the fly.

Valid project names should consist of alpha-numeric sequences with dots or underbars. Other characters should not be used especially spaces, colons, commas, astericks, slashes, backslashes, # etc. These characters create problems when we are using regular expressions to scan directories.

A more descriptive name can be used in the info.txt file associated with each project.

DIRECTORIES OF PROJECTS:

A directory of projects is a directory which contains one or more synergy projects.

A directory of projects contains information about the types of projects contained in the directory in the file dinfo.txt. The information contained in dinfo.txt is patterned after a the dirtemplate file located in

```
/usr/weber/database/properties/Projects.
```

We recommend naming the directories with a name which has .d as a suffix. This is not a requirement.

Archive your job

It's time to save the changes you've made. You can do this by using Synergy's Archive feature. From the Main Menu, choose **File → Archive → Save → Archive**

The following instructions are for creating a new project directory from scratch:

The prompt line box will ask:

```
SELECT PROJECT New directory, List?
```

This prompt means that you have to tell Synergy where you want to archive your file. Because this part has never been archived before, you are going to create a new directory.

Click the word **New** in the prompt line. The prompt will ask you to name the new directory:

```
NEW PROJECT DIRECTORY NAME?
```

Enter: **Training.d**

Fill out the Multi-entry dialog box. You can name this job anything you want to in the name box (but avoid numeric expressions). Enter a description. Enter your name. Enter a status. Then click the **OK** button.

The system has placed you (although it doesn't show you at the top yet), in the usr/weber/archive/training.d directory.

```
SELECT PROJECT New directory, List?
```

This question is now asking for the project name.

Name the project by typing into the input line, then press <Enter>. Call this sample file: ex_one.

Fill out the next multi-entry dialog. Enter the following data:

```
Project name: Exercise One
Description: (Describe the part)
Author: (Put your name here)
Status: In Progress
```

When you have finished, click OK.

Now that the part has been archived, you can always keep track of that directory the part is stored in because the directory path will be displayed on Synergy's title bar.

Now choose **Main Menu: File → Archive → List → Archive List** and find your new project in the list.

Data Exchange

IGES

IGES is an acronym for *Initial Graphics Exchange Specification*, an ANSI graphics file format for three-dimensional wire frame models.

Import

You are going to start by importing an IGES File. From the main menu, find and click File → Import → IGES → New

The prompt line will ask you:

```
THIS ERASES THE CURRENT JOB LOAD Yes / No ? REP = Y
```

This is a safety mechanism. When you import an IGES file Synergy assumes that you are starting from scratch. Any data that you might have on the screen before you clicked the Import IGES button will not be saved once you answer "Yes" to this question.

When you choose your file from the box that comes up, it looks in that file to see whether certain attributes have been set.

Click on or type <Y> and press <Enter>. The prompt line will instruct you:

```
CHOOSE EXISTING IGES FILE FROM FILE BOX
```

Synergy will present you with a file dialog box where you will locate the file you want to open.. You may need to use the scroll bars and scroll the list of file names until you find it.

Reading the IGES Global Data

After Synergy has imported an IGES file, it will display Synergy's IGES global data window. This window gives you information about the file, including product identification, the name, the system filename, translator version, creation date, the scale, the IGES version number, and so on.

If you scroll down this information using the scroll bars on the right side of the global data box, under a section headed "Entity Totals," you may also see information about the geometric entities contained in the file. For example, an IGES type number 100 is a circle, and it will give you the entity count of what it has just imported.

Look at the bottom of the list, you'll see a key to understanding the markings that may or may not appear next to the entity listings.

The key says: "* these entities are not translated," and "+ these entities are only partially translated."

This means that when an entity is marked with an asterisk (*), Synergy is indicating that the information was contained in the original IGES file, and for whatever reason, Synergy was unable to read it. This could be

because the entity contained corrupt data so it was skipped. If an entity is marked with a plus sign (+), it indicates that the entity was read, but it may not have been translated on a 1-1 basis. Normally, this is because the original file contained a custom color, or a color that's not directly translatable according to the IGES standards. In this case, Synergy will translate that information to the nearest similar Synergy color.

Once you have looked at the IGES global data window, you can print the information if you find it important, or you can click Cancel, and move on.

The IGES Widget

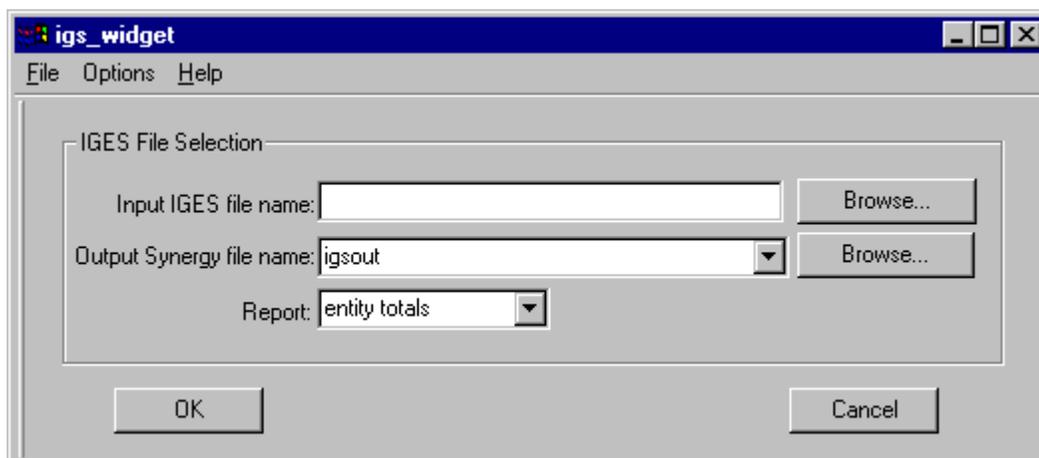
The IGES widget is a tool that gives you the ability to select which parts of an IGES file you want imported into Synergy. You have the option of not using this widget and importing the file in its entirety, however, there may be certain entities in some files that do not serve any purpose when imported into the Synergy environment. The IGES widget lets you choose not to import these entities.

The variable for turning the IGES widget on and off can be found in Synergy's main menu. Click Settings → Variables. Look for the variable named: "use IGES widget."

A switch box called the IGES WIDGET (program name: igs_widget) is available through the named variable "use IGES widget". When "use IGES widget" is set to "yes", the widget will be run, while a value of "no" indicates that the widget is not in use.

When you are running the widget, it will start automatically when you attempt to import an IGES file.

More information is available in the IGES widget helpfile, located in `usr/weber/filing.d/importigs`



Using the IGES widget

The IGES widget consists of a select box with a menu bar, IGES file name selection line, Synergy file name selection line, Synergy name option menu and an output report option menu.

IGES widget menus: Options**Filters:**

Brings up a window where you can pick and choose how you would like to filter six different aspects of an Iges file: including Surfaces, Drafting, Properties, UV Trims, XYZ Trims, Dependencies.

Trimming:

Brings up a window where you can select how you would like Synergy's IGES translator to treat the trimming methods used on five surface types (Nurbs, Revolutions, Tabcyls, Parasplines, and Ruled).

Special:

This brings up a dialog box that lets you control miscellaneous options. You can set the IGES translator's Tolerance factor, Entity timeout, Convert nurb surfs degree, and determine with which directories and entities you would like the translator to skip.

IGES Entity Types

```

100  Circle
102  Composite curve
104  Conic arc
106  Copious data - Linear path - Simple closed planar path
108  Plane
110  Line
112  Parametric spline curve
114  Parametric spline surface
116  Point
118  Ruled surface
120  Surface of revolution
122  Tabulated cylinder
124  Transformation matrix
126  Ration b-spline curve
128  Ration b-spline surface
140  Offset surface
142  Curve on a parametric surface
144  Trimmed parametric surface

```

IGES Annotation Entites:

```

202  Angular dimension
204  Curve dimension
206  Diameter dimension
208  Flag note
210  General label
212  General note
213  New general note
214  Leader
216  Linear dimension
218  Ordinate dimension
220  Point dimension
222  Radius dimension
228  General symbol

```

IGES Structure Entities

```

308  Subfigure definition
402  Associativity instance
404  Drawing
408  Subfigure instance
410  View

```

DXF

DXF is an abbreviation of *Data Exchange File*, a two-dimensional graphics file format supported by virtually all PC -based CAD products. It was created by AutoDesk for the AutoCAD system.

DXF is a very verbose, ASCII text representation of a file. There are four parts to a DXF file. The HEADER section defines values for most AutoCAD system variables. Most of these are not useful for another CAD system. The TABLES section defines items like line styles and user-defined coordinate systems. The BLOCKS section contains the definitions of blocks instanced in the model. The ENTITIES section contains the entity definitions and block instances. Using AutoCAD it is possible to create a valid DXF file that only contains an ENTITIES section.

Problem Elements with DXF

- Text
- Splines
- Views
- Surfaces
- Dimensions
- Fill Patterns
- Properties

Elements Missing From DXF

- Solids
- Nested Shape
- Offset Curve
- Curve on Surface
- Chain Dimension
- Curve Length Dimension
- Grid

Import

Synergy is capable of reading DXF ASCII data files in a limited way. **You can not export DXF files from Synergy.** The following are instructions for importing DXF data:

You are going to start by importing an DXF File. From the main menu, find and click File →

Import → DXF. The prompt line will ask you::

SELECT DXF FILE:

The selection box will appear, from which you will find and locate your file. The prompt line will ask you:

Undo or Merge:

Enter "U" to undo (delete) the file placement. Enter "M" to finalize the merge. Merge must be selected to finalize the process, **you will not import the file if you do not choose merge!** The prompt line will ask you:

DELETE REFERENCED BLOCKS (Yes or No):

Some DXF files may have conflicting data in their BLOCK generation and insertion group codes. Alternating responses to this question may solve this problem.

Problem Elements with DXF

- Text
- Splines
- Views
- Surfaces
- Dimensions
- Fill Patterns
- Properties

Elements Missing From DXF

- Solids
- Nested Shape
- Offset Curve
- Curve on Surface
- Chain Dimension
- Curve Length Dimension
- Grid

Import

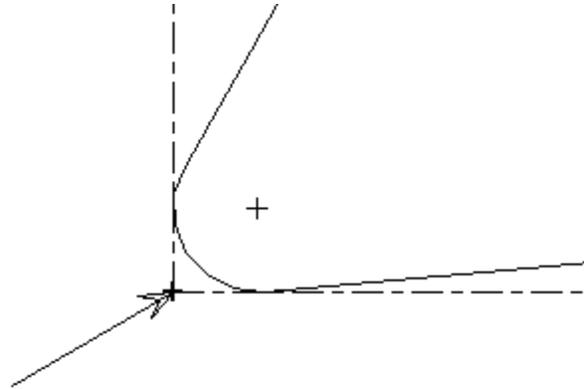
Synergy is capable of reading DXF ASCII data files in a limited way. **You can not export DXF files from Synergy.** The following are instructions for importing DXF data:

You are going to start by importing an DXF File. From the main menu, find and click File → Import → DXF. The prompt line will ask you::

SELECT DXF FILE:

The selection box will appear, from which you will find and locate your file. The prompt line will ask you:

Undo or Merge:



Theoretical Tool Tip

The standard synergy lathe post-processors use theoretical tool tip programming. The theoretical tool tip is located by intersecting tangent lines drawn horizontal and vertical to the tool nose radius. This point is not actually a point on the tool.

If theoretical tool tip programming is not used the most common alternate method is to program the tool nose circle center.

Theoretical tool tip programming has the advantage that turns and faces are programmed with the numbers that appear directly on the part design. On the other hand tapers and radii do not correspond to numbers on the design.

Tool nose radius programming has the advantage that the numbers which appear are just a uniform offset from the design contour.

Lathe Machining paths in synergy are generated to the tool nose radius center. This is the path drawn in synergy. In turbo-post this path will be output in either theoretical tool tip mode or tool nose radius mode, depending on the post processor setup. Again standard lathe turbo posts are normally set up to do theoretical tool tip programming.

Auto Link

Autolinking creates a contour (a special type of group), that is a chain of connected elements. An Auto Link should only be done ONCE for any group of elements. Each time Auto Link is used the new contour is added to the list of groups. To check to see if something has already had the Auto Link option used, use the HIGHLIGHT option or check the list of named groups.

Auto link can only be used on your geometry once. It cannot be repeated. Double elements will prevent a successful Auto Link.

INSTRUCTIONS:

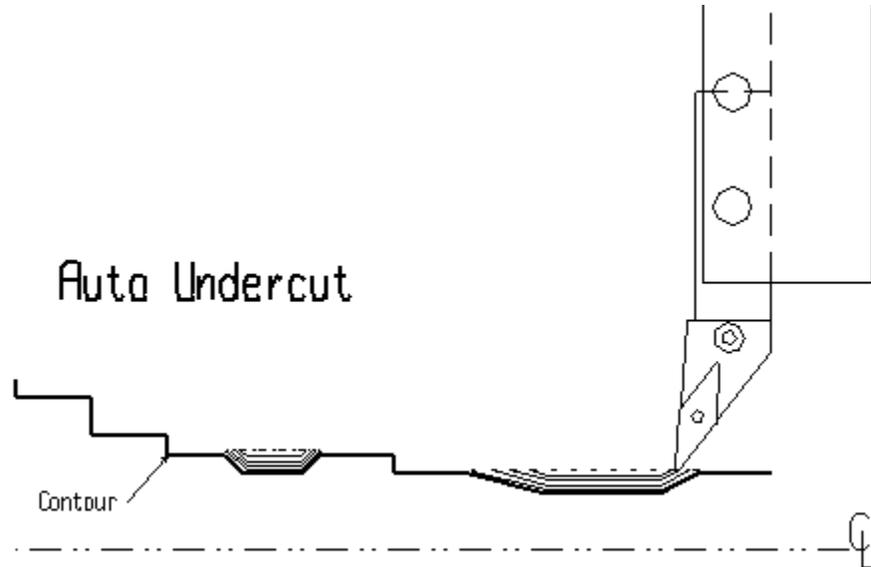
PICK START ELEMENT, Single: Pick a point on the starting element near the desired start point. This pick will determine the direction of the Auto Link. Single allows the Auto Link to proceed one element at a time and should be used if there are a lot of places where a decision about which direction to continue.

PICK NEXT, Backup, Single, New: Backup allows the Auto Link to be moved back one element at a time, starting at the ending element. This element will no longer be included in the contour created by the Auto Link. Single allows the Auto Link to proceed one element at a time and should be used if there are a lot of places where a decision about which direction to continue. New allows a new group of elements to use the Auto Link option and creates a new contour. The new contour is automatically named contour#, where # is incremented each time a contour is created. You can always rename the contour later.

PICK END ELEMENT: Pick the element where the Auto Link process should stop. If the first element is picked twice, the Auto Link will proceed around the contour as far as it can, back to the first element if the contour is closed and there are no choices of a direction.

Autoundercut: Undercut Roughing Cycle

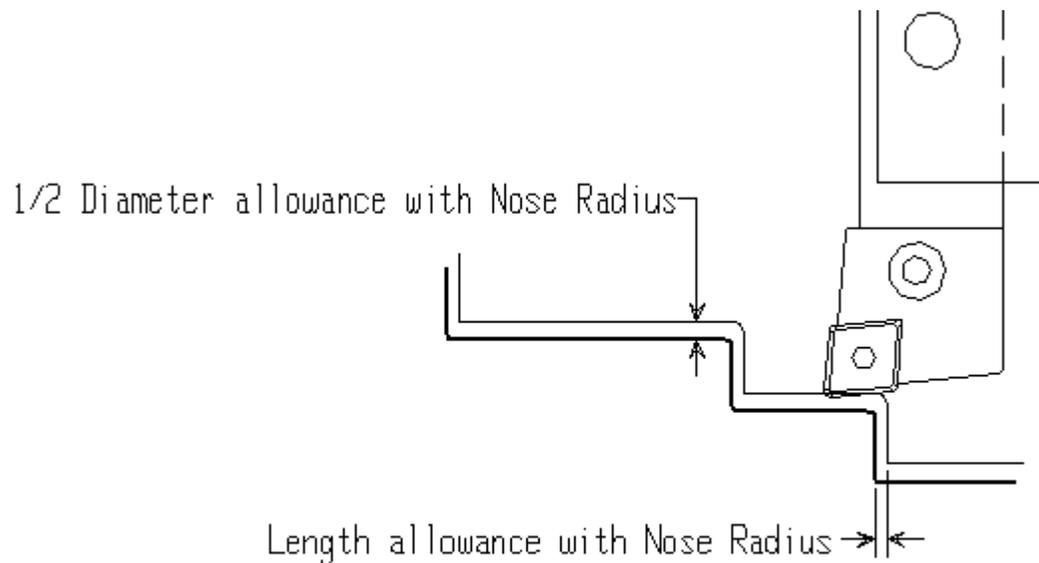
The tool clearance angle is used to determine unreachable areas of the starting contour, called undercuts. Successive paths rough out these undercuts where the depth of cut is determined by the amount of material to remove and the number of cuts requested.



INSTRUCTIONS:

- SELECT TOOL: Select the undercut tool.
- NUMBER OF CUTS: Enter the number of passes.
- FINISH ALLOWANCE: Enter the diameter and length amount of material to be left for a finish pass.
- PICK PATH: You have two choices. 1) Pick the contour to be roughed. This will force a tool clearance angle question to define the undercuts. 2) Pick a previously defined roughing path. This will use the data associated with the rough path to determine undercuts.
- TOOL CLEARANCE ANGLE: Enter the cutting edge clearance angle of the tool that left the undercut. If the tool is not selected the following questions will appear:
- TOOL NOSE RADIUS: Enter the tool nose radius of the tool to be used.
- TOOL QUADRANT: Enter the tool nose quadrant.

Diameter and Length Finish Allowance

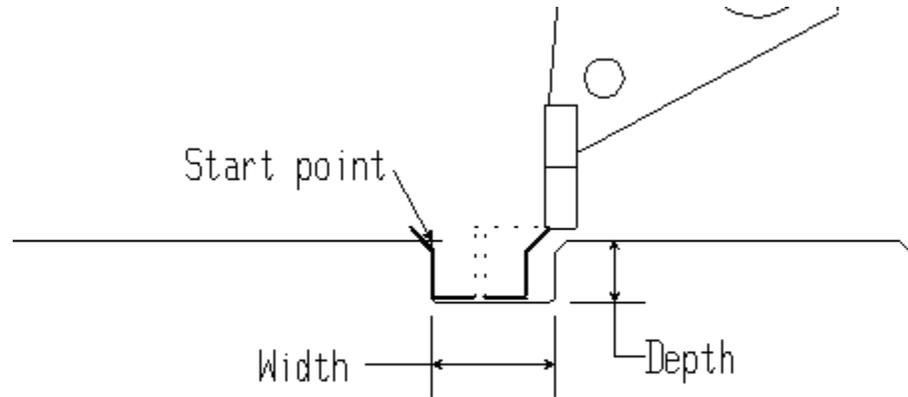


INSTRUCTIONS:

- DIAMETER FINISH ALLOWANCE:** The total finish allowance added in diameter to the contour to be machined.
- LENGTH FINISH ALLOWANCE:** The total finish allowance added in length to the contour to be machined. The contour being machined is shifted in X and Z by an amount equal to half the diameter finish in X and the length finish in Z. The direction of the shift is determined by the quadrant of the tool.
- UNDERCUTS:** When dealing with undercuts the following rules apply: For longitudinal cuts. The length finish allowance is added to the tool nose offset and the undercut contour is offset by this amount. The excess diameter finish allowance is then used to shift the offset contour in X. For radial cuts, half the diameter finish allowance is added to the tool nose offset and the undercut contour is offset by this amount. The excess length finish allowance is then used to shift the offset contour in Z.

Finish Groove

Use this technique to create a path to finish cut a groove with specified width and depth. A roughing groove path may be chosen to use pre-specified values for groove width and depth.



INSTRUCTIONS:

SELECT TOOL: Select the finish grooving tool.

GROOVE WIDTH: Enter the width of the groove to be created.

GROOVE DEPTH: Enter the depth of the groove to be created.

GROOVE START POINT ABS X,Y: Enter the X,Y coordinates for the start point of the groove. **NOTE:** See XYZ INPUT OPTIONS.

FINISH ALLOWANCE: Enter the amount of material to remain. If the tool is not selected the following questions appear:

TOOL WIDTH: Enter the width of the grooving tool.

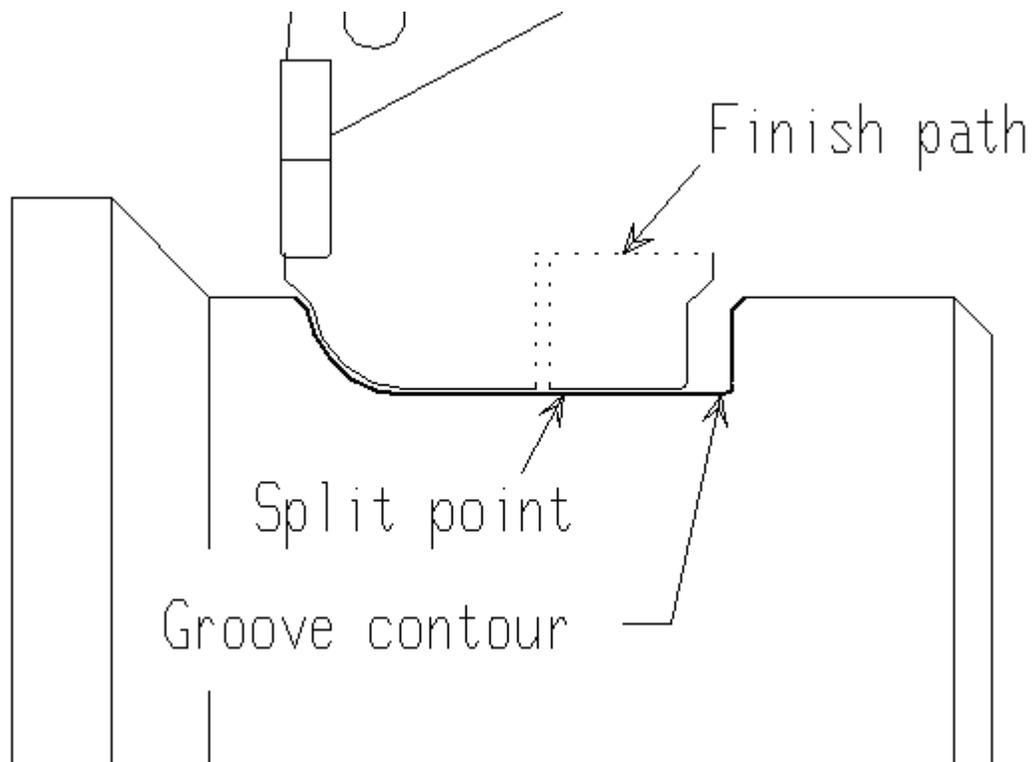
TOOL NOSE RADIUS: Enter the tool's corner radius.

TOOL QUADRANT: Enter the tool nose cutting quadrant.

Finish Groove A Contour.

Use this command to create a pathgroup to finish cut a contour with a grooving tool. The contour must be "groove like," that is, for an Od groove that the diameter decreases and then it increases.

The finish cut will start cutting the contour with one side of the tool until its center reaches the split point plus the overlap. It will then retract, go to the other end of the contour, and cut with the other side of the tool down to the where the last cut left off, plus the overlap. The grooving tool must fit in the groove. This cycle will not produce a correct path if the tool cannot fit in the location of the split point.



INSTRUCTIONS:

SELECT TOOL: Select the finish grooving tool.

FINISH ALLOWANCE: Enter the amount of material to remain.

PICK CONTOUR: Select the contour to be grooved.

SPLIT POINT: Input or Pick the point that the tool will switch sides. The split point plus an overlap distance determine where the bottom center of the groove tool will stop. If the tool is not selected the following questions appear:

TOOL WIDTH: Enter the width of the grooving tool.

TOOL NOSE RADIUS: Enter the tool's corner radius.

TOOL QUADRANT: Enter the tool nose cutting quadrant.

Lathe Offset

This command generates a cutter path with full, minimum or no look ahead. It will also change the path geometry to eliminate undercuts based on the tool clearance angle. The Minimum or None look ahead options should be tried first. If the path is unacceptable then use FULL LOOK ahead.

INSTRUCTIONS:

LATHE Forward OR Reverse: Forward will generate a path that has the same direction as the contour. R - Reverse will generate a path that is in the opposite direction of the contour.

TOOL NOSE RAD: Enter the toolnose radius of the cutter to be used.

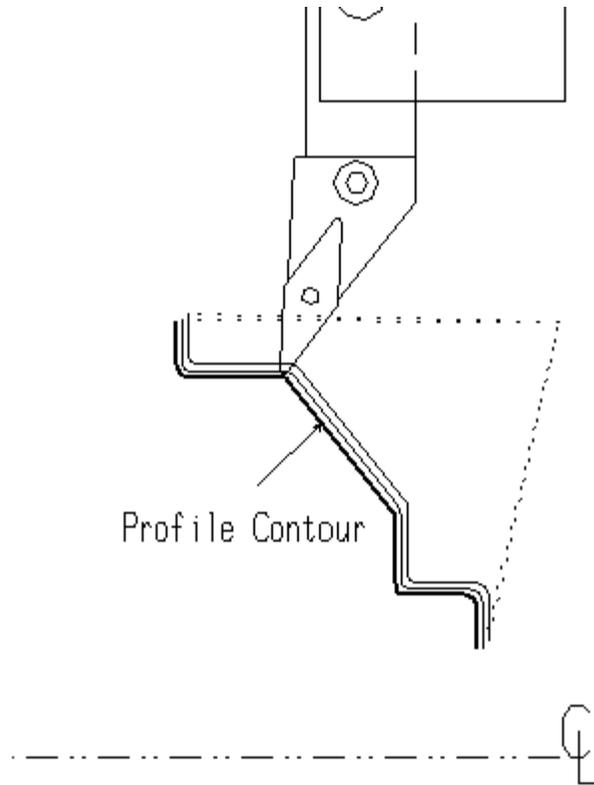
FINISH ALLOWANCE: Enter the diameter and length amount of material to remain.

PICK AN ELEMENT OF CONTOUR TO BE OFFSET: Pick an element on the contour to be offset.

PICK WHICH SIDE: Pick which side of the contour the path should be generated.

Profile: Roughing Cycle

Use this command to create a group of paths spaced at equalized distances to remove stock in length and diameter.



INSTRUCTIONS:

SELECT TOOL: Select the tool for profiling.

NUMBER OF CUTS: Enter the number of passes.

FINISH ALLOWANCE: Enter the diameter and length amount of material to be left for a finish pass.

PICK CONTOUR: Pick the contour to be roughed. If a tool is not selected the following questions will appear:

TOOL NOSE RADIUS: Enter the tool nose radius of the tool to be used.

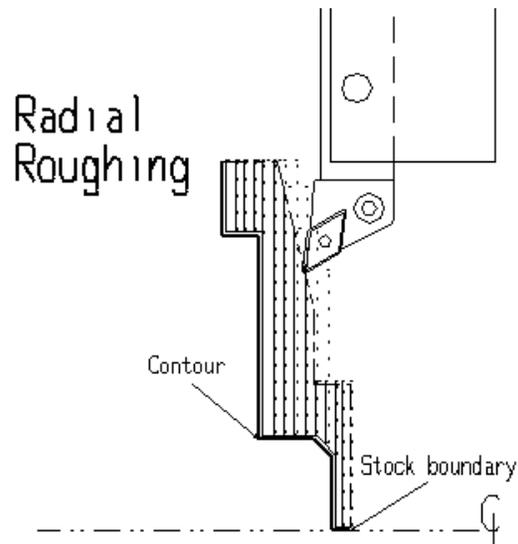
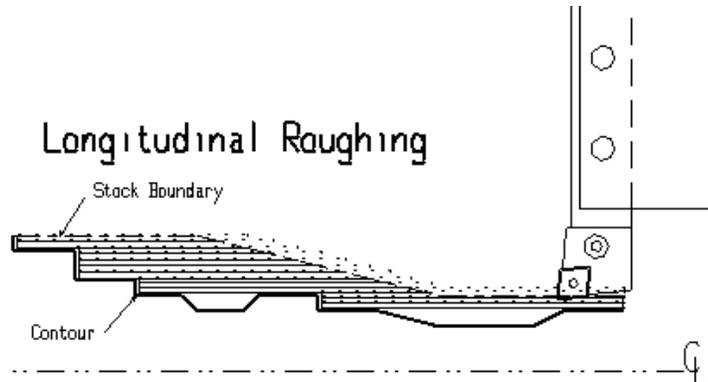
TOOL QUADRANT: Enter the tool nose quadrant.

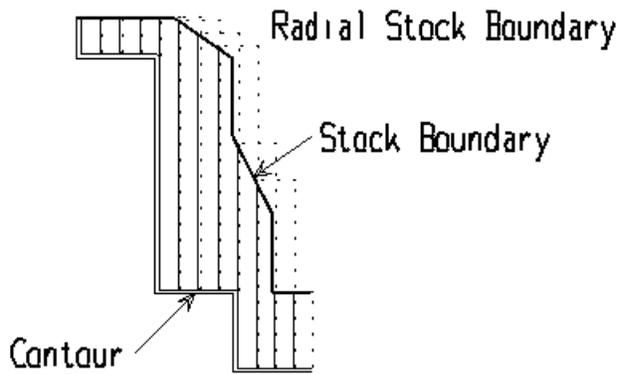
Rough: Roughing Cycle

DESCRIPTION:

This command creates successive cuts in a horizontal or vertical orientation. The tool clearance angle is used to determine unreachable areas of the starting contour, called undercuts.

The file `lathe.d/setvals` sets user definable constants. The stock contour must have no undercuts. It must be described in the same direction as the roughing contour. With a stock boundary the Clearance angle is assumed zero.





INSTRUCTIONS:

SELECT TOOL: Select the turning tool.

DEPTH OF CUT: Enter the depth of cut (the distance between successive passes).

FINISH ALLOWANCE: Enter the diameter and length amount of material to be left for a finish pass.

PICK CONTOUR: Pick the contour to be roughed.

PICK STOCK CONTOUR: Pick the contour used for the stock.(optional) If the tool is not selected the following questions will appear:

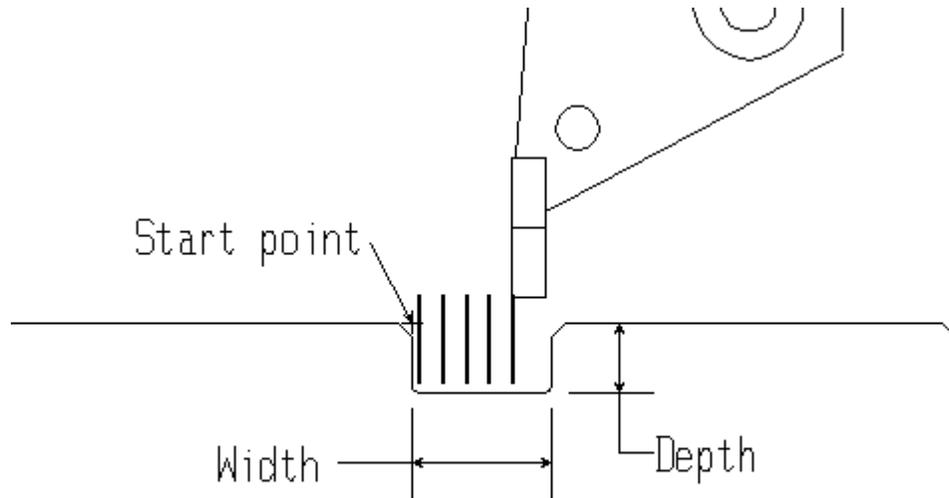
TOOL NOSE RADIUS: Enter the tool nose radius of the tool to be used.

TOOL QUADRANT: Enter the tool nose quadrant.

TOOL CLEARANCE ANGLE: Enter the cutting edge clearance angle.

Rough Groove

This command creates a path to rough cut a groove with specified width and depth. There are two methods of cutting the groove: `Cut_type= WsSequentialCuts`, where the cuts are made in a right to left or left to right direction, and `Cut_type= WsAlternateCuts`, where the cuts alternate from side to side.



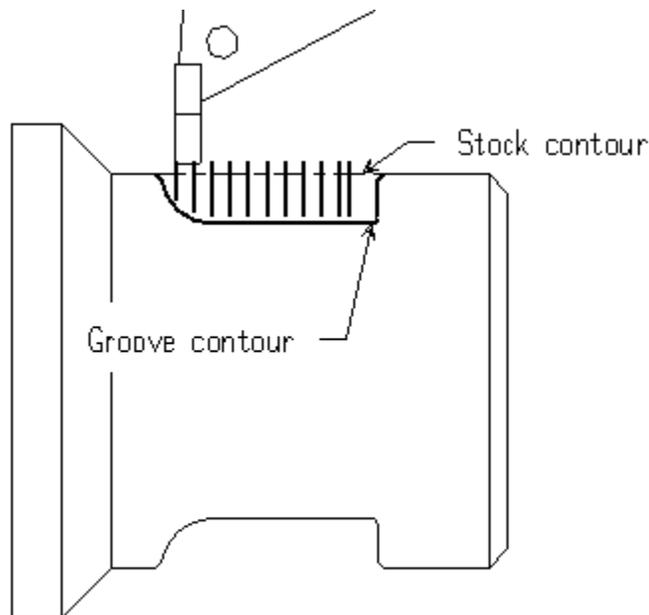
INSTRUCTIONS:

SELECT TOOL: Select the finish grooving tool.
GROOVE WIDTH: Enter the width of the groove to be created.
GROOVE DEPTH: Enter the depth of the groove to be created.
DEPTH OF CUT: Enter the depth of cut (this distance will be used on successive passes).
GROOVE START POINT ABS X,Y: Enter the X,Y coordinates for the start point of the groove. **FINISH ALLOWANCE:** Enter the amount of material to remain. If the tool is not selected the following questions appear:
TOOL WIDTH: Enter the width of the grooving tool.
TOOL NOSE RADIUS: Enter the tool's corner radius.
TOOL QUADRANT: Enter the tool nose cutting quadrant.
WIDTH OF CUT: Enter the width between cuts.

Rough Groove A Contour With A Stock Contour.

This command will create a pathgroup to rough cut a contour with a grooving tool. The stock contour is required and limits the cutting motion to between the stock contour plus clearance, and the groove contour. There are two methods of cutting the groove: Cut_type= WsSequentialCuts, where the cuts are made in a right to left or left to right direction, and Cut_type= WsAlternateCuts The cuts alternate from side to side.

The groove contour must be "groove like". If the contour has bumps or indentations which deviate from a "groove like" shape then this procedure may fail. At each cut location this routine calculates the first intersection of the leading and trailing edge of a standard shaped grooving tool. It will then generate the cut to that location. The cuts are generated in such a way as to insure cuts along walls which are vertical to the groove.



INSTRUCTIONS:

SELECT TOOL: Select the groove roughing tool.

DEPTH OF CUT: Enter the depth of cut (this distance will be used on successive passes).

FINISH ALLOWANCE: Enter the amount of material to remain.

PICK CONTOUR: Select the groove contour.

STOCK CONTOUR: Select the stock contour. If the tool is not selected the following questions will appear:

TOOL WIDTH: Enter the width of the grooving tool.

TOOL NOSE RADIUS: Enter the tool's corner radius.

TOOL QUADRANT: Enter the tool nose cutting quadrant.

WIDTH OF CUT: Enter the width of cut.

Undercut: Undercut Roughing Cycle

The contour is made into an undercut by connecting the endpoints. The depth of cut is determined by the amount of material to remove and the number of cuts requested.

This routine can fail if it cannot locate an undercut on the contour. The undercut contour needs elements at the ends to form an undercut. This can be accomplished by a small linear move or a corner radius at the ends of the contour.

INSTRUCTIONS:

SELECT TOOL: Select the undercut tool.

NUMBER OF CUTS: Enter the number of passes.

FINISH ALLOWANCE: Enter the diameter and length amount of material to be left for a finish pass.

PICK CONTOUR: Select the contour.

If the tool is not selected the following questions will appear:

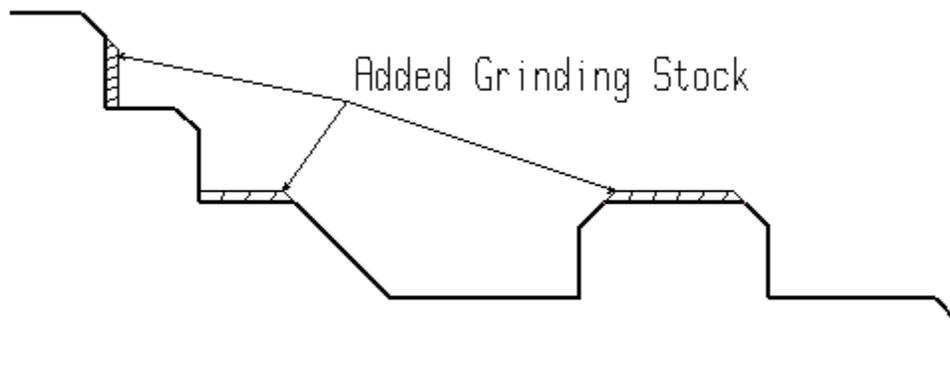
TOOL NOSE RADIUS: Enter the tool nose radius of the tool to be used.

TOOL QUADRANT: Enter the tool nose quadrant.

Add Grinding Stock To A Contour Line.

Use this command to add grinding stock to selected elements of a contour. It has provisions to remove small corner radii from the contour so that they don't interfere with this procedure.

This script is designed to be used when the contour may have been designed externally to Synergy or it is necessary for drafting purposes the first draw the finished shape. This script allows the grinding stock needed for machining to be added without having to reconstruct the geometry of the contour.



INSTRUCTIONS:

GRINDING STOCK: Thickness of stock allowance for grinding.

MINIMUM RADIUS FOR SHARP CORNER: Input the minimum radius to be used on corners. All radii smaller than this amount will be made into sharp corners.

PICK AN ELEMENT OF CONTOUR: Pick an element of the contour that the stock is to be added to.

PICK WHICH SIDE: Pick the side the stock is to be added.

PICK ELEMENT, Backup, Proceed: Pick all the elements which are to have the same grinding stock.

REPLACE ORIGINAL CONTOUR Yes, No, Cancel: Y - for yes will replace the original contour with the new contour. N - for no will not delete the original but will keep the new contour. C - for cancel will delete the new contour but leave the original unchanged.

Introduction to Variational Design

The Variational Design Module is a powerful tool for designing flexible part geometry. First introduced in the mid-eighties, parametric and variational design systems have allowed engineers a new degree of freedom to explore geometric relationships.

VDM allows programmers to build new designs that provide for future engineering changes. Manufacturing alterations that used to take days to complete can be finished in mere minutes. Manufacturing engineers can design with change in mind. Nothing is "hard-coded" and therefore customer requests can be anticipated in advance.

For example: if a programmer uses a tapered endmill to machine the wall of a component, he can assign variables matching the taper angle. If the engineer changes the taper angle, variational design will allow the automatic selection of a new endmill that matches these new manufacturing requirements. The engineer can build material requirements, tooling information, and quality control data directly into the part design. If the customer changes the design, manufacturing data such as CNC code, tooling lists, fixturing layouts and quality control all update automatically.

Another benefit is the cost and time estimation capabilities of variational design. Unforeseen changes or different part sizes can dramatically affect material requirements and manufacturing schedules. Synergy automatically tracks these changes and prints change bulletins so that communication between departments remains intact. Improved communication increases a manufacturer's ability to meet delivery deadlines.

Records and Objects

Every piece of geometry created in variational design has an ASCII data string that describes how that geometry was created. That ASCII data-string is that geometry's record. A record contains all applicable variational information for creating that item.

A group of records is called an object. There may be more than one object in a variational file. Objects are groups of records that make up geographic representations.

Variables

There are three types of variables used in Variational Design:

Independent Variables are created with any value and are not dependent on any existing geometry. They do not have a variational record constraining their values. Modifying an independent variable will affect all entities whose records reference that variable.

Dependent Variables are created dependent on existing geometry. Formula, Length, Radius, Angle, Angle Two Lines, and Dist Two pts are dependent variables. Dependent variables have variational records associated with them.

Free Parameters are independent numeric parameters referenced in a variational record. They are the means by which input data not specified by independent or dependent variables can be later referenced or modified for that particular record. Modifying a free parameter will affect only the entity created by that record.

Tools

The tool library is located in: `usr/weber/util/tooldata/`. Each set of tools is separated into the following directories:

Directory	Tools
<code>tools_in</code>	inch-based milling tools
<code>tools_mm</code>	metric milling tools
<code>ltools_in</code>	inch-based lathe tools
<code>ltools_mm</code>	metric milling tools

The tools are then further subdivided into categories of tools based on the type of machining each tool are used for.

Creating custom tools:

You should only ever make changes to copies of the tool files. That way, you won't overwrite the tool files that came with Synergy. We suggest copying an entire tool directory and renaming it. For example, if you are going to make custom inch-based tools, copy the entire `tools_in` directory, and give it a custom name, like: `custom_in`.

Steps to create a mill tool

1. Choose **Edit** → **Tool_library** → **Create lib tool**
2. Pick the tool category from the selections box.
3. Choose a tool that's similar to the tool you want to create.
4. Make changes to the data in the pop up create toolbox-. Click OK
5. Save the changes you've made. If you haven't given the tool a unique file name, Synergy will ask if you are sure that you want to overwrite the original file.

Making Lathe Tools and their Images

Here are the instructions for creating tool images for the Lathe and placing them in the proper directory.

1. In Synergy, click from the main menu: **File** → **Open** → **Geometry** and locate the `/usr/weber/util/tooldata/images` directory.
2. Choose a tool that is similar to the one that you want to create.
3. Modify the geometry to fit your new tool.
4. Choose **File** → **SaveAs** and save it as a different name. This will be your image name.
5. In Synergy: Choose **File: Copy File** and choose the file you just saved as the copy name.
6. If you haven't already, copy and rename your tools. Then modify them accordingly. In the tool files, the tool image file should give the name and location of geometry file that you copied to `"/usr/weber/util/tooldata/images/."`
7. Now when you load the tool and make the Lathe image, the images that you created will be associated with the appropriate tools.