



TRAINING KIT

Synergy

Synergy Mill Manual

Synergy Mill

Training Manual

Table of Contents

SYNERGY MILL	2
TABLE OF CONTENTS	3
INTRODUCTION	5
ABOUT THIS GUIDE	5
CONVENTIONS USED IN THIS HANDBOOK	5
MOUSE NOTATION	5
TEXT NOTATION	5
MENU NOTATION	6
SYNERGY'S FILE STRUCTURE	7
INTRODUCTION TO THE INTERFACE	9
THE FILE MENU	9
THE EDIT MENU	12
THE MISC MENU	14
THE PROPERTY MENU	15
THE SHOW MENU	15
ON-OFF SWITCHES	16
THE INQUIRE MENU	17
THE INFO MENU	18
WORKING WITH THE GRAPHICS MENU	18
INPUT OPTIONS	22
DYNAMIC INPUT MODE	24
DYNAMIC INPUT KEYBOARD SHORTCUTS	24
EXERCISE 1: CIRCLES AND HOLES WITH DIMENSIONS	25
CONSTRUCT OUTSIDE CONTOUR	25
CONSTRUCT INSIDE CONTOUR	31
PLACE CIRCLES	33
AUTOLINKING	34
INTRODUCTION TO THE ARCHIVE	35
REVIEW DRAWING EXERCISE 1	36
APPLYING HOLE FEATURES	37
GENERATE CUTTER PATHS	38
TURBOPOST THE JOB	40
NC DOCUMENTATION	41
RE-ARCHIVE THE JOB	41
EXERCISE 2: ADVANCED 2D	42
CONSTRUCT OUTSIDE GEOMETRY	43
CONSTRUCT LARGE POCKET	45
AUTOLINK CONTOUR	47
DIMENSIONING	48
EXERCISE 2 MANUFACTURING	49
POCKET MILLING	49
GENERATE NC CODE	51
RUNNING AMO	53
DATA EXCHANGE IGES	54
THE IGES WIDGET	55
IGES ENTITIES	57
DXF	58
TOOL LIBRARIES	59
UTILITY KITS	60
USER DEFINED HOTKEYS	61

EXERCISE 3: TRIMMING
COPYRIGHTS

62
63

Introduction

Welcome to Weber System's Synergy CAD/CAM Software. Synergy combines the world's most powerful CAD/CAM automation tool with a specialized interface aimed at the manufacturing professional.

Weber's flagship product, Synergy, has the ability to create CNC programming code for almost all types of production machining. Synergy has the unique capacity to produce standardized processes that reflect a company's specific needs. Manufacturers around the globe are using Synergy to produce quality CNC programs, using fewer resources, faster than ever before.

About this Guide

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

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 to farthest right on your 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 you 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 a regimented directory structure. All of the following directories reside under **C:\usr\weber**.

ARCHIVE

This is the default location for the Synergy's storage Archive. The file archive is designed to store the synergy file along with all associated file components into a directory structure.

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 ascripts; 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.

SCHEMAS

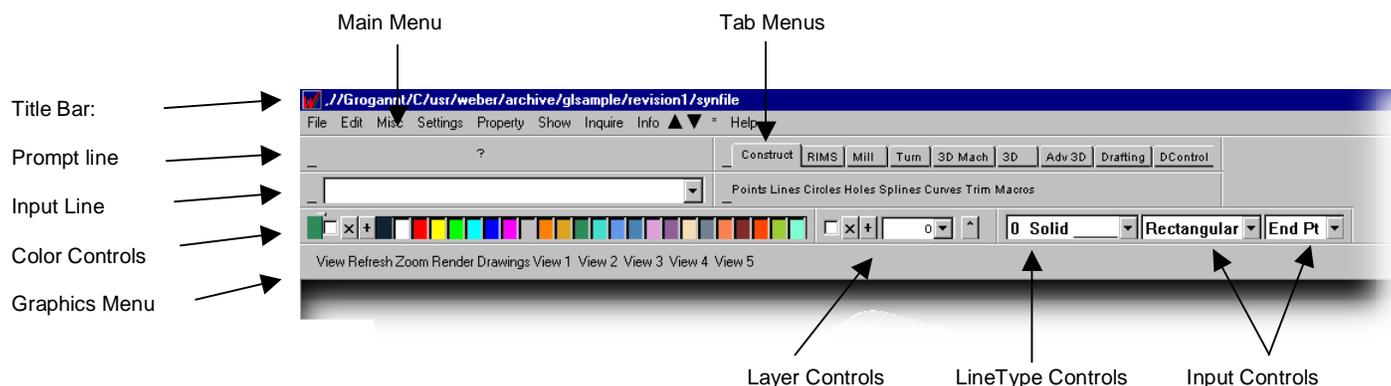
This directory holds files that the system needs to run parasolids.

ST

This is the default storage directory for Synergy. When the user does a **Save** or **Save As**, this is where the files get stored. This directory can easily be mapped to a different location.

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.



Raising the Tab menus

Synergy is divided into four basic systems, Design, Variational, Solids and Variational Solids. Each system is accessed through the Tab Menus. Four buttons are provided to raise the appropriate system.

- To raise the Design D tabs, click D
- To raise the Variational tabs, click V
- To raise the Solid Modeling tabs, click S
- To raise the Variational Solid Modeling tabs, click V/S

Utility Kits

The Synergy user configured Utility Kits are raised in Synergy by clicking on them from the ▲ menu.

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.

The “except” check box

The visible Colors/Layers can be inverted by use of the “except” check box.. When the box is checked, the “except” mode is active.

Color Change Buttons

The **Entity**, **Box**, **Contour** and **Inherit** buttons have been integrated into a pop up menu accessed by right clicking on the area on the far left of the color controls that indicates the active color.

Synergy's 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 info.

- 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 → Merge → Graphical

This command reads a file and allows the user to graphically place it.

Main Menu: File → Archive

The file archive is designed to store the Synergy file along with all associated file components into a directory.

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 to a desired 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 → Work Space

This will allow you to work in two sessions of Synergy at once. You can switch back and forth by choosing primary and secondary. Primary and secondary are totally independent of each other.

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 → Import

This is where a user can import an IGES file, DXF file or a Prompt External. A Prompt external is geometry data created in Weber System's first generation software called Prompt.

Main Menu: File → Export

This is where a user can Export out of Synergy, an IGES file or a CNC program either stored down in a Synergy file or just created.

Main Menu: File → Exit Main Menu: File → Exit

This is the recommended command for exiting the Synergy system.

The Edit Menu

Main Menu: Edit → Undo / Redo

Synergy 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, reading 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 or objects
- 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, hole depths, and 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 → User Constants

This will allow a user to customize Synergy by setting defaults. An example may be mapping the storage directory to another drive. Once this file is changed it gets stored down to /usr/weber/contrib./ascrpt/ and will never get over written by an update.

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 → Execute Script

This will allow a user to execute a script that may not be attached to a button. The script must be located under /usr/weber/ascrpt.

Main Menu: Edit → Last Scripts

This command with give you a list of the scripts that have been run recently. You can pick a script name and re-execute the script.

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 Synergy menus, tool kits, or also referred to as Utility Kits. See the section on utility kits for more details.

The Misc Menu

Main Menu: Misc → Re-Start

Use this command when you need to get out of an error in Synergy.

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 non-variational hole operation performed.

Main Menu: Misc → Calculator & Term Window

These commands give you access to software that was not written by Weber Systems. Calculator launches your systems default calculator. 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 → Defaults

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 sets the resolution for plotting.

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 layers can be shown under the Layer 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 an 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 which 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 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.

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 by a drawing.

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]

This option creates a point that lies on a sphere and is defined by a radius and two angles.

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.
L	Enters Lathe Input Mode.
E	Exits the dynamic mode and leaves script.
B	Backup. Destination determined by the script.
X	Restart script. Goes to beginning of script.

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.

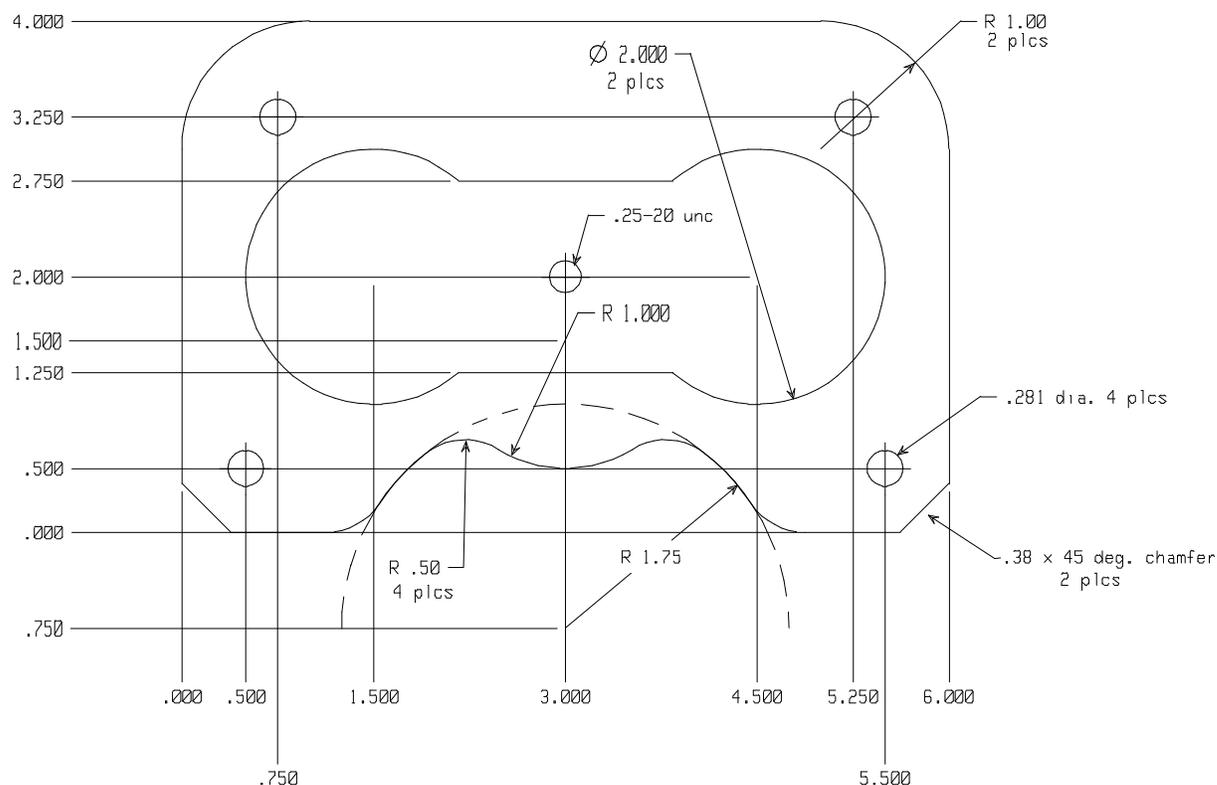
Dynamic Input Keyboard Shortcuts

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.

Exercise 1: Circles and holes with Dimensions



Construct Outside Contour

Start a new file

Click on **File**→**New**

START NEW JOB FILE Yes or No? REP=Y

With the left mouse button click on the word **Yes**.

CAD SETTINGS FILE NAME?

Choose the **default_in** cad settings file.

Change Graphical Mode

From the main menu, choose **Settings**→**Variables**. Highlight **Graphical mode=On** then press OK. This will change the setting for the graphical mode to off.

GraphicalMode=0ff

Choose cancel to close the select box.

This will turn off Synergy's Dynamic mode, which makes these exercises less complicated for the beginner user. Hit Cancel to close the named variables menu.

Place a point on the screen

Click Points → Input XY, directly under Construct, located on the tab menu. The Prompt line will ask you where you want your point located. Enter 3 in the Input line for ABS X, hit enter, Enter 2 in the Input line for ABS Y, hit enter.

The questions for ABS X and ABS Y will keep repeating. To get out of this loop you can either choose a new menu pick or choose **Misc → Leave Script**.

Click on ▲ in the main menu. Then click on Design: **Construct → Lines → Point Angle**

```
ANGLE : 0
LINE LOCATION ABS X? 0
LINE LOCATION ABS Y? 0
LINE LOCATION ABS Z? 0
ANGLE: 90
LINE LOCATION ABS X? 0
LINE LOCATION ABS Y? 0
LINE LOCATION ABS Z? 0
```

You can choose **MISC → Leave Script** to get out of the Line questions or by choosing another menu pick the questions will go away.

Construct line parallel at specified parallel distance

Design (you're already there): **Construct → Lines → Parallel**

```
PARALLEL DISTANCE: 4
```

PICK PARALLEL LINE?

Pick the horizontal line, then pick above the horizontal line

Input New distance:

```
PARALLEL DISTANCE: 6
```

Pick the vertical line, then pick on the right side of the vertical line

Construct circle defined by radius and center

Design: **Construct → Circles → Rad: Center**

```
CIRCLE RADIUS: 1.75
CIRCLE CENTER ABS X: 3,-.750,0
CIRCLE RADIUS: 1
CIRCLE CENTER ABS X: 3,1.5,0
```

Split element

Design: **Construct → Trim → Split**

PICK ELEMENT TO BE SPLIT?

a. Select the lower horizontal line as the element to be split

SPLIT POINT: PICK NEAREST ELEMENT?

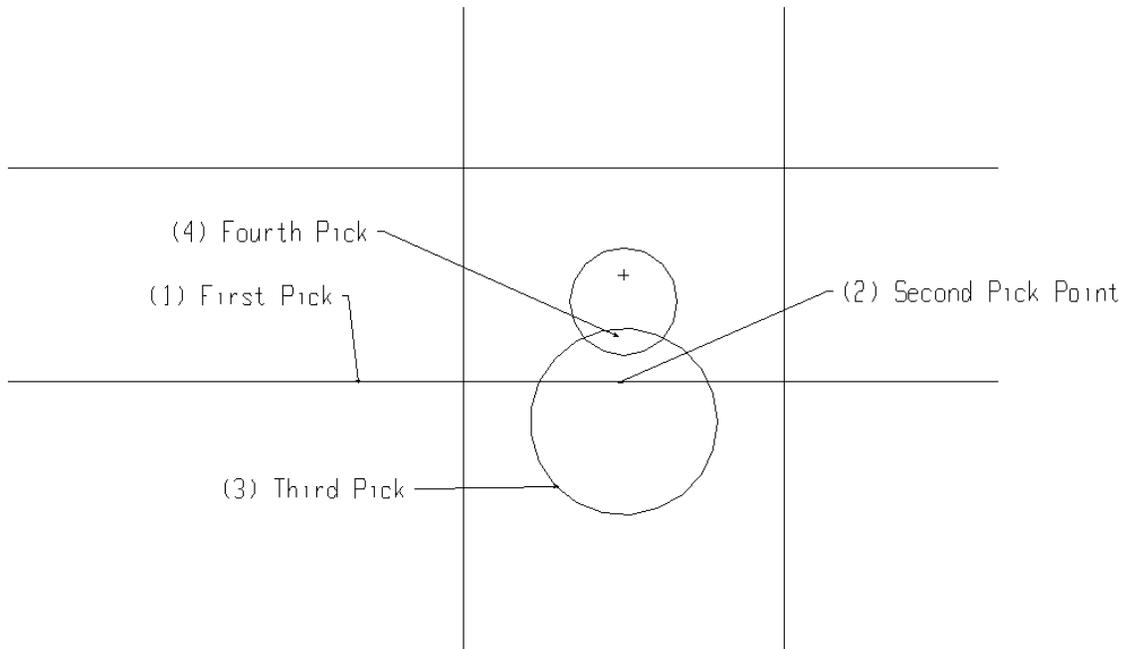
b. Pick the split point on the line near the middle of the 1.75 radius circle.

PICK ELEMENT TO BE SPLIT?

c. Select the 1.75 radius circle as the element to be split.

SPLIT POINT: PICK NEAREST ELEMENT?

d. Pick its split point at the top of the circle at 90 degrees.



Construct corner fillet

Design: Construct → Trim → Fillet

FILLET RADIUS : .5

PICK ELEMENT TO BE TRIMMED, None, Single, Radius

Note: When putting in fillets, always pick the portion of the geometry you want to save

Pick the left side of the lower horizontal line.

PICK SECOND ELEMENT

Pick the left side of the 1.75 rad. Above the lower horizontal line.

PICK ELEMENT TO BE TRIMMED, None, Single, Radius

Pick the left side of the 1.75 rad.

PICK SECOND ELEMENT

Pick the lower left side of the 1.0 rad.

PICK ELEMENT TO BE TRIMMED, None, Single, Radius

Pick the right side of the 1.75 rad above the lower horizontal line.

PICK SECOND ELEMENT

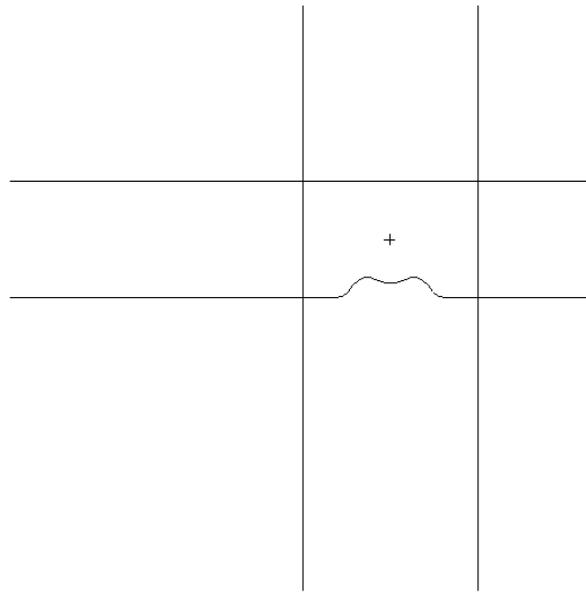
Pick the lower right side of the 1.0 rad.

PICK ELEMENT TO BE TRIMMED, None, Single, Radius

Pick the right side of the lower horizontal line.

PICK SECOND ELEMENT

Pick the right side of the 1.75 rad.



PICK ELEMENT TO BE TRIMMED, None, Single, Radius
(Click on the word *Radius*)

FILLET RADIUS : 1.0

PICK ELEMENT TO BE TRIMMED, None, Single, Radius
Pick the left side vertical line below the upper horizontal line.

PICK SECOND ELEMENT

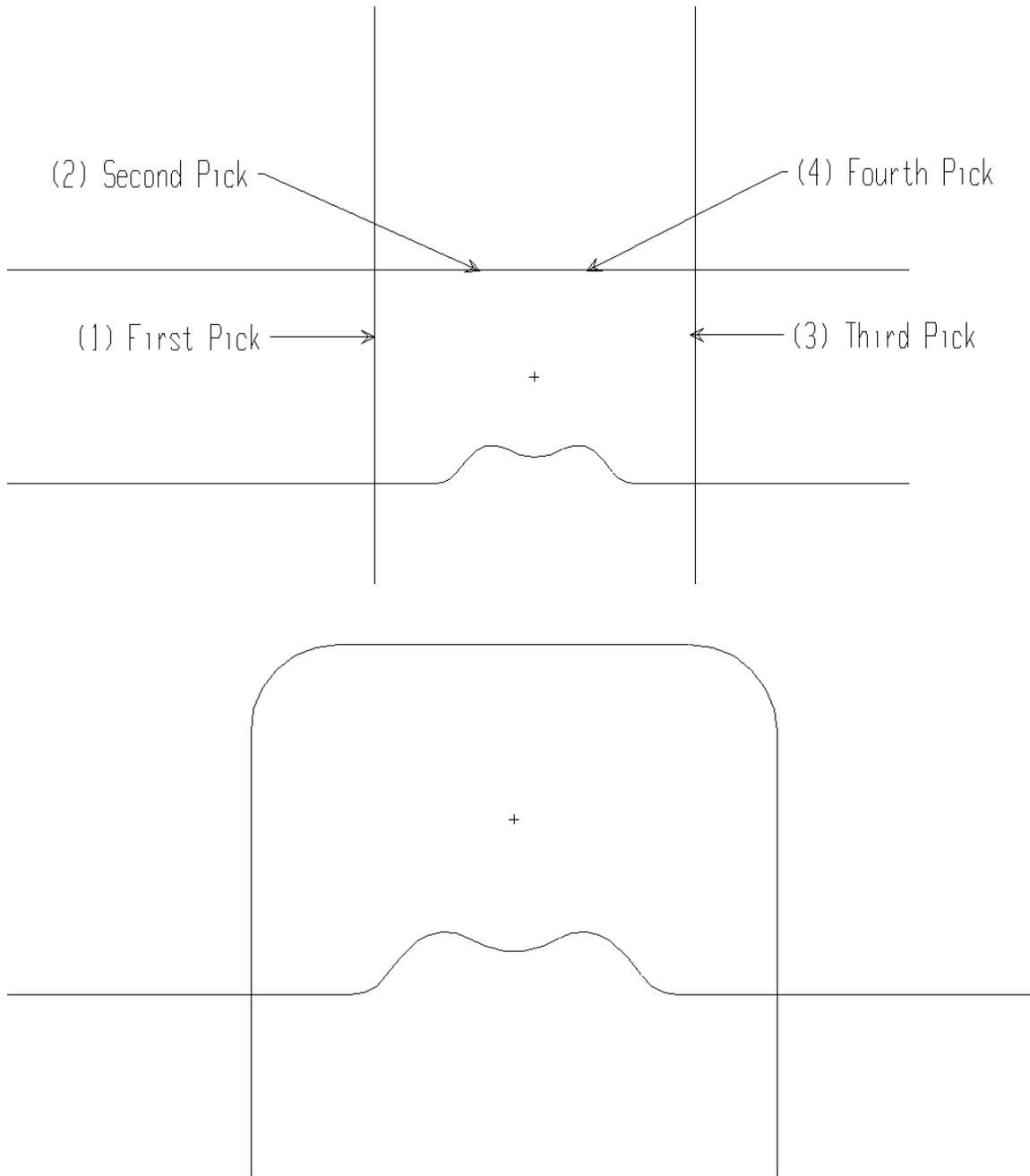
Pick the upper horizontal line making sure to pick the portion of the elements you want to save.

PICK ELEMENT TO BE TRIMMED, None, Single, Radius

Pick the right side vertical line below the upper horizontal line.

PICK SECOND ELEMENT

Pick the upper horizontal line.



Construct corner chamfer

Design: Construct → Trim → Chamfer

CHAMFER LENGTH: .38

PICK ELEMENT TO BE TRIMMED, None, Single, Radius

Pick the left side of the lower horizontal line (near the 1.75 rad.)

PICK SECOND ELEMENT

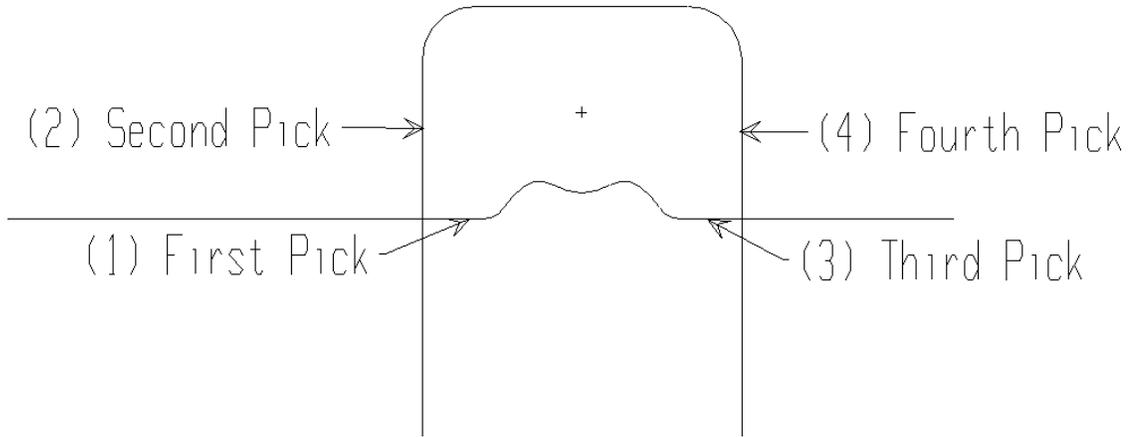
Pick the left side vertical line.

PICK ELEMENT TO BE TRIMMED, None, Single, Radius

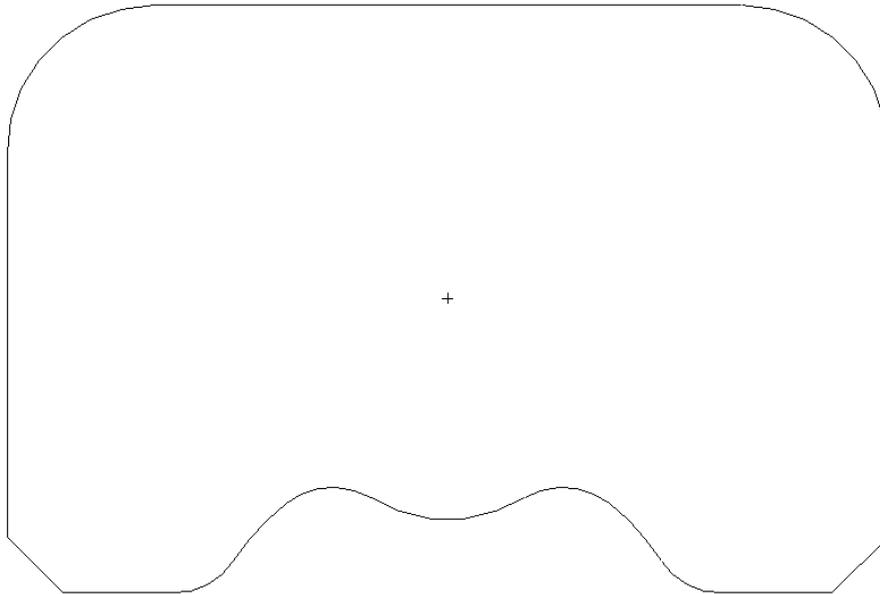
Pick the right side of the lower horizontal line (near the 1.75 rad.)

PICK SECOND ELEMENT

Pick the right side vertical line.



Choose Refresh →Fit



Construct Inside Contour

Construct circle defined by radius and center

Design: Construct → Circles → Rad: Center

CIRCLE RADIUS : 1

CIRCLE CENTER: 4.5,2,0

CIRCLE RADIUS: 1

CIRCLE CENTER: 1.5,2,0

Construct line parallel at specified parallel distance

Design: Construct → Lines → Parallel

PARALLEL DISTANCE : 1.25

Pick a section of the lower horizontal line, then pick in the area above that line.

PARALLEL DISTANCE : 2.75

Pick a section of the lower horizontal line, then pick in the area above that line.

Trim two intersecting elements

Trim (to sharp edges) the inside intersections of the lines with the 2.0 DIA. circles.

Design: Construct → Trim → Corner

PICK FIRST ELEMENT TO BE TRIMMED?

Click on the top line between the 2 circles.

PICK NEXT ELEMENT TO BE TRIMMED?

Click the left side circle above the top line.

PICK FIRST ELEMENT TO BE TRIMMED?

Pick on the top line between the 2 circles.

PICK FIRST ELEMENT TO BE TRIMMED?

Click the right side circle above the top line.

PICK FIRST ELEMENT TO BE TRIMMED?

Pick the lower line between the 2 circles.

PICK NEXT ELEMENT TO BE TRIMMED?

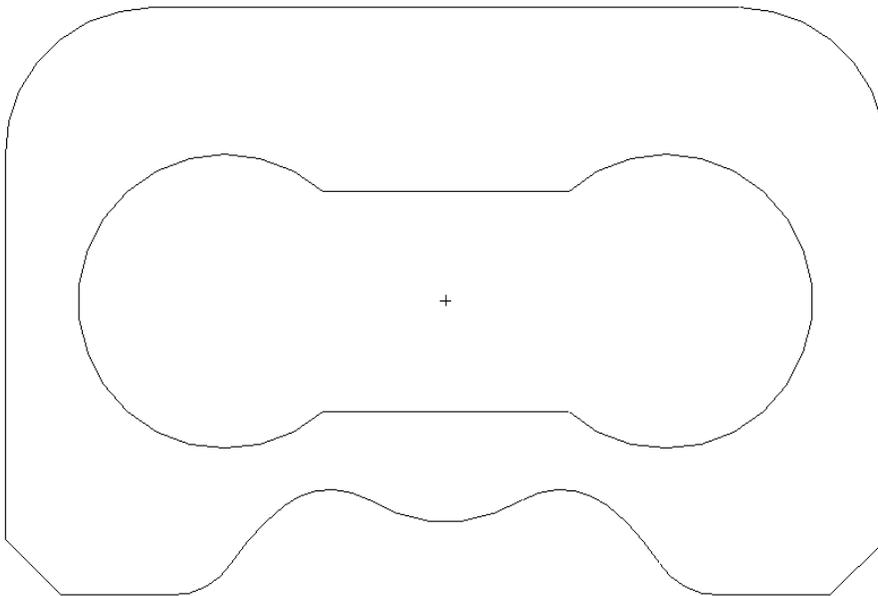
Pick the left side circle below the lower line.

PICK FIRST ELEMENT TO BE TRIMMED?

Pick the lower line between the 2 circles.

PICK NEXT ELEMENT TO BE TRIMMED?

Pick the right side circle below the lower line.



Place Circles

Place hole locations by creating Circles

DESIGN → Construct → Circles → Rad: Center

CIRCLE RADIUS: $.281/2$

Place the circles at the following locations:

Circle one:

CIRCLE CENTER ABS X: $.5$

CIRCLE CENTER ABS Y: $.5$

CIRCLE CENTER ABS Z: 0

Circle two:

CIRCLE RADIUS : Hit the repeat button. (Middle mouse button)

CIRCLE CENTER ABS X: $.75$

CIRCLE CENTER ABS Y: 3.25

CIRCLE CENTER ABS Z: 0

Circle three:

CIRCLE RADIUS : Hit the repeat mouse button.

CIRCLE CENTER ABS X: 5.25

CIRCLE CENTER ABS Y: 3.25

CIRCLE CENTER ABS Z: 0

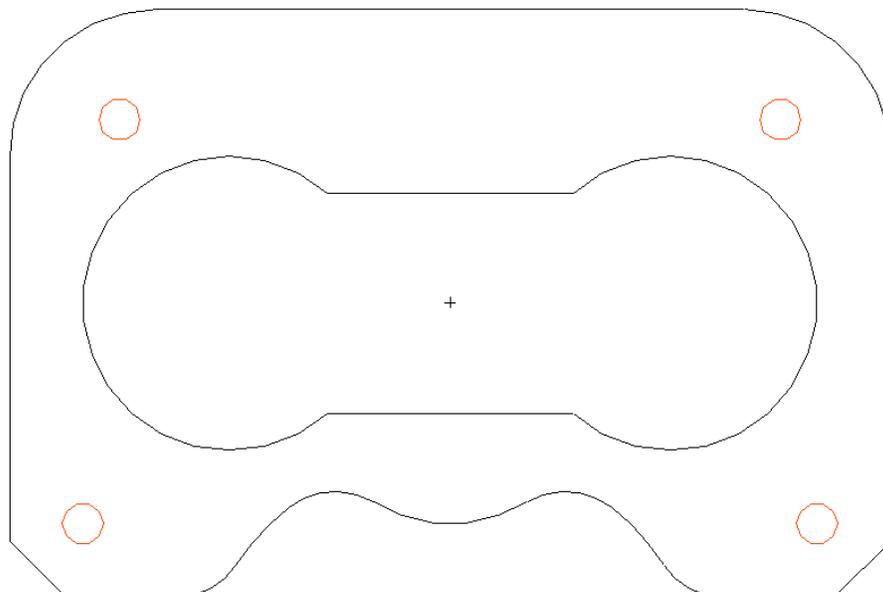
Circle four:

CIRCLE RADIUS : Hit the repeat mouse button.

CIRCLE CENTER ABS X: 5.5

CIRCLE CENTER ABS Y: $.5$

CIRCLE CENTER ABS Z: 0



AUTO LINKING

Autolinking lets you join independent geometry together. Autolinking also tests to see that this geometry is properly connected for machining. Create a piece of 2D geometry, and then use Autolinking to quickly create a contour.

Create the contour

Choose **DESIGN** (you're already there): → **Construct** → **Group** → **Autolink** <Ctrl-A>

PICK START ELEMENT, Single?

The system is asking you to indicate where your contour should start. Pick anywhere on the outside contour. The prompt line will then read:

PICK END ELEMENT?

Your next click will tell Synergy where you want the contour to end. Pick the same location. You will notice that Synergy placed directional arrows on all of those entities. This ensures that all entities, regardless of how they were created, are all traveling in the same direction.

The Prompt line now reads:

PICK NEXT, Backup, Single, New?

You want to stop, move the tool, and start an entirely different contour so you need to click with the left mouse button on the word **New**. Autolink the inside contour by clicking the same location twice.

Misc → **Leave Script**

Now choose **Main Menu: Inquire** → **Highlight**. Pick the outside contour. This will show the whole contour has been highlighted. Use the **Inquire** → **Highlight** on each contour to verify it is Autolinked. You must choose the **Inquire** → **Highlight** for each contour because it only allows you to pick one at a time.

Name the Contour

Give the outside contour a name

DESIGN: Construct → **Group** → **Name**

PICK THE CONTOUR TO BE NAMED:

Click on the outside contour.

CONTOUR NAME:

Give the contour a name by typing in the input prompt. Call the outside contour: `outside_contour`.

Give the inside contour a name

PICK THE CONTOUR TO BE NAMED:

Click on the inside contour.

CONTOUR NAME:

Give the contour a name by typing in the input prompt. Call the inside contour: `inside_contour`.

Choose **Main Menu: Inquire** → **Contour** and pick the contour to see the contour information which includes the names.

Create a Group – later this group will be used to apply holes by a single key stroke.

Group the circles by common diameters

DESIGN: Construct → Group → Create → Circle Diameter

Diameter to Group By?

Click on any of the .281 circles you just created

Layer Number, current layer=0?

Click on any .281 circles you just created.

This question is asking you to indicate what layer the circles are on now. By left clicking on one of the circles, the system will grab the layer number that the circles are currently on.

Group Name?

Name the group: 281_holes

Color to Assign Group?

Pick the color that the group will be – you must enter a number for the color. Refer to the color controls. 0 is black, 1 is white, 2 is red, etc.

Layer to Assign Group?

This is the layer that the Group will appear on. Type 1.

You can assign layers and colors specific to the group at this point. If you click the REP key, Synergy will accept the defaults.

You will notice that all the circles of that diameter turned the color that you assigned the group. Grouping of circle diameters can be very useful when working on a job that came through in IGES with many circles that need operations applied to them.

Archive your job

It's time to save the geometry that you've created. You can do this by using Synergy's Archive feature. The file archive is designed to store the Synergy file along with all associated file components into a directory structure.

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 Exercise1 by typing into the input line, then press <Enter>.

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

Project name: Exercise1

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.

Review: Drawing Exercise 1

Here are the five steps that you used while creating the geometry in exercise one.

1. You drew the geometry using synergy's geometry construction techniques.
2. You placed circles to indicate where your holes are going to be.
3. You created a group of circles to use later for hole features.
4. You autolinked the geometry to create contours for milling.
5. You've named the contours.
6. You've archived your work.

Manufacturing:

Exercise 1: Circles and Holes

Hole Features

Hole features are a way to place a hole or group of holes that contain all the relevant manufacturing data in a single instance.

Place hole features for the .281 group

Click ▲ on main menu, then choose Variational: **VMill → Holes → Drills**

VARIATIONAL OBJECT NAME?

Type in Exercise.mfg.

FEATURE FILE NAME?

Click on twistdr.2812 and hit enter.

Drill depth?

Hit the middle mouse button to except the REP value of 1.0.

Hole type (0)BLIND or (1)THRU?

Type 1 <enter> for THRU.

PICK HOLE CENTER POINT, Group, New?

Click with the left mouse button on the word **Group**.

PICK GROUP, Point, New? REP=P

With the left mouse button, click on one of the circles that you previously grouped.

You will notice that Synergy placed the drilling hole feature on all four of the holes with one pick. This hole feature consists of a .5000 Spot drill and a .2812 Twistdrill. The hole features are totally user configurable to way your company manufactures.

Place hole features for the second group.

Choose **VMill → Holes → Dowels**

FEATURE FILE NAME?

Choose the dowel.250 hole feature and click OK.

Drill depth?

Type .5 and hit <enter>

Ream depth?

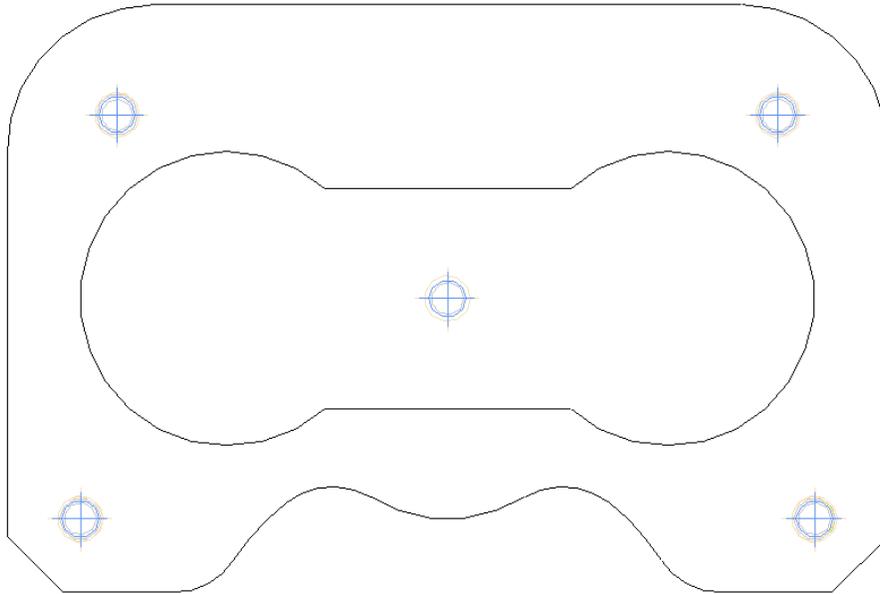
Type .25 and hit enter.

Hole type (0)BLIND or (1)THRU?

Type 0 and hit <Enter>

PICK HOLE CENTER POINT, Group, New?

With the left mouse button click on the point in the center of the part.



Generate Cutter Paths

Machining the inside and outside contours

Choose **Vmill** → **Path** → **Finish Mill**

INPUT TYPE OF TOOL?

Choose endmills

INPUT TOOL FILE NAME?

Choose endmill_0.65

TOOL NAME, Default?

Click on the word Default with the left mouse button.

INPUT HOLDER TYPE, None?

Choose the bt40.inch.

MILL Climb OR Conventional :

Click with the middle mouse button to except CL.

FINISH ALLOWANCE :

Type .02 and then hit enter.

PICK AN ELEMENT OF CONTOUR TO BE OFFSET:

Pick an element of the outside contour, then pick in the area outside of that contour to determine which side will be machined.

FINISH DEPTH: .5

Pick an element of the inside contour, then pick in the area inside of that contour to determine which side will be machined.

FINISH DEPTH : .5

Vmill → Path → Finish Mill

This will pop up a selections box where you will select the endmill that you selected in step 2.
Choose .65 Endmill.

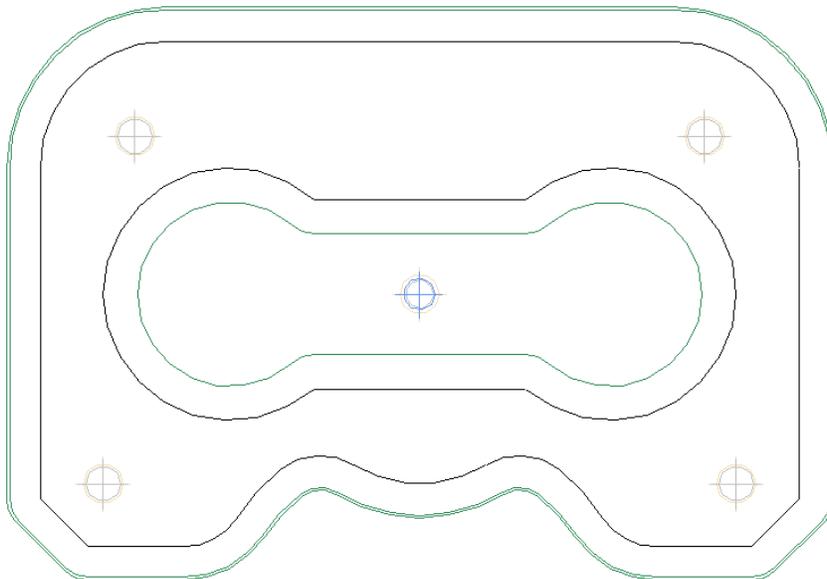
MILL Climb or Conventional: CL

FINISH ALLOWANCE: 0

PICK AN ELEMENT OF CONTOUR TO BE OFFSET?:

Pick an element of the outside contour, then pick in the area outside of that contour to determine which side will be machined.

FINISH DEPTH: .5



Remove Duplicate tools

Variational: VMill → Tools → Remove Duplicate Tools

When holefeatures are used the tools are loaded for each hole that you put in. When running the turbopost manually (Not using AMO), you must remove the duplicate tools or you will have duplicate tools in the tool section box when posting the job.

Turbopost the Job

TURBO POST setup consists of a series of ASCII files which configure the NC output, define user variables, user commands, templates and macros; and define question sequences. Posting the job will create CNC tapecode, time calculations, tooling lists and a cost analysis.

To TurboPost the job, choose **Variational: Vmill → Execute → Turbopost**

From the selections box, choose the “globalhmill.” Then select “New” from the command file types presented in the selections box.

The Prompt line will tell you to select the post command.

Enter these Turbopost commands:

```
STARTUP :  
ENTER THE PROGRAM DESCRIPTION: Exercise 1  
ENTER THE PROGRAM NUMBER: 1  
ENTER THE TOOL NUMBER: 1  
ENTER THE NEXT TOOL NUMBER: 2  
ENTER THE INITIAL Z VALUE: 1.0
```

Choose the .500 Endmill from the selections box as the first tool.

```
SPINDLE SPEED: 1175  
CUT_CONTOUR :  
PICK PATH, Proceed?
```

Pick all three of the cutterpaths in the order that you want to machine them. Then choose **proceed**.

```
TOOLCH :  
ENTER THE NEXT TOOL NUMBER: 3  
ENTER THE INITIAL Z VALUE: 1.0
```

Choose the .5 spotdrill as the next tool.

```
SPINDLE SPEED: 1175  
FEED RATE: 2.35  
DRILL :  
PICK HOLE, Sequential, Group, Manual, Proceed?
```

Pick the five center dilled holes. Then choose **proceed**.

```
TOOLCH :  
ENTER THE NEXT TOOL NUMBER: 4  
ENTER THE INITIAL Z VALUE: 1.0
```

Choose the .2812 Twist Drill.

```
SPINDLE SPEED: 1175  
FEED RATE: 2.35  
PECK_DRILL :  
PECK DISTANCE: .1  
PICK HOLE, Sequential, Group, Manual, Proceed?
```

Pick the four twist drilled holes. Then choose **proceed**.

```
TOOLCH:  
ENTER THE NEXT TOOL NUMBER: 5  
ENTER THE INITIAL Z VALUE: 1.0
```

Choose the .2187 twist drill.

```
SPINDLE SPEED: 1175  
FEED RATE: 2.35  
PECK_DRILL:  
PECK DISTANCE: .1  
PICK HOLE, Sequential, Group, Manual, Proceed?
```

Choose the single twist drilled hole. Then choose **proceed**.

```
TOOLCH  
ENTER THE NEXT TOOL NUMBER: 6  
ENTER THE INITIAL Z VALUE: 1.0
```

Choose the .2500 reamer.

```
SPINDLE SPEED: 1175  
FEED RATE: 2.35  
DRILL:
```

Pick the single reamed hole. Then choose **proceed**.

```
END_CODES:  
CANCEL.  
SELECT POST FUNCTION : EXECUTE COMMAND FILE  
SELECT POST FUNCTION : CANCEL
```

Load tool images:

Variational: VMill → Images → Make all Mill

Show Documentation

Main Menu: Misc → NC Docs

The NC docs will show you a Time analysis, Tooling list, Cost analysis and the NC code.

Archive

Main Menu: File → Archive → Save → Archive

REVISION 2 NOTE: None:

Type added manufacturing.

Construct Outside Geometry

Begin a new file

Main Menu: File → New
Choose the Default CAD SETTINGS

Draw axis lines

Design: Construct → Lines → >Axis Lines

Construct circle defined by radius and center

Design: Operations → Circles → Rad: Center

CIRCLE RADIUS : 1.466

INPUT ABS X : 0

INPUT ABS Y : 0

INPUT ABS Z : 0

CIRCLE RADIUS : .896

INPUT ABS X : 0

INPUT ABS Y : -1.41

INPUT ABS Z : 0

CIRCLE RADIUS : .25

INPUT ABS X : -2.2

INPUT ABS Y : -1.87

INPUT ABS Z : 0

CIRCLE RADIUS : .19

INPUT ABS X : -1.91

INPUT ABS Y : .88

INPUT ABS Z : 0

CIRCLE RADIUS : .19

INPUT ABS X : 0

INPUT ABS Y : 1.625

INPUT ABS Z : 0

Tangent line trim

Design: Construct → Trim → Tangent Line

PICK FIRST TANGENT ELEMENT:

Pick on the lower right side of the 1.466 rad.

PICK NEXT TANGENT ELEMENT:

Pick on the lower right side of the .896 rad.

Tangent circle trim

Design: Construct → Trim → Tangent Circle

TANGENT CIRCLE RADIUS : .5

PICK FIRST TANGENT ELEMENT:

Pick the upper left side of the 1.466 rad.

PICK NEXT TANGENT ELEMENT:

Pick on the upper right side of the left hand .19 rad.

Construct line tangent to element at angle

Design: Construct → Lines → Tangent Lines → Known Angle: Tangent

PICK TANGENT ELEMENT:

Pick on the bottom of the .25 rad.

ANGLE OF TANGENT LINE: 0

PICK TANGENT ELEMENT:

Pick on the left side of the .25 rad.

ANGLE OF TANGENT LINE : 100

PICK TANGENT ELEMENT:

Pick on the upper left side of the left hand .19 rad.

ANGLE OF TANGENT LINE : 45

PICK TANGENT ELEMENT:

Pick on the upper left side of the right hand .19 rad.

ANGLE OF TANGENT LINE : 90-37.5

PICK TANGENT ELEMENT:

Pick on the upper right side of the right hand .19 rad.

ANGLE OF TANGENT LINE : 90+37.5

Construct corner fillet

Design: Construct → Trim → Fillet

FILLET RADIUS : .2

Place the fillet on the left side between the bottom horizontal line and the .896 rad.

PICK THE ELEMENT TO BE TRIMMED : Radius

FILLET RADIUS : .25

Place the fillet between the 10 degree and the 45 degree angled lines.

Trim two intersecting elements

DESIGN: Construct → Trim → Corner

Corner trim both sides of the bottom left .25 rad., Then corner trim the three remaining untrimmed lines to their respective .19 radii.

2D rotate multiple elements

DESIGN: Rims → Rotate → Multiple Elements → 2D Duplicate

PICK ELEMENTS TO BE ROTATED:

Pick the top .19 rad. and the two remaining untrimmed lines.

PICK ELEMENTS TO BE ROTATED, Backup , Proceed : P

REFERENCE POINT ABS X : 0

REFERENCE POINT ABS Y : 0

ROTATION ANGLE : -60

NUMBER OF TIMES : 1

2D invert multiple elements

Design: Rims → Invert → Multiple Elements → 2D Duplicate

PICK ELEMENTS TO BE INVERTED:

Pick the top .19 rad. and the two untrimmed lines same as previous step.

PICK ELEMENTS TO BE INVERTED, Backup , Proceed : P

REFERENCE POINT ABS X : 0

REFERENCE POINT ABS Y : $1.625 - (2.34 + 1.625) / 2$

INVERSION ANGLE : 0

Split element

Design: Construct → Trim → Split

Split the 1.466 rad. in two places in line with the .19 radii at 30 degrees and 90 degrees, then split the .896 rad. in line with the .19 rad. at 270 degrees.

Construct corner fillet

Design: Construct → Trim → Fillet

FILLET RADIUS : .2

Fillet the remaining untrimmed lines to the 1.466 and .896 radii.

Construct large pocket**Change Layer**

Create Layer 1 by middle mouse clicking in the white part of the layer area that says 0. You will know if you have created a new layer when it appears in the layer controls. Your current layer will be highlighted. You can also check to see what layer you are on by checking the info menu from the main menu.

Construct circle defined by radius and center

Design: Construct → Circles → Rad: Center

CIRCLE RADIUS : 1.34

INPUT ABS X : 0

INPUT ABS Y : 0

INPUT ABS Z : 0

CIRCLE RADIUS : .77

INPUT ABS X : 0

INPUT ABS Y : -1.41

INPUT ABS Z : 0

Construct line tangent to two entities

Design: Construct → Lines → Tangent Lines → Two Tangents

PICK FIRST TANGENT ELEMENT:

Pick the left side of the .77 rad.

PICK NEXT TANGENT ELEMENT:

Pick as far as you can to the right side of the .5 rad.(outside contour) so that the system will locate an implied tangency point.

Construct corner fillet

Design: Construct → Trim → Fillet

FILLET RADIUS : .25

Place the fillet between the new line and at the top of the the 1.34 rad.

Tangent line trim

Design: Construct → Trim → Tangent Line

Place the tangent line between the 1.34 and .77 radii on the right side.

Trim element

DESIGN → Construct → Trim → Element

Trim the .77 rad. to the line on the left side.

Construct small pocket**Change Layer**

Middle click the [+] button on the Layer controls on the interface. Then enter a new layer number in the pop-up dialog box. In this case, create layer 2. You will know if you have created a new layer when it appears in the layer controls.

Construct line parallel at specified perpendicular distance

Design: Construct → Lines → Parallel

PARALLEL DISTANCE : .25

1. Pick the large pocket left side line, then pick to the left of that line.

PARALLEL DISTANCE : .25

2. Pick the 45 degree line (outside contour), then pick to the inside of the outer contour.

PARALLEL DISTANCE : .25

3. Pick the 100 degree line (outside contour), then pick to the inside of the outer contour.

Construct line at specified angle and distance from point

Design: Construct → Lines → 2D Lines → Point: Dist: Angle

Pick the .77 rad. on the large pocket.

PERPENDICULAR DISTANCE : .625

ANGLE OF LINE : -10

PICK LEFT/RIGHT OF LINE:

Pick in the area below the newly created line.

Pick the .77 rad. on the large pocket.

PERPENDICULAR DISTANCE : 2

ANGLE OF LINE : 80

Pick in the area to the left of the newly created line.

Construct a line parallel at a perpendicular distance

Design: Construct → Lines → Parallel

PARALLEL DISTANCE : 1.05

Pick the bottom -10 degree line, then pick in the area above that line.

Construct concentric circle

Design: → Construct → Circles → Additional Methods →

Offset Circle

PARALLEL DISTANCE : .25

Pick the .5 rad. on the outside contour, and then pick in the area outside of the .5 rad (inside of the outer contour).

Construct corner fillet

Design: Construct → Trim → Fillet

FILLET RADIUS : .25

Fillet trim all the elements to complete the small pocket.

Place Circle Geometry**Change your current layer**

On the layer controls, click on the number that you want to make your current layer. Change back to layer zero. You can check your current layer on the info menu.

Select a new line type

From the property controls, click the line type menu to get a pop up menu for picking a new line type.

Select the dotted line, line type 2.

Construct circle tangent to three elements

Design: Construct → Circles → Three Tangents

Pick the 100 degree line on the outside contour, then the other two tangent lines of the small pocket to place the hidden line circle.

Reset Original line type

Use the line type menu to return to your original line-type. (Line type 1)

Construct line parallel at specified perpendicular distance

Design: Construct → Lines → Parallel

PARALLEL DISTANCE : .125

Pick the line on the left side of the large pocket, then pick to the left of that line.

Design: → Points → Additional Points → Element and Y value

INPUT ABS Y: -1.87

PICK INTERSECTING ELEMENT:

Pick anywhere on the parallel line that you just created.

Autolink Contours

Be sure that the untrimmed line has been deleted. This will ensure that the Autolink feature works correctly.

Autolink around the outside geometry

Design: Mill → Path → Autolink

Pick anywhere on the outside geometry, and then pick again to indicate the contour's direction.

PICK NEXT , Backup, Single, New : N

Autolink around the large pocket.

Pick "New" from the prompt line, then pick anywhere on the large pocket. Pick again to indicate the contour's direction.

PICK NEXT , Backup, Single, New : N

Autolink around the small pocket.

Pick "New" from the prompt line.

Pick anywhere on the small pocket, and then pick again to indicate the contour's direction. Name this contour: "inside_right"

Name the Autolinked Contours

RIMS → Group → Name

Name the 3 contours you just created.

Crosshatching**Hatch the area inside exterior contour and outside interior islands**

Design: Drafting → Hatching → Crosshatch

PICK EXTERIOR CONTOUR:

Pick the outside contour.

NUMBER OF ISLANDS : 2

PICK ISLAND #1:

Pick one of the pockets.

PICK ISLAND #2:
Pick the other pocket.

Dimensioning

Assign a new current layer

Create and switch to layer 5.

Create dimensions using reference point

Design: Drafting → Datum → Auto

DIMENSION ANGLE : 90

DATUM ABS X : 0

DATUM ABS Y : 0

DIM LOCATION: PICK FREE POINT:

Pick to the left side of the part where you want the dimensions located.

PICK FIRST CORNER OF PICK ZONE:

Pick opposing corners of a box that will include the entire part.

Construct linear vertical dimension

Design: Drafting → Standard → Vertical → Single

VERTICAL DIM START ABS X:

Pick the circle located at X0 Y0.

VERTICAL DIM END ABS X:

Pick the .19 rad at 30 degrees.

DIM LOCATION: PICK FREE POINT:

Pick to the right where you would like to place the dimension.

Change text orientation in a dimension.

Design: → DrftCtrl → Modify → Text → Orient

PICK DIMENSION:

Pick the .8125 dimension previously created. (When you are picking text, be certain to pick the upper left corner of the text.)

Construct radius dimension

Design: → Drafting → Circular → Radius

PICK CIRCLE:

Pick on the right side of the 1.466 rad., then place the dimension.

Construct line of text

Design: → Drafting → Text → Draw → Text

INPUT TEXT LINE : Enter some meaningful text

Using FREE POINT mode pick a point to the lower right of the part.

INPUT ABS X: F

Archive the job

Refer to the archive method described on page 35.

Exercise 2: Advanced 2D Manufacturing

Holes

Create tapped hole operation

Variational: VMill → Holes → Taps
Locate the tap.250-20NC and click <OK>.

DRILL DEPTH : 1
TAP DEPTH : .75
GROUP, Single, New : S

Individually place five of the .25-20 holes by picking the radii on the outside contour. (You must choose **S** between each hole.)

Create socket head cap screw operation

Variational: VMill → Holes → Cap Screws
Locate the feature: capscrew.375 and click <OK>

DRILL DEPTH : 0.5
GROUP, Single, New : S
Pick the hidden line circle to place the hole.

Load Tool

Load tools from tool library

Variational: VMill → Tools → Read Tool
Select a material, and select "endmills." Find and load the .500 endmill. Accept the default toolname.
Click Cancel twice to leave the read a tool selections box.

Pocket Milling

Create a new layer

Create and switch to layer 10.

Follow pocket with full look ahead

Variational: → VMill → Rough → Pocket Mill
Choose your milling tool.

MILL CLimb OR Conventional : CL
Width of cut: .3125
FINISH ALLOWANCE : 0.02
Pick the large pocket.

FINISH DEPTH: .5
DEPTH OF CUT: .5
PICK EXTERIOR CONTOUR:

Generate path for roughing area using zig-zag motion

Variational: VMill → Rough → Zigzag pocket
TOOL : .500 Endmill
MILL CLimb OR Conventional : CL
WIDTH OF CUT : 0.3125

FINISH ALLOWANCE : 0.02
 ANGLE OF ZIGZAG LINES : 0

Pick the line on the right hand side of the small pocket to define the angle of cuts.

PICK EXTERIOR CONTOUR:

Pick the small pocket.

NUMBER OF ISLANDS : 0

FINISH DEPTH: .5

DEPTH OF CUT: .5

Cutter offset with full look ahead

Variational: → Vmill → Path → Finish Mill w Leadin

TOOL : .500 Endmill <OK>

SELECT Leadin type: 1

SELECT Leadout type: 1

MILL CLimb OR Conventional : CL

FINISH ALLOWANCE : 0.02

PICK AN ELEMENT OF CONTOUR TO BE OFFSET:

Pick the outside contour, then pick to the outside of that contour.

FINISH DEPTH: .5

Modify Offsets

Edit offset data

Variational: → Vcontrol → Edit → Record Free Parameter

PICK VARIATIONAL RECORD, NUMBER?

Pick the outside contour cutter path. This will bring up the list of variables used in this record. Pick “Depth of Cut” from the selections box. At the prompt line, enter a new value for the variable. Enter “.25.” Then click “Refresh” from the prompt line.

Next, pick the Large pocket cutter path. Then click the “Width of Cut” variable. Change this variable’s value to .28, and click “Refresh” from the prompt line.

Create Tool Symbols

Create tool geometry

Create the tool geometry the same way you did in exercise one.

Images → Make All Mill

Archive

Main Menu: File → Archive → Save → Archive

Choose a Material

This step may not be necessary, particularly if you have already selected a material when you loaded the hole features.

Design: Mill → Tools → Choose a Material

Recalculate for Feeds and speeds

Design: Mill → Tools → Recalculate

Group holes according to associated tool

Design: Group → Create → Tool
INPUT TOOL ID NAME : .5000 spotdrill
GROUP NAME : sdrill
INPUT TOOL ID NAME : #7 Twist Drill
GROUP NAME : 7drill
INPUT TOOL ID NAME : .250_20NC Tap
GROUP NAME : tap
INPUT TOOL ID NAME : CANCEL

Generate NC code for applicable machining

Design → Mill → Execute → Turbo Post
TURBO POST NAME : globalhmill
SELECT COMMAND FILE TYPE : NEW
SELECT POST FUNCTION : SELECT POST COMMAND
SELECT COMMAND : 1 STARTUP:
ENTER THE PROGRAM DESCRIPTION : exercise 2
ENTER THE PROGRAM NUMBER: 2
ENTER THE TOOL NUMBER: 1
ENTER THE NEXT TOOL NUMBER: 2
ENTER THE INITIAL Z VALUE: 1.0
TOOL DESCRIPTION : .5000 spotdrill
SELECT COMMAND : DRILL:
PICK STARTING HOLE, Sequential, Group, Manual, Proceed : G
PICK HOLE GROUP, Proceed :

Pick one of the grouped holes shown.

PICK HOLE GROUP, Proceed : P
SELECT COMMAND : TOOLCH:
NEXT TOOL NUMBER 3
INITIAL Z VALUE: 2.0
TOOL DESCRIPTION : #7 Twist Drill
SELECT COMMAND : PECK_DRILL:
PECK DISTANCE : .12
PICK STARTING HOLE, Non-sequential, Group, Manual, Proceed : G
PICK HOLE GROUP, Proceed :

Pick one of the grouped holes shown.

PICK HOLE GROUP, Proceed : P
SELECT COMMAND : TOOLCH:
NEXT TOOL NUMBER: 4
INITIAL Z VALUE: 1.0
TOOL DESCRIPTION : .250_20Nc Tap
SELECT COMMAND : TAP:
PICK STARTING HOLE, Non-Sequential, Group, Manual, Proceed : G

Pick one of the grouped holes shown.

PICK HOLE, Sequential, Group, Manual, Proceed : P
SELECT COMMAND : TOOLCH:
NEXT TOOL NUMBER: 5
INITIAL Z VALUE: 1.0
TOOL DESCRIPTION : .750 Spot Drill
SELECT COMMAND : DRILL:

PICK STARTING HOLE, sequential, Group, Manual, Proceed :

Pick the hole shown.

PICK HOLE GROUP, Proceed : P
SELECT COMMAND : TOOLCH:
NEXT TOOL NUMBER: 6
INITIAL Z VALUE: 1.0
TOOL DESCRIPTION : .4062 Twist Drill
SELECT COMMAND : PECK_DRILL:
PECK DISTANCE : .12
PICK STARTING HOLE, Sequential, Group, Manual, Proceed :

Pick the hole shown.

SELECT COMMAND : TOOLCH:
ENTER THE NEXT TOOL NUMBER: 8
INITIAL Z VALUE: 1.0
TOOL DESCRIPTION : .625 Counterbore
SELECT COMMAND : COUNTERBORE:
PICK STARTING HOLE, Sequential, Group, Manual, Proceed :

Pick the hole shown.

SELECT COMMAND : TOOLCH:
NEXT TOOL NUMBER: 9
INITIAL Z VALUE: 1.0
TOOL DESCRIPTION : .500 Endmill
Z FEED RATE: 2.57
SELECT COMMAND : CUT_PATH_GRP:

PICK PATH, Proceed :
Pick the large pocket cutter path.

PICK PATH, Proceed : P
SELECT COMMAND : CUT_PATH_GRP:
PICK PATH GROUP, Proceed :

Pick one of the grouped small pocket cutter paths.

PICK PATH GROUP, Proceed : P
SELECT COMMAND : CUT_GRP_PGRPS:
PICK PATH, Proceed :

Pick the outside contour cutter path.

PICK PATH, Proceed : P
SELECT COMMAND : 2 END_CODES:
SELECT COMMAND : CANCEL
SELECT POST FUNCTION : EXECUTE COMMAND FILE
SELECT POST FUNCTION : CANCEL

Show Documentation

Main Menu: Misc → NC Docs

The NC docs will show you a Time analysis, Tooling List, Cost analysis and the NC tapecode.

Archive

Main Menu: File → Archive → Save → Archive

Export the Current tape file

Main Menu: File → Export → Current Tape file

Choose your post processor, or select Copy File. This will copy the tapefile to **C: \usr\weber\ST\tapes** and allow you to rename the tapefile.

Running AMO

Use Automated Manufacturing Output as a pre-processor

Using the Automated Manufacturing Output is by far the easiest and fastest method of generating your CNC code. However, any changes you wanted to make to the CNC code using AMO must be made manually. You can use AMO as a pre-processor in conjunction with Turbopost to greatly reduce the amount of manual editing you may need to do. Begin by running AMO on this exercise the same way you ran it in exercise two.

Manually Sort your Tasks

Design: Mill → Tasks → Sort

Mill the part

Choose the Horizontal AMO execute by clicking: Design: Execute → Global Hmill

AMO creates the command_file automatically based on the geometry and tooling information. You can use AMO to create the command_file, and then take that into the post processor to make any edits you may need. It's faster than running Turbopost manually because there is no need to pick commands off the menu or choose the geometry that you need to machine. Everytime you execute AMO the command_file is over written.

Making edits to the command file

Bring the command_file into Turbopost to perform any edits.

Design: → Mill → Execute → Turbopost

CHOOSE POST FORMAT :

Choose the "globalhmill" post format from the selections box.

CHOOSE COMMAND FILE TYPE : EXISTING

This will bring in the command_file that AMO just created.

SELECT POST FUNCTION : EDIT → EDIT COMMAND FILE

You can then execute the command file in Turbopost and keep your changes.

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

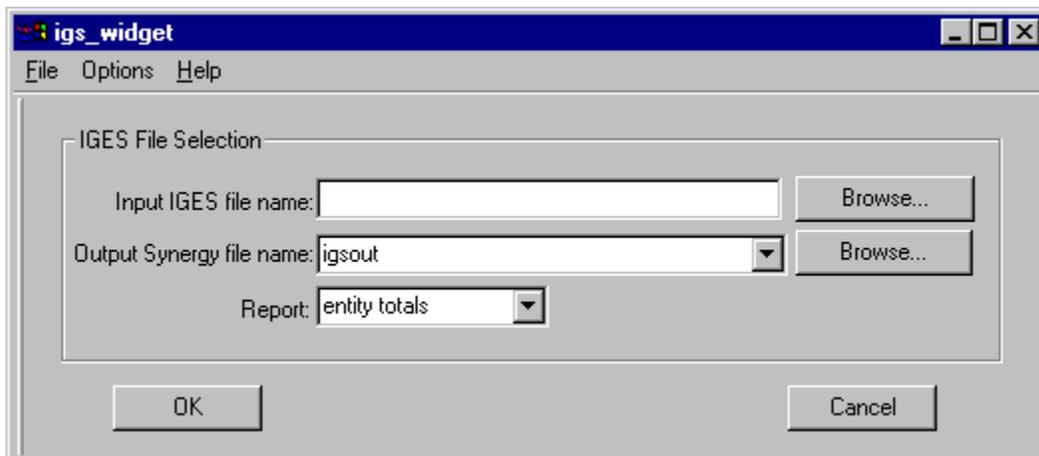
The IGES widget is a tool that gives you the ability to select which parts of an IGES file you want imported into Synergy. You have the option of not using this widget and importing the file in its entirety, however, there may be certain entities in some files that do not serve any purpose when imported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



The IGES widget

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 Entities:

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.

Libraries

Synergy comes with an extensive tools and materials library.

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 lathe 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`. Then you need to tell Synergy to run your own custom library by:

1. Modify the default values for the libraries in `/usr/weber/ascrpt/setconstants`. Then copy `setconstants` to `/usr/weber/contrib/ascrpt/`. Synergy will look under `contrib` for `setconstants` first.
2. Add the directory paths for the libraries to `/usr/weber/ascrpt/userconstants`. Then copy `userconstants` to `/usr/weber/contrib/ascrpt/`. `Userconstants` gets read in after `setconstants` and never gets modified by us.
3. Add the library directories to a Cad setting file and read it in when starting a new job. Copy `/usr/weber/database/properties/CadSettings.d/default_in` to another name. Add the following line:

```
SystemVars.tools dir: /usr/weber/util/tooldata/tools_in/
```

Change `tools_in` to be your custom tool library directory.

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.

Utility Kits

Utility Kits are special menus that you can create and customize with your own buttons.

Utility Kits allow you to instantly and effortlessly build your own menus "on-the-fly." You can either copy existing buttons or add new ones to your own menus. Once your custom menu is complete, you can save it for use in future Synergy sessions. It will be available at any time via Synergy's ▲ menu.

Create a Utility Kits

Edit → Interface → New Utility Kit

To create a menu, choose New Utility Kit. Type in a label and a new kit filename.

Click on CREATE.

You will then see a small box appear in Synergy. This is the new utility kit. Start adding buttons.

Add a Button to your new utility kit

Copy an existing button

Choose Edit → Interface → Copy a Button

1. Click on the button that you want to copy. This will create a "floating" button.
2. Drag the "floating" button and drop it into the Utility kit you want the button to appear in.

Delete a Button

Edit → Interface → Delete a Button

Click on the button that you want to delete. Be sure to close the Delete Button Dialog box when you are finished deleting buttons.

Delete a Utility Kit

Edit → Interface → Delete Utility Kit

1. Choose Edit → Interface → Delete Utility Kit.
2. Click the title bar of the utility kit that you want to delete.

Special Rules for Utility Kits:

You can not create a button that will toggle system functions. "On/Off" Toggle buttons are not supported and may crash the system. This feature is designed for standard button functions only.

System and User-Defined Hot Keys

HotKeys allow a user to enter a combination of keystrokes to be used as a hot key, which enables users to perform an action quickly.

Synergy provides a HotKey editor that allows users to define hot keys. A hot key is a key combination that sends a message equivalent to pressing a button. For example, by defining a hot key consisting of the CTRL+T key combination, a user could draw a Top View.

To define a hot key, use the HotKey editor to specify the combination of keys that will generate the desired action. The HotKey editor can also be used as a reference to view the current hotkey actions.

There are many predefined hotkey combinations called System hotkeys. For example, CTRL-H brings up the HotKey editor. The System hotkeys cannot be overridden by user hotkeys.

Using the HotKey Editor

The Copy/Add Button functions are available in the **Tools** drop-down menu in the HotKey editor.

++ Using the Utility Kit Copy Button function:

Users can copy existing interface buttons to Utility Kits or the HotKey editor. When Copy function is used, a copy of the button appears under the cursor and the user can drag the button to the desired location. The HotKey editor will fill in the Name and Action fields with the information in the button.

++ Using the Utility Kit Add Button function:

The Utility Kit "Add Button" function can be used to supply the name and action fields for a HotKey. This function also creates a draggable button that can be dropped in either editors or kits.

++ Defining the key combination:

To complete the HotKey definition, the user only needs to click the HotKey combination box and then press an unused combination of Shift/Ctrl/Alt and one other key. Any combinations already in use will not be displayed.

After all three fields are filled in, the user can use the Add button to append the HotKey to the user HotKey list.

A HotKey can also be defined by filling in all three fields manually.

++ Clear key

The Clear key clears the Combination, Name and Action fields.

++ Delete key

If the Combination, Name and Action fields exist in the user HotKey list, the Delete key will remove the entry from the list.

++ OK and Cancel buttons

Clicking OK will save any changes to the user HotKey list. Cancel abandons any changes made since the HotKey editor was first brought up.

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Notes

Some of the images used in this manual are based on early versions of the product. While we have made every attempt to keep these images as up to date as possible, there may be some cosmetic differences between what you see on your screen and what is contained in this tutorial.

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