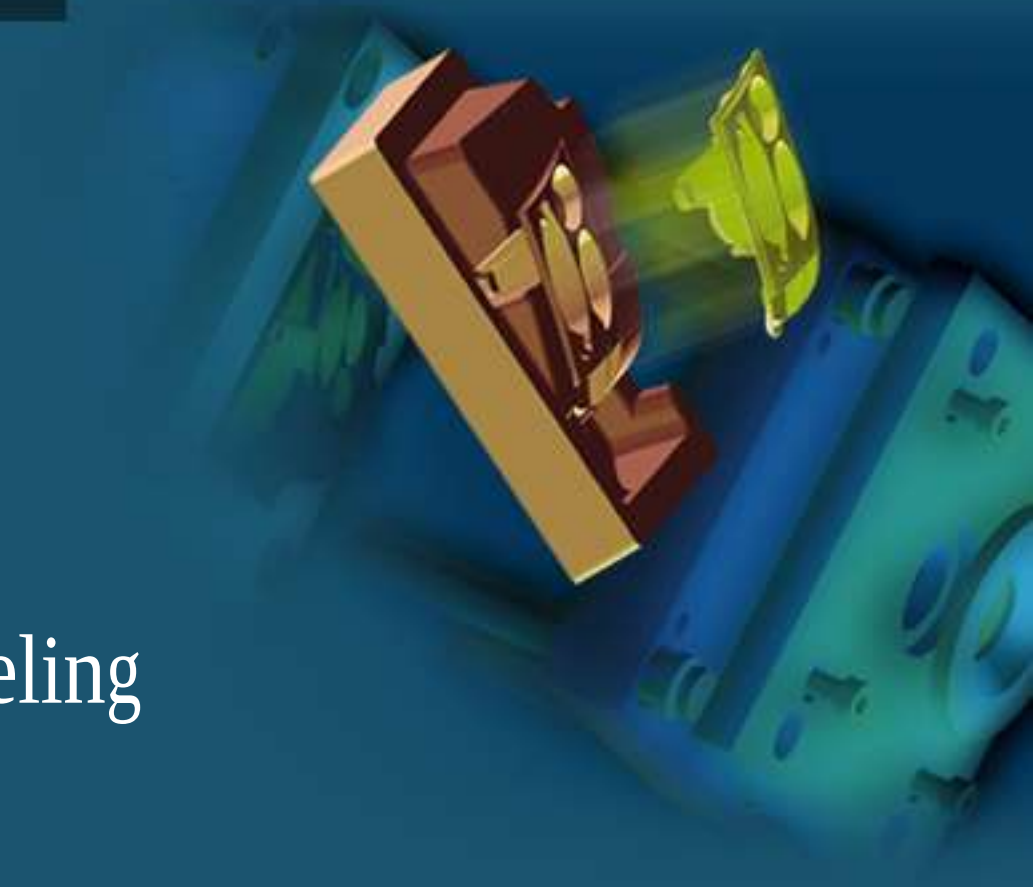
Cimatron

Cimatron
CAD/CAM Solutions for Manufacturing



Solid Modeling

Version 10.0

Data DI DFM NC Steel

Revised 1999.

Preface

Cimatron develops, markets and supports tools to automate the mechanical engineering process. Our systems support all phases of product development, with solutions for computer aided design (CAD) and manufacturing (CAM). **Cimatron**'s *integrated technology* approach combines design tools with optimized command output to computer-controlled manufacturing equipment. Drafting-table-to-shopfloor integration lets Cimatron clients realize dramatic efficiencies in product development and manufacturing.

Cimatron^{it} - Cimatron's flagship product - covers the entire spectrum of design, engineering and manufacturing processes, including:

- A complete range of wireframe, surface and parametric solid modeling tools with rendering capabilities;
- Advanced assembly, sub-assembly and part management and associative drafting functionality;
- Comprehensive, accurate data exchange interfaces;
- Powerful and intelligent NC applications for precise multi-axis machining.

The modular yet integrated structure of Cimatron^{it} grows to accommodate cutting edge tools and techniques.

Cimatron's automated engineering expertise benefits many industries, as competition requires tighter development cycles and efficient fabrication.

Powerful modules within Cimatron^{it} expand your system's capabilities. These may be purchased from your Cimatron representative.

This publication provides a detailed description of the major features of the appropriate Cimatron^{it} application/topic. It is intended to help users in the daily operation of Cimatron^{it}.

A list of Cimatron publications is shown on the next page.

Cimatron Documentation

Documentation is divided into Reference Manuals and Tutorials which together comprise a comprehensive guide to Cimatron[it].

For Cimatron companion packages, each Manual provides detailed instructions for its operation.

The list of Cimatron documentation is as follows:

	Publication	Description	
Cimatron[it] Reference Manual	Fundamentals & General Functions	Introduction to the fundamentals of Cimatron[it] and description of the General functions.	A O
	Modeling	Description of the wireframe and surface Modeling functions.	A O
	Drafting	Description of the Drafting functions.	A O
	Solid Modeling	Description of the Solid Modeling functions including Sketcher.	A O
	Numerical Control	Description of the NC functions.	A O
	General Post Processor	Description of the General Post Processor (GPP) functions.	A
	CimaDEK	Description of Cimatron's specialized Developer's kit, for programming customized functions.	A
	Finite Element Modeling	Description of FEM functions.	A O
	Utilities	Description of various utilities that may be used with Cimatron[it.] These utilities are either Internal, run internally through Cimatron, or **External**, run from the operating system environment.	A
	Data Interface Utilities	Description of data interface utilities DXF, DWG, IGES, JAMA-IS, VDA, PTC, STEP, SAT and CATIA.	A
	CimaRender Pro	A photo-realistic rendering package.	A
	MPDM: Administrator MPDM: User	Description of how to use Manufacturing Product Data Management to track and organize all files and data associated with a project.	A
	MoldBase 3D	Description of the functions associated with the detailed design of mold plates and components.	A
	Re-Enge	Description of Reverse Engineering design functions.	A

Cimatron[it] Tutorials	Design - covers the differences between versions 9 and 10.	A O
	Drafting - covers the differences between versions 9 and 10.	A O
	NC - covers the differences between versions 9 and 10.	A O
		A O
	MoldBase 3D	A O
	Simulator	A O

* Legend:

A Acrobat PDF File

O On Line

0

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GROUP >> EXPORT

An Export operation that enables you to extract objects from one file to another.

To perform the Export operation, you must do the following:

1. Enter the name of the target file (the file you will be exporting objects to)
2. Select the entities to be exported
3. Define the UCS origin

After executing and confirming these three steps, the target file will be created.

CREATE >> IMPORT

An Import operation that enables you to insert solid objects from another file into your current work file.

To perform the Import operation, you must do the following:

1. Enter the name of the source file (the file from which you will be importing objects)
2. Define the designated UCS origin

After executing and confirming these two steps, all objects will be imported from the source file and will be placed at the indicated position in you current work file.

DETAIL >> ROUND - Variable Radius

Two new options have been added: LINEAR / NON LINEAR.

These options define the linear/non linear changes of the radius value between every pair of given points.

COPY >> MOVE OBJECT

This new option enables you to move objects.

The following move methods are available:

DELTA

ROTATE

RELOCATE

MIRROR

MODIFY >> SCALE

A UNIFORM SCALE option has been added to SCALE. Now you just enter one scale value.

To perform this operation, you must do the following:

1. Indicate a reference point, from which all calculations will be performed.
2. Pick the entities to be scaled.
3. Enter a scaling value.

SURFACE >> DRIVE >> PIPE

The new PIPE option has been added.

This option enables you to create a parametric surface by driving a circle section of fixed or variable radius along a 3D trajectory.

DETAIL >> STITCH

A new stitching capability has been added for many objects.

The first object picked is considered as the initial object, other objects can be picked by using the MULTI PICK options.

MODIFY >> BREAK EDGE

This new option enables you to break a solid edge on 2 edges at the indicated point.

MODIFY >> ADJOIN FACE

The ADJOIN function has now been implemented in the SOLID environment.

This option enables you to close gaps in open solid objects by “repairing” faces according to reference faces.

VERIFY >> SOLID

The SOLID option has been added to the VERIFY menu. This option contains the following sub-options.

OBJECT	Enables you to receive general information regarding a picked object and to check this object as well.
FEATURE	Enables you to receive the following information regarding a feature: feature name, all options used to create it, and all feature references.
FACE/SURFACE	Enables you to verify a surface type and offset.
DATUM	Enables you to receive general information regarding the depth and normal vector of a datum plane, and about start point coordinates and direction of an axis.
UNDERCUT	Enables you to analyze all faces of solid bodies and to detect on solid faces areas with certain slope, which you can change from –90 to +90 degrees. Using this option, you can make a map of the slopes for a part, and detect areas on the faces where undercut will be smallest.

DISPLAY >> DISP. CURVES

The DISPLAY function is now available for solid faces and surfaces.

By DISP. CURVES enables you to set and/or change the number of display curves of solid face(s)/surface(s), as in Wireframe surfaces.

BLANK

The blanking and unblanking of solid and datum objects is now possible from the general BLANK function.

The previous BLANK option has been removed from the UTILITY function.

LINATT (COLOR)

Color changing for solid objects and datums is now available through the general LINATT function.

New solid objects and datum objects are created in the default colors (white and cyan); afterwards their color may be changed by LINATT.

Active PEN, LFONT and COLOR settings are valid also for the Sketcher both in CREATE and EDIT modes. This helps to clearly separate sketch elements from other displayed items.

The previous COLOR option has been removed from the UTILITY function.

SM (Select Mask)

Solids and Datum can now be masked from the SM Immediate Access function.

A new option has been added: SOLID

There are the following options for masking:

OBJECT –for solid objects and surfaces,

REF_AXIS

REF_PLANE

REF_CURVE

REF_POINT

PARTING_LINE

Display tolerance

A unified display tolerance parameter (DISPLAY TOL) has been included in FILE >> SETUP.

By changing the value of this parameter, you will be defining the display tolerance value in all subsequent executions of Wireframe and Solid functions.

The display tolerance for some existing elements (including solid objects) can be changed via the DISPLAY REFINE option.

The previous DISPLAY TOL option has been removed from UTILITY >> SET TOLERANCE.

CREATE >> SHELL

A visualization of problematic faces is available now if the SHELL operation does not succeed due to the self-intersection of resulting faces.

The loops with self-intersections are displayed in yellow.

SURFACES TO SOLID (Previously SRFSOL)

SRFSOL is now an option in the TRANSL function under the option name SURFACES TO SOLID.

New algorithm

A new algorithm provides more accurate results for the calculation of edges and vertices of Solid Objects from Wireframe surfaces.

Interrupting during SRFSOL.

It is now possible to interrupt the SURFACES TO SOLID process at any stage of the operation by pressing the space bar.

Three interruption options are available:

CONTINUE	Continue the SURFACES TO SOLID operation.
PARTIAL RESULT	Display SURFACES TO SOLID results at the time of the interrupt.
CANCEL	Cancel the SURFACES TO SOLID operation.

Recommended Tolerance in SURFACES TO SOLID

This option is available if the parameter WITH RECOMMEND.TOL. is used.

This new feature can be used to calculate the optimal tolerance for creating more accurate models. Based on the surfaces of the model and the user-defined tolerance in the SURFACES TO SOLID, the system analyses the gaps and small edges, and displays a message regarding the recommended tolerance. The recommended tolerance is calculated only if the parameter WITH RECOMMENDED TOL is set.

Automatic Fairing of Bad Surfaces.

While finding bad surfaces during the SURFACES TO SOLID process, the system now tries to fair them in order to convert them into correct solid faces.

Update boundaries

During the SURFACES TO SOLID operation, the procedure corrects trimmed surfaces boundaries.

MoldExpert

Mold environment

The PART and MOLD environments have been combined into the PART environment. Due to this, the locations of some of the MOLD functions/options have changed.

- SPLIT is now an option of the MODIFY function.
- MOLD2ASM has been re-named EXTR2ASM and is now in the first overlay of the SOLID environment. Besides Mold operations, this function may also be used to create assembly files.
- SETUP has been re-named MOLDPREP and is now in the second overlay of the SOLID environment.
- SEPARATE is now in the second overlay of the SOLID environment.

MOLDPREP >> SHRINKAGE

This option now uses uniform scaling instead of non-uniform scaling. (See SCALE on the following page).

Workpiece

In Version 9.0, when the mold designer wished to use a workpiece with a non-standard shape, the shape must first have been created in a separate file and then imported into the mold file using option FROM FILE.

In Version 10.0, the new option, BY PICK, will enable you to create a workpiece directly within the mold file.

SEPARATE >> PARTING LINE >> EDIT - Parting-line editor

A Parting Line EDIT option has been added. This enables you to carry out the following operations:

Edit separate loops of a complex PARTING LINE

Edit a single loop of a simple PARTING LINE

Remove loops from the PARTING LINE

Extract loops to a new PARTING LINE

In Version 10.0 it is possible to move the PARTING LINE to another level and to blank it.

SEPARATE >> PARTING LINE >> CREATE >> INTERNAL ONLY

A new option has been added to the process of creating a PARTING LINE: INTERNAL LOOPS. This option enables you to create separate collection(s) of internal loops in the same, or in a different, direction as the parting line, and after this to unite these internal loops to external by using the option PARTING LINE/MERGE.

PARTING LINE for Open Objects

When a mold designer uses a body created using the SURFACES TO SOLID function, and this model is not a closed solid body, its direction (vector that indicates the outside direction) may be incorrect. To prevent these problems, you can now check the body and change the direction.

Parting Surface

In Version 9.0, when a parting surface was not completed, and it was stopped on the middle of the edge, this edge was automatically marked as a break edge.
In Version 10.0, these edges are not automatically marked as a break edges and to complete the parting surface you have to first use the new option MODIFY >> BREAK EDGE to prepare edges to be the DRIVE component for surfaces.

Introduction

We recommend that new users of **Cimatron**[it] read at least the first three chapters of the **Fundamentals & General Functions Manual**, to acquire a working knowledge of **Cimatron**[it].

As you work with the system, different function overlays will be displayed. At the beginning of many chapters, you will find an overlay diagram which shows the functions described in the chapter and indicates how to access them. Within chapters, functions are listed in alphabetical order. The names of the functions and their options appear in capital letters. At the top of each page a status line tells you which option or sub-option is described on that page.

Although you may only need one particular option within a function, we recommend that you read the description of the interaction for the entire function. Modal parameters, which determine the mode which will be active when the function is executed, are explained at the end of each function.

After you are familiar with the basic system, scan the manual occasionally to discover functions you are not using and to learn how to take full advantage of the power of **Cimatron**[it]**.**

About This Manual

This manual provides explanations of the Solid Modeling functions and their use.

Chapter 1	**Using the Solid Modeling Module** gives an overview of the functions and explains how to create a new solid object.
Chapter 2	**Solid Modeling Application Functions**, contains explanations of each modeling function contained in the SOLID application menu.
Chapter 3	**General Functions in the PART Environment** describes functions specific to the Solid Modeling Part Environment.
Chapter 4	**Drafting**, explains the use of the Drafting application with a solid object.
Chapter 5	**Assembly**, explains the procedures used to assemble solid components.
Chapter 6	**General Functions in the ASSEMBLY Environment**, describes functions specific to the Solid Modeling Assembly Environment.
Chapter 7	**Mold Functions**, explains each function contained in the Mold Application menu.
Appendix A	**Working With Solid Assemblies,** provides important information for users of the Solid Modeling Assembly Environment.

Typographical Conventions

Throughout this manual, certain conventions have been used to present different types of information.

For each function in the manual the following information is presented (in the order listed below):

Function & Purpose

The name of the function as it appears in the function bank, presented in large bold upper case type, followed by the purpose of the function. For example:

EDIT | The purpose text, presented in regular type, describes what the function does.

Main Options

Presents the first level of options within each function in table format, as they appear on the screen. For example:

Main Options:

PARAMETERS
SKETCH
RELATION
DELETE
SUPPRESS
RENAME
UPDATE
REPLAY
TRIM

The interaction for each main option is described separately within the function. Where the description begins, the main option name is presented, preceded by the function name and graphic arrows. For example:

EDIT >> PARAMETERS

Sub-options

When branching occurs in the interaction of a function, the interaction for each option is described separately. The option names are presented in bold, block letters and the option path is represented by graphic arrows. For example:

CURVE >> SPLINE

How To (optional)

Summarizes the recommended action to be followed in the current function/option, presented in sans serif type. For example:

How To:

1. Choose a reference plane or face.
2. You now enter the **Sketcher.** Sketch and dimension a contour and then exit the **Sketcher.**
3. Define the extent of extrusion of the feature by selecting:
   If you select DELTA, enter the value. If you select FROM - TO, pick the relevant faces planes or surfaces (if nothing is indicated for FROM, the default is the active face/plane.
4. The object will be created.

Interaction

The prompts which tell you how to execute the function are listed in upper case sans serif italic letters.

Prompt explanations are shown to the right of the prompts. (Some additional notes are also provided in the right column.) For example:

<CR> TO CONTINUE Execute the function.

Modal Parameters

The modal parameters for each function appear at the end of the function under the heading <function name>: Modal Parameter Definitions. For example:

EDIT: Modal Parameter Definitions

Within this section, the modal parameters are listed in alphabetic order. Modal parameters are represented in the text by a filled box as shown below. Modal parameters marked with an asterisk are system generated and cannot be changed.

Parameter explanations are to the right of the parameters.

For example:

■ SCALE Scale the section.

Notes

Provides information to help you avoid problems and achieve accurate results. Each note is preceded by a bullet character for immediate identification. For example:

Notes: • The default active work plane is the XY plane of the active coordinate system.

System Messages

System messages appear in upper case sans serif type and are preceded by an explanation stating that they are system messages. For example:

When you have supplied sufficient dimensions to fully define the sketch, the message FULLY DIMENSIONED will appear.

End of Function

The end of a function or section is marked by a box character ❑ .

Chapter 1
Using the Solid Modeling Module

The Solid Modeling Module Functions

Cimatron provides the following modeling modules: WIREFRM (**Wire Frame**), for wire frame modeling, and SOLID (**Solid Modeling Module**) for solid modeling.

MODELING

WIREFRM
SOLID

The Solid functions are contained in an overlay in the Solid Modeling Module application menu. These functions are described in Chapter 2. A number of additional functions are located in the General Function and User Function menus, and in the Drafting application menu. These functions will be addressed in later chapters.

The Solid Application Menu

The Solid Modeling Module application menu consists of the following functions:

EDIT
CREATE
DETAIL
SURFACE
MODIFY
DATUM
COPY
GROUP
TRANSL
UTILITY
EXTR2ASM

EDIT	Edit existing solid objects.
CREATE	Construct solid objects.
DETAIL	Add detail features to solid objects
SURFACE	Create parametric surfaces.
MODIFY	Modify parametric surfaces, reference curves or objects.
DATUM	Create reference entities
COPY	Copy and position a feature or a set of connected features.
GROUP	Create, place, and explode sets of parametric features.
TRANSL	Translate entities between wireframe and solid.
UTILITY	A collection of SOLID utilities.
EXTR2ASM	Produce and manage an assembly while in a multi-object part environment.

Creating a New Object

The CREATE options (EXTRUDE, REVOLVE, and DRIVE) are used to create a new solid object (See Chapter 2, Solid Modeling Functions). This section explains how to use these functions to create a new object. These explanations assume that there is no solid object in the file when you begin. The first solid object will be created on the default plane if it exists. If a previous object exists, a prompt will ask you to pick a reference plane or face upon which the new feature will be created.

EXTRUDE

Create a solid object by extruding a 2D shape.

How To:

1. Choose a reference plane or face.
2. You now enter the **Sketcher**. Sketch and dimension a contour and then exit the **Sketcher**.
3. Define the extent of extrusion of the feature by selecting:
   If you select DELTA, enter the value. If you select FROM - TO, pick the relevant faces planes or surfaces (if nothing is indicated for FROM, the default is the active face/plane.
4. The object will be created.

Notes:

- The THROUGH option can be chosen only when in the REMOVE mode.
- The contour must be closed only when creating the first feature.

REVOLVE

Create a solid object by revolving a 2D contour about an axis.

How To:

1. If needed, create a reference axis or edge using DATUM >> AXIS.
2. Select the option REVOLVE.
3. <PICK> the reference axis you have created or a solid's edge.
4. You are now in the **Sketcher**.

   Sketch and dimension a contour, then EXIT the **Sketcher**.
5. Define the extent of the revolution by selecting FULL (360 DEG) or DELTA. If you select DELTA, enter the value of revolution in degrees.
6. The object will be created.

Defining Open Contours on Faces in EXTRUDE and REVOLVE

When using EXTRUDE or REVOLVE to generate a new feature on an existing object, you have the option of sketching an open contour on a face. The open contour, together with the boundaries of the face, must create a closed contour, as shown in examples a, b, and c in Figure 1-1.

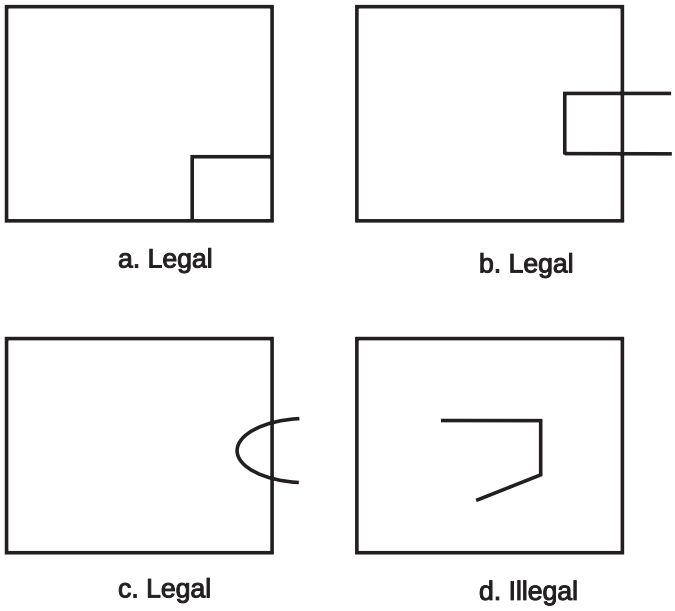


Figure 1-1: Examples of Legal and Illegal Open Contours

The contour in example d is illegal, since it does not intersect the boundaries of the face.

A legal open contour may also extend past the edges of the face, as shown in examples b and c.

When removing material, an open contour that extends past the edges of the face, along a straight line, is considered to be infinite (e.g., example b). When an open contour extends past the edge

along curved lines (e.g., example c), only the part of the contour that lies within the boundaries of the face will be considered.

Picking Faces

When you work on an object that already exists, a function may prompt you to indicate a face upon which to carry out an operation. Since two adjoining faces have a common edge, this task sometimes requires an additional step to specify the desired face.

How To:

1. <PICK> an edge of the desired face. The system will highlight one of the two faces that share the edge.
2. If the highlighted face is the one you want, release the <PICK> button. If not, move the cursor until the desired face is highlighted, and then release the button.
3. To pick a different edge: while pressing <PICK>, move the cursor away from the currently picked edge. When the corresponding face is no longer highlighted, release the <PICK> button and repeat the process from Step I.

Editing Features

When editing, you will be prompted to indicate features upon which to carry out editing operations. There are two techniques to pick objects:

How To:

1. <PICK> by directly pointing at the desired feature.
2. Activate the submenu. When activated, a list of all features appears. Clicking on the wanted feature will pick it. After finishing the edit of the feature, the list reappears and you may pick another feature or exit. Only features that the box to the left of their name is highlighted, can be picked.

Assembling Solid Objects

Assembly operations can be performed on solid objects within the ASSEMBLY application, described in Chapter 5, **Assembly**. This application is accessed by adding a flag when invoking **Cimatron (i.e.** comates -apenv).

Sketcher

Sketcher - General

The **Sketcher**, an intelligent tool for creating 2D contours, is invoked automatically by the following options of the CREATE function: EXTRUDE, REVOLVE, DRIVE, HOLE, RIB, SHAFT HOLE, and DATUM >> REF_CURVE or when creating DETAIL >> CLIP in the Drafting environment. The **Sketcher** lets you sketch a contour which is then used to generate a new solid object or a protrusion or cut-out in an existing solid object.

The **Sketcher** helps you sketch by:

- providing you with all the functions available in **Cimatron** for the creation, modification and dimensioning of lines, circles and arcs. These functions are described in the section The Sketcher Functions on page 1-6.
- producing intelligent feedback lines (construction lines) that reflect a relationship between the feature you are sketching and another entity that has already been sketched. When you start to draw a line and drag the cursor across the screen, the **Sketcher** displays a dashed line to indicate the lines that are parallel, perpendicular or tangential to previously drawn features.
- displaying feedback lines for three types of implicit points: intersection, end, and center. You can select which types of implicit points are active. Special symbols appear for each type of point (see the **Sketcher** function DISPLAY >> THROUGH POINTS, page 1-8).
- snapping the current feature to align it with the end or center point of other features, or according to a geometric condition. When a feedback line appears on the screen, and you release the <PICK> button on or close to the target, the resulting feature will align itself accordingly.

You leave the **Sketcher** by selecting EXIT from the **Sketcher** main menu.

Using the Sketcher

Certain functions automatically invoke the **Sketcher**. The first time you enter the **Sketcher**, you create a 2D contour on the plane of the screen. When modifying an existing object, you are prompted to select the plane or face on which to sketch the contour.

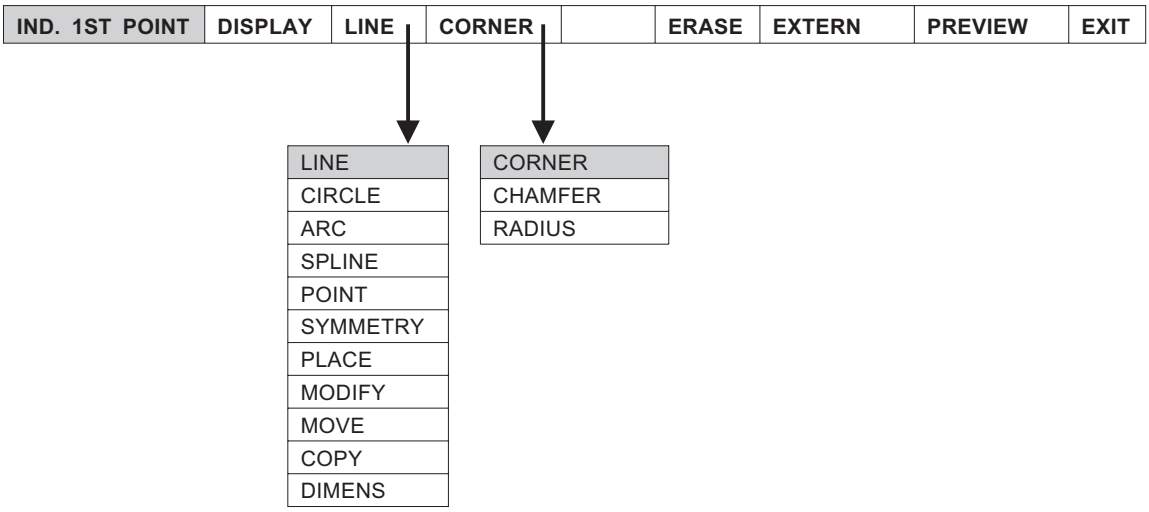
How To:

1. Sketch the contour using lines, arcs and circles. Press the <EXIT> button on the mouse when you finish creating each type of contour.
2. Set dimensions, or leave the **Sketcher** without dimensioning and supply them at a later stage.

   When you <PICK> an entity to dimension, the application intuitively understands which dimension to display: length, radius, angle, etc.
3. To leave the **Sketcher**, select EXIT. The sketch is automatically regenerated with the defined dimensions, and you are returned to the active function.

The Sketcher Functions

When entering the **Sketcher**, the following menu appears:



The main **Sketcher** options are:

IND. 1ST POINT — By default, the LINE option is activated and you are prompted to indicate the first point of a line. Either <PICK> a point on the screen, or select the LINE modal and choose another type of entity. Before picking the first point of an entity, you can also press <SUBMENU> and activate a constraint. Select criteria for displaying feedback lines and points.

DRAW — Once you have picked the first point, the system goes into "drag" mode, and this prompt appears. Proceed to sketch the contour. When picking additional points, you can either use the system defaults, search dynamically using the feedback lines, or activate a constraint from the submenu. Applying a constraint reduces the number of additional constraints that may be applied to subsequent points.

DISPLAY — Select criteria for displaying feedback lines and points, and save sketch.

LINE (Creation) — Sketch a selected type of entity, enter dimensions or modify the sketch.

CORNER — Select the type of corner that will be created when drawing lines. This option only appears when LINE is selected.

ERASE — Delete the selected line/circle/arc/spline up to the first intersection.

EXTERN Pick an external geometric entity according to the following choices:

Add Ref	Use Geom	UnRef

- Add Ref — Create additional reference geometry by projecting existing edges onto the sketching plane.
- Use Geom — Use existing geometry in the solid for creating new sketches.
- UnRef — Disconnect an entity from its constraints. This option is used to disconnect entities that were created using USE GEOM or ADD REF.

PREVIEW Preview the sketch as it will be constructed after exiting the **Sketcher.**

EXIT Exit the **Sketcher** and continue with SOLID functions.

Notes:

- Avoid creating complicated contours in the **Sketcher**. It is difficult to dimension them properly.
- When dimensioning a contour in **Sketcher** it is better to either provide all the dimensions or none at all. Avoid giving partial dimensions.
- Pay attention to construction lines and points. Do not make unnecessary connections.
- If, after changing parameters, the PREVIEW option is selected, and then <REJECT> is pressed, the following message is displayed:

  EXIT SKETCH? YES NO

  <PICK> YES to exit the **Sketcher** or NO to remain in the **Sketcher**.
- If the ERASE option, ALL suboption, is selected, the following message is displayed:

  CANCEL SKETCH? YES NO

  <PICK> YES to cancel the sketch or NO to leave the sketch as is.

Sketcher >> DISPLAY

Display feedback lines according to selected criteria.

SELECT	THROUGH POINTS	CONSTRUCTION MODES	GRID ANGLES	SAVE TO FILE

DISPLAY >> THROUGH POINTS

SELECT	INTERS	END	CENTER	PT

Activate feedback lines are shown as dashed, and symbols appear for three types of implicit points: intersection, end, and center. By default, all three types are activated. The following symbols appear for each type:

INTERS (intersection)	
END and PT	
CENTER	
MID	

The PT indicates the end and center points of the outer boundary.

Notes:

- If the sketch is drawn on an existing face, all the end and center points of this face will be displayed as **+**.
- When the cursor is located on these points, they will appear as **X**.

DISPLAY >> CONSTRUCTION MODES

SELECT	NORMAL	THRU PT	PAR/PERP	ON LINE	EXTERNAL

Note: • Horizontal and vertical feedback lines emanating from the cursor are always displayed.

The following descriptions apply when creating a line:

NORMAL Display a feedback line which extends from the location of the cursor to an active implicit point. The feedback line is normal to the line that you are drawing.

THRU PT Display a feedback line in the current line direction to an active implicit point.

PAR/PERP Display a feedback line when the line you are drawing is parallel or perpendicular to an existing line. The feedback line is superimposed on the existing line.

ON LINE Display a feedback line when the cursor is located on an existing line (or on an infinite extension of the line).

EXTERNAL Relate or remove relation to the external boundary curve.

Note: • By default, the options PAR/PERP and ON LINE are activated.

DISPLAY >> GRID ANGLES

ENTER GRID ANGLE	ANGLE 1 = 45.000	ANGLE 2 = 60.000	ANGLE 3 = 0.000	ANGLE 4 = 0.000	ANGLE 5 = 0.000
	ANGLE 6 = 0.000	ANGLE 7 = 0.000	ANGLE 8 = 0.000	ANGLE 9 = 0.000	ANGLE 10 = 0.000

Display feedback lines for up to ten predefined angles, in addition to horizontal and vertical feedback lines.

As a default, all angles are set to 0.

Note: • Grid angle settings are only active for one session of **Cimatron**. In order to activate the feedback lines, the user must choose the grid angles for every new sketch.

DISPLAY >> SAVE TO FILE

Save the current sketch to a file.

ENTER FILE NAME The system prompts for a file name:

The sketch will be saved in the current directory as **<filename.skf>** or full path name.

The sketch can be inserted into the **Sketcher** with the PLACE option.

Sketcher >> LINE

Create lines by selecting points.

Note: • The default corner option is a sharp corner. To create other types of corners, see CORNER, page 1-17.

IND. 1ST POINT — By default. <PICK> the starting point of the line.

Constraints can be imposed by using the submenu. These constraints are only active for one operation.

POINT
PARALLEL
NORMAL
TANGENT

POINT — The default. The point you <PICK> is the starting point of the line.

PARALLEL — The line is constrained to be parallel to an existing line.

PICK PARALLEL — <PICK> an existing line to which the new line will be parallel.

IND. 1ST POINT — <PICK> the starting point of the new line.

NORMAL — The line is constrained to be normal to an existing curve.

PICK CURVE — <PICK> the curve to which the new line will be normal.

IND. 1ST POINT — <PICK> the starting point of the new line.

TANGENT — The line is constrained to be tangent to an existing curve.

PICK 1ST TANGENT — <PICK> a curve to which the new line will be tangent.

DRAW — <PICK> the ending point of the line. You may also use the submenu to activate additional constraints.

Only those constraints that are applicable will appear in the submenu.

Sketcher >> CIRCLE

Create a circle.

IND. CENTER

By default, <PICK> a point to set the center of the circle, then drag the mouse to size the circle.

Constraints can be imposed by using the submenu.

CENTER
POINT
TANGENT
DIAMETER

CENTER

The default. The point you <PICK> is the center of the circle.

POINT

The point you <PICK> is a point on the circumference of the circle. Since this is the mode used to create arcs, the following prompt appears:

IND. 1ST CIRCLE POINT

<PICK> a point on the circumference of the circle.

If you activate this constraint again, you will be prompted to <PICK> additional arc points, up to a total of three.

TANGENT

Constrain the circle to be tangent to an existing entity.

PICK 1ST TANGENT

Select an entity to which the circle will be tangent.

DIAMETER

Constrain the diameter of the circle to that of an existing circle.

PICK CIRCLE/ARC

Select the circle or arc whose diameter will be copied.

DRAW

Move the cursor to size the circle.

The submenu may allow you to impose additional constraints, depending on the constraints applied previously.

Only those constraints that are applicable will appear in the submenu.

Sketcher >> ARC

Create arcs.

IND. 1ST ARC POINT — The default method is to <PICK> two endpoints on either side of the arc and then size the arc by dragging the mouse.

Constraints can be imposed by using the submenu.

CENTER
POINT
TANGENT
DIAMETER

CENTER — The point you <PICK> will be the center of the arc.

POINT — The default. The point you <PICK> will be a point on the arc.

TANGENT — Constrain the arc to be tangent to an existing entity.

PICK 1ST TANGENT — Select the entity to which the arc will be tangent. You can <PICK> more than one entity.

DIAMETER — Copy the diameter of an existing arc or circle.

PICK CIRCLE — Select an arc or circle to copy.

DRAW — Drag the mouse to size the arc.

The submenu allows you to impose additional constraints, depending on the constraints applied previously.

Only those constraints that are applicable will appear in the submenu.

Note:

- If a definition is ambiguous, a full circle will be created.

Sketcher >> SPLINE

The spline is defined by selecting points that it will pass through and defining the slope at each point.

The slope will be defined automatically in the following case:

The spline starts after a line or arc without pressing <EXIT>. The slope will be taken from the line/arc.

If no slope option has been chosen, the slope will be defined after the next point is selected.

To enter SLOPE, press <SUBMENU>.

POINT
SLOPE
SLOPE BY PICK
TANGENT

POINT The default

SLOPE Dynamically select the slope at a point. A yellow rubber-like line will be displayed. Rotate the yellow rubber-like line and press <PICK> to define the slope, or use <SUBMENU>. The following menu appears:

FREE
NORMAL
PARALLEL

FREE Default

NORMAL The slope will be normal to the line picked.

PARALLEL The slope will be parallel to the line picked.

SLOPE BY PICK Pick the line that will define the slope.

TANGENT Select the line/arc to which to define the tangent. This option is available only to define the slope at implicit points.

Note:

- The selection TANGENT appears only when the sketch is initiated with ARC.

Sketcher >> POINT

Create points of reference. These may be used, for example, to mark the center points of holes.

When creating points in the **Sketcher**, the points must be placed on existing geometry such as lines curves and defined constraints etc. The exception to this rule is the location point for a hole, which can be placed anywhere on the plane.

Sketcher >> SYMMETRY

The SYMMETRY option appears only at the beginning of the sketch. If the sketch starts with another option (such as LINE), the symmetry option disappears.

One or two symmetry lines can be defined. Where two lines are defined, the second will start from the end of the first.

The sketching will be done only on one side of the symmetry line, and will be clipped from the other side.

In the case of a symmetry line, another option will be added to the **Sketcher** menu.

VIEWING where the result after mirroring according to the symmetry line can be displayed.

Note:

- Dimensions that refer to the symmetry line are considered to be symmetry dimensions.
- Using a symmetry line is recommended for symmetric sketches.

Sketcher >> PLACE

Adds an existing sketch to the current sketching session.

SELECT SKETCH	BY PICK	FROM FILE	LAST

PLACE >> BY PICK

Pick a feature created by the **Sketcher**. All existing external dependencies will be discarded.

If the feature consists of more than one sketch (when creating DRIVE, SHAPED HOLES and SHAPED SHAFTS features), the following prompt will appear:

SELECT FEATURE Pick a sketch.

PLACE SKETCH Locate the sketch. The sketch can now be moved and/or rotated according to the rules described below.

PLACE >> FROM FILE

Pick the sketch from an existing file that was created using the SAVE SKETCH utility.

ENTER FILE NAME Enter the name of the sketch file that was saved.

PLACE SKETCH Locate the sketch. The sketch can now be moved and/or rotated according to the rules described below.

PLACE >> LAST

Load the last sketch that has been saved.

PLACE SKETCH Locate the sketch. The sketch can now be moved and/or rotated according to the rules described below.

Moving the Sketch

The mode of sketch movement (MOVE or ROTATE) is highlighted in the upper right corner of the screen. The contour appears on the reference plane. Pick a point on the sketch and drag it to its new position. The mode then changes to ROTATE. The center of rotation is the last point used to move the sketch. Pick a point on the sketch to define the angle of rotation. Rotate the body. After every pick, the mode of movement switches automatically. To change the mode of operation manually, pick the desired option from the submenu.

MOVE
ROTATE
MIRROR

By selecting Mirror from the submenu, the sketch will be mirrored along the X axis.

When picking a point of reference on the sketch for rotating or moving the sketch, it will be highlighted with one of the following:

 indicates the place mode.

 indicates the rotate mode.

Note:

- The placed sketch may include internal dependencies (constraints and dimensions).
- The imported sketch will be fully integrated into the current sketch and can be edited with all the relevant tools.
- When a line in the sketch falls under a constraint, it is highlighted in green.
- When the placed sketch is rotated, the **Sketcher** will examine constraints related only to the lines exiting the point being dragged.

Sketcher >> MODIFY

Modify points or curves of a sketch.

PICK PT/LINE/CIRC	DISPLAY	MODIFY			ERASE	EXTERNAL		EXIT

PICK PT/LINE/CIRC/DIM <PICK> a point or a curve (line, circle, arc) of the sketch and drag it to a new position. The sketch will change according to the new location of the moved entity.

When choosing a circle or arc, the following submenu appears:

RADIUS
CENTER

RADIUS Change the radius of the circle or arc.

CENTER Change the location of the center of the circle or arc.

Notes:

- Splines cannot be edited by this method.
- All constraints that are not affected after the modification will be kept.
- Some dimensions may be automatically deleted to allow the new constraint set.

Sketcher >> MOVE

Move the entire sketch or part of a sketch to a new location, and/or change the orientation of the sketch.

SELECT	ALL	PART

If ALL was picked:

PLACE SKETCH	DISPLAY	MOVE			ERASE	EXTERNAL	PREVIEW	EXIT

PLACE SKETCH <PICK> a point on the sketch and place the sketch in a new position. The mode then changes to Rotate.

To rotate the sketch, <PICK> another point on the sketch. This point will be rotated about the original point used to place the sketch.

When picking a point on the sketch, the point will be highlighted with one of the following symbols:

◇ Indicates the place (move) mode.

⬡ Indicates the rotate mode.

After every <PICK>, the mode of movement switched automatically. To change the mode of movement manually, select the appropriate option from the submenu.

MOVE
ROTATE
MIRROR

By selecting Mirror from the submenu, the sketch will be mirrored along the X axis.

If PART was picked:

PICK ENTITIES & EXIT Pick the entities to be moved and/or rotated, <EXIT> when finished.

Proceed as if ALL was picked.

Sketcher >> COPY

Copy the sketch or part of the sketch.

Follow the same procedure outlined above for the option MOVE. The original sketch will remain in its location; only the copy can be moved and rotated.

Sketcher >> DIMENS

Dimension entities.

PICK PT/LINE/CIRC	DISPLAY	DIMENS		SHOW	ERASE	EXTERN	PREVIEW	EXIT

PICK PT/LINE/CIRC/DIM <PICK> an entity to be dimensioned.

IND. POSITION/EXIT The dimensions, as drawn, will appear. To change the value, select the modal (or press the <TAB> key) and type the new dimension. Drag the dimension to define its location and press <EXIT>.

SHOW This option highlights the points/curves that have not as yet been constrained. This is relevant only when DIMENS is selected.

Notes:

- When you have supplied sufficient dimensions to fully define the sketch, the message FULLY DIMENSIONED will appear.
- If you receive the message SKETCH IS NOW OVER-CONSTRAINED, erase the incorrect dimensions or entities and correct the sketch.
- When dimensioning a spline, all the X and Y coordinates of the points must be supplied. If the slope has been defined, an angular dimension should be defined on that point.
- To change a dimension value in the **Sketcher**, pick it and a parameter modal box will appear.

Drawing Lines

When drawing lines, select the type of corner: sharp, chamfered or rounded. This option is only relevant when LINE is selected.

CORNER
CHAMFER
RADIUS

Sketcher >> CORNER

Create a sharp corner. This is the default.

Sketcher >> CHAMFER

Create a chamfered corner. Once two lines have been drawn, the following prompt appears:

IND. CHAMFER SIZE Indicate the chamfer size. Once a chamfer size has been defined, the modal toggles to SAME AS. Subsequent corners will produce chamfers the same size as that defined previously. Change the size by toggling back to the IND.CHAMFER SIZE prompt.

Sketcher >> RADIUS

Create a round corner. The interaction is similar to that of the CHAMFER option.

Sketcher >> ERASE

Delete the selected line/circle/arc/spline up to the first intersection, delete a single entity or all entities.

PICK TO ERASE <PICK> an entity to be deleted.

Click <SUBMENU> to obtain the following options:

SINGLE
ALL

ERASE >> SINGLE

Delete a single entity.

ERASE >> ALL

Delete all displayed entities. The following prompt appears:

CANCEL SKETCH?	YES	NO

Click YES to delete or NO to cancel the deletion.

Note:

- When deleting a spline, the spline point is not always deleted. To ensure that it is deleted, use the ERASE >> ALL option.

Sketcher >> EXTERN

Creates additional sketch entities or reference entities.

Add Ref	Use Geom	UnRef

EXTERN >> Add Ref

Create additional reference geometry by projecting existing edges onto the sketching plane.

PICK REF. EDGES <PICK> the edges to be projected onto the sketching plane and <EXIT>. The projected edges will appear in red on the sketching plane.

Note:
- The outer boundary of the sketch is selected by default as reference geometry.

EXTERN >> Use Geom

Use existing geometry in the solid for creating new sketches.

PICK EDGE/EXIT Pick the curve that will be projected onto the sketching plane.

OFFSET Determine the offset of the projected curve relative to the original curve.

FLIP SIDE Toggle between sides to offset.

The default is edge. If the submenu is pressed, the following options are offered:

SINGLE
LOOP

SINGLE Choose a single curve.

LOOP Choose a closed loop of curves.

Note:
- End points of a curve that has been offset should be dimensioned.

EXTERN >> UnRef

Disconnect an entity from its constraints. This option is used to disconnect entities that were created using USE GEOM or ADD REF.

PICK REFERENCE <PICK> the entity that you wish to disconnect from its constraints. The reference lines and points from which the entity was created disappear.

Note:
- You cannot apply UnRef to splines.

Sketcher >> PREVIEW

Solve the sketch constraints without exiting the **Sketcher**. This option is useful for viewing how the sketch will look when the input dimensions are imposed on the sketch.

Sketcher >> EXIT

Exit the **Sketcher** and continue with SOLID functions or DETAIL >> CLIP in the Drafting environment.

Sketching on a Plane

Because a plane is infinite, geometric references must be imposed in order to constrain the sketch.

SELECT	PICK REF. EDGES	IND. +X AXIS

PICK REF.EDGES Define reference curves by projecting edges onto the sketch plane.

PICK REF. EDGES <PICK> edges.

IND.+X AXIS Define a reference X axis by projecting an origin point and a point in the +X direction onto the sketch plane.

IND. ORIGIN Indicate an origin point.

IND. +X AXIS Indicate a point in the +X direction.

Once the reference edges or the +X axis has been defined for the sketch, the **Sketcher** can be invoked by <EXIT>.

Reference edges will be projected onto the sketch plane and will produce reference feedback lines as if they were real edges on this plane.

The X axis will be displayed on the sketch plane as a cross. The reference feedback lines will be parallel or perpendicular to this axis.❑

Chapter 2
Solid Modeling Functions

Introduction

In this chapter, the Solid Modeling functions are presented. The functions are presented in one overlay:

EDIT
CREATE
DETAIL
SURFACE
MODIFY
DATUM
COPY
GROUP
TRANSL
UTILITY
EXTR2ASM

COPY

Copy or move a feature or a set of connected features.

Main Options:

COPY FEATURE
COPY OBJECT
MOVE OBJECT

COPY FEATURE Copy a feature.

COPY OBJECT Copy an object.

MOVE OBJECT Move objects.

COPY >> COPY FEATURE
COPY >> COPY OBJECT

Copy a feature or object.

The following options are displayed:

ARRAY
ROTATE
RELOCATE
REFERENCE
MIRROR

ARRAY Make a grid of points on which the feature/object will be copied.

ROTATE Copy a feature/object around an axis.

RELOCATE Copy a feature/object by indicating a position on a new plane relative to the former plane.

REFERENCE Copy a specific feature according to an existing copied feature.

MIRROR Generate a mirror image of a feature/object relative to a plane/face.

Picking points can be assisted by invoking the sub menu:

END
MID
CENTER
KEY IN
INTERS
PIERCE
PICK

For explanations of these picking choices, see page 2-53.

Note:

- Both copying functions that position the object/feature in an array of points (ARRAY and ROTATE), have a LOCATION option. Picking this option allows you to move the matrix of points so that the relative position between the feature/object and the matrix changes. The matrix will relocate so the picked point moves to the feature.
- Once the copy grid points have been drawn, individual points can be deleted from the grid by picking them. The feature or object will not be copied to these deleted points.

COPY FEATURE >> ARRAY
COPY OBJECT >> ARRAY

Copy a feature/object repeatedly to points of a 2D grid.

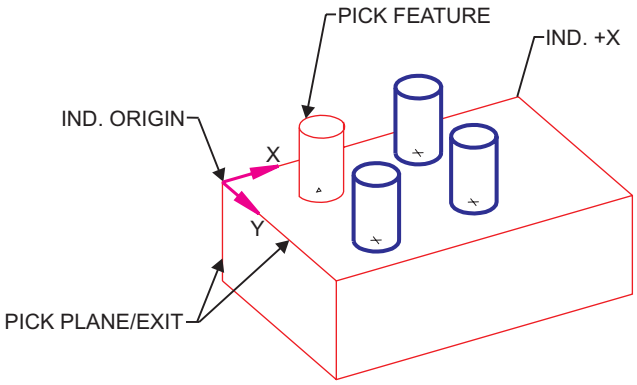


Figure 2-1: COPY >> ARRAY

PICK FEATURE(OBJECT) <PICK> an object or an object feature to be copied.

PICK PLANE / EXIT Pick the plane onto which the feature/object will be copied. If you press <EXIT>, the plane on which that the feature/object was defined will be selected.

INDICATE ORIGIN Indicate the origin on the copy plane.

INDICATE +X AXIS Indicate the direction of the positive X axis.

IND. BASE POINT / EXIT Indicate the point from which the grid will be drawn.

SELECT & EXIT	NX=	NY=	DX=	DY=	FILL/BOUNDARY
	LOCATION				

COPY FEATURE >> ROTATE
COPY OBJECT >> ROTATE

Copy a feature/object repeatedly around an axis.

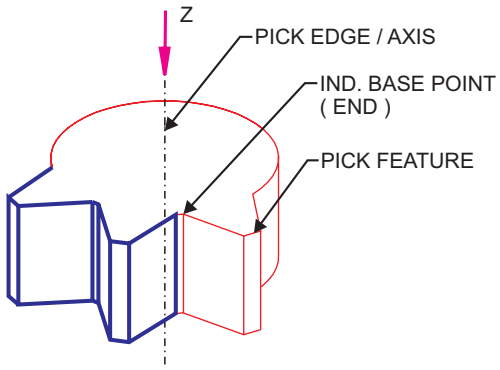


Figure 2-2: COPY >> ROTATE (COUNT = 3, ANGLE = 30)

PICK FEATURE(OBJECT) <PICK> an object or an object feature to be copied.

PICK EDGE / AXIS Pick an edge or an axis around which the feature/object will be copied.

IND. BASE POINT / EXIT Indicate the first point of the series onto which the feature/object will be copied.

SELECT & EXIT	COUNT=	ANGLE=	DZ=	LOCATION

COPY FEATURE >> RELOCATE
COPY OBJECT >> RELOCATE

Copy a feature by indicating a new position on a new plane relative to the original position.

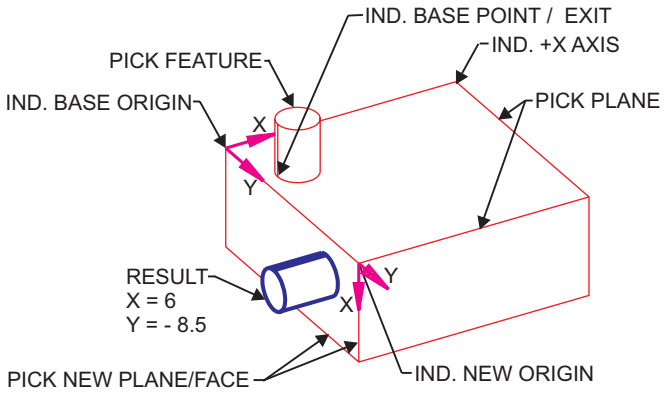


Figure 2-3: COPY >> RELOCATE

PICK FEATURE(OBJECT) <PICK> an object or an object feature to be copied.

PICK PLANE / EXIT <PICK> the plane that will serve as the reference plane for the feature/object. The default is the plane from where the sketch originated.

IND. BASE ORIGIN Select the base origin.

IND. BASE +X AXIS Select a point for the X direction of the axis.

IND. BASE POINT/EXIT Select the base point or exit.

PICK NEW PLANE/FACE <PICK> the face or plane on which the copy will be located.

IND. NEW ORIGIN Select the origin on the new plane.

IND. NEW +X AXIS Select a point for the X direction of the new axis.

<CR> TO CONTINUE	X=	Y=

- X= The X coordinate of the base point on the new plane.
- Y= The Y coordinate of the base point on the new plane.

COPY FEATURE >> REFERENCE
COPY OBJECT >> REFERENCE

Copy a specific feature according to an existing copied feature.

PICK REFERENCE FEATURE — Pick the feature/object relative to which the copied one will be positioned.

COPY FEATURE >> MIRROR
COPY OBJECT >> MIRROR

Generate a mirror image of the feature/object relative to a plane/face.

PICK MIRROR PLANE — Pick the plane around which the mirror image will be generated.

COPY >> MOVE OBJECT

Move objects.

The following options are displayed:

DELTA
ROTATE
RELOCATE
MIRROR

DELTA — Move an object(s) a specified delta distance.

ROTATE — Move an object(s) a specified angle around an axis.

RELOCATE — Move an object(s) by indicating a new origin.

MIRROR — Move an object(s) as a mirror image relative to a plane/face.

MOVE OBJECT >> DELTA

Move an object(s) a specified delta distance.

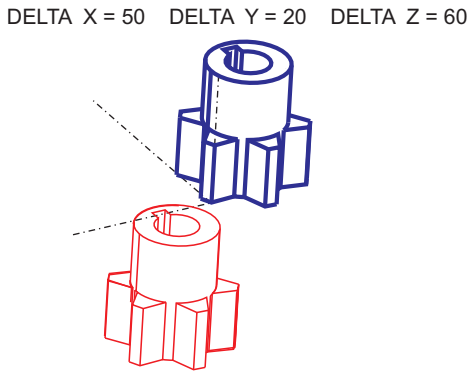


Figure 2-4: COPY >> MOVE OBJECT >> DELTA

PICK OBJECTS & EXIT Pick the objects to be moved and press <EXIT>.

SELECT	TWO POINTS	KEY IN

TWO POINTS

IND. BASE POINT Indicate a reference point on the object.

IND. TARGET POINT Indicate a point to which you want the object to be moved, such that the reference point on the object is repositioned to the target point.

KEY IN

<CR> TO CONTINUE	DELTA X = 50.000	DELTA Y = 20.000	DELTA Z =60.000

Key in the delta X, Y and Z co-ordinates and press <CR>.

MOVE OBJECT >> ROTATE

Move an object(s) a specified angle around an axis.

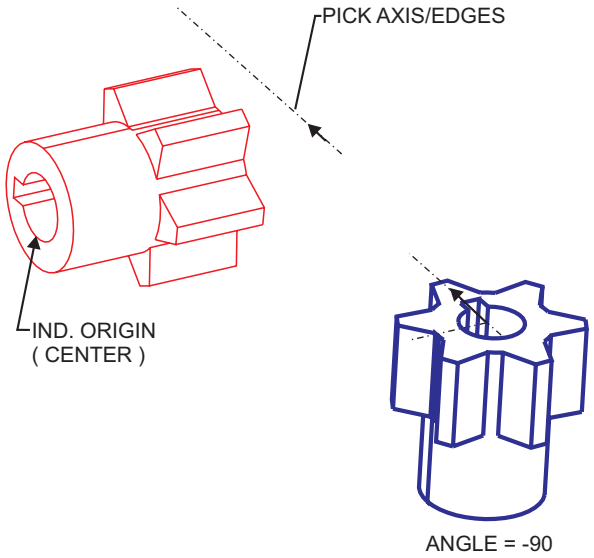


Figure 2-5: COPY >> MOVE OBJECT >> ROTATE

How To:

1. Pick the objects to rotate and press <EXIT>.

   There are two ways to define an axis; steps 2 and 3.

2. Use the direction of an existing edge or axis as a reference direction.

   a). Choose an axis or straight edge which will be used to define the direction. Use the AXIS/EDGE parameter.

   b). Reverse the indicated direction, if desired. Use the FLIP parameter.

   c). Pick APPLY to accept the defined direction.

3. Use the direction of a vector defined by two points: an origin and a point indicating the positive direction.

   a). Indicate an origin point. Use the ORIGIN parameter.

   b). Pick a point to indicate the positive direction of the vector. Use the + DIRECTION parameter.

   c). Reverse the indicated direction, if desired. Use the FLIP parameter.

   d). Pick APPLY to accept the defined direction.

4. Specify the angle that you want the object to be rotated and press <CR>.

Interaction:

PICK OBJECTS & EXIT Pick the objects to be rotated and press <EXIT>.

DEFINE AXIS	AXIS/EDGE	FLIP	ORIGIN	+ DIRECTION	APPLY

DEFINE AXIS Define an axis. See How To, above.

<CR> TO CONTINUE	ANGLE = 0.000

Specify the angle that you want the object to be rotated and press <CR>.

MOVE OBJECT >> RELOCATE

Move an object(s) by indicating a new origin.

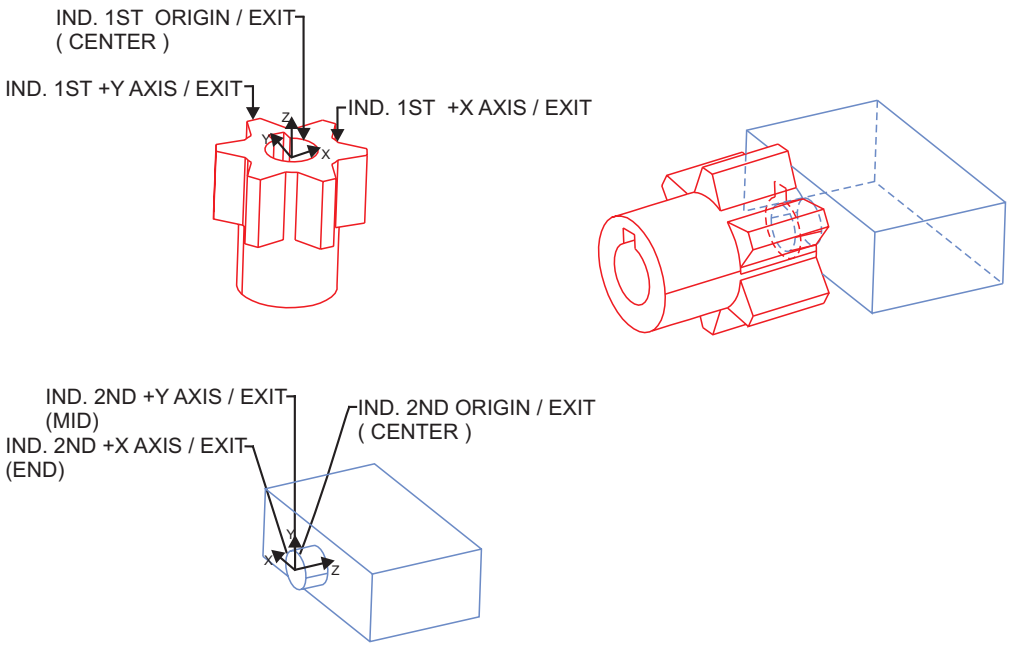


Figure 2-6: COPY >> MOVE OBJECT >> RELOCATE

PICK OBJECTS & EXIT Pick the objects to be relocated and press <EXIT>.

IND. 1ST ORIGIN/EXIT	FLIP

IND 1ST ORIGIN / EXIT
IND. 1ST +X AXIS/EXIT
IND. 1ST +Y AXIS/EXIT
IND 2ND ORIGIN / EXIT
IND. 2ND +X AXIS/EXIT
IND. 2ND +Y AXIS/EXIT

Indicate the first and second origin selection.

It is possible to flip the direction as necessary.

POINTS OK? Y/N

YES Accept the points.

NO Re-indicate the origin.

Indicate another origin if necessary and press <EXIT>.

MOVE OBJECT >> MIRROR

Move an object(s) as a mirror image relative to a plane/face.

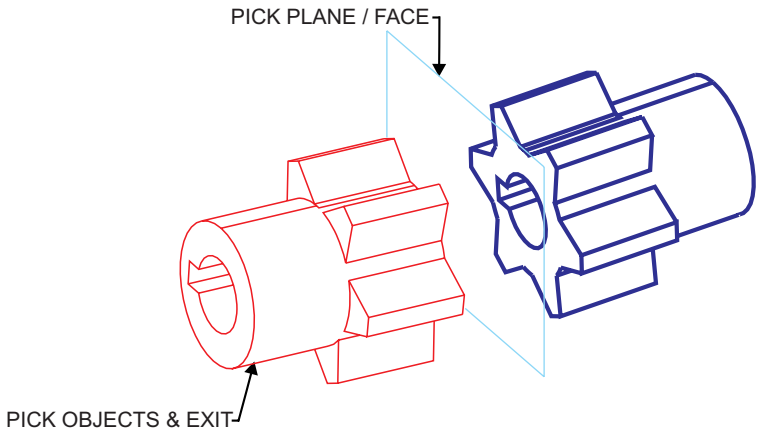


Figure 2-7: COPY >> MOVE OBJECT >> MIRROR

PICK OBJECTS & EXIT Pick the objects to be mirrored and press <EXIT>.

PICK PLANE/FACE Pick the plane or face around which the mirror image will be generated.

PLANE OK? Y/N

YES Accept the plane.

NO Re-indicate the plane.

COPY: Modal Parameter Definitions

- + DIRECTION — Pick a point to indicate the positive direction of the vector.
- ANGLE — The angle of rotation/revolution.
- APPLY — Use the direction you indicated above as an axis definition.
- AXIS/EDGE — Choose an axis or straight edge which will be used to define the axis.
- BOUNDARY — Copy the feature/object only to the points defining the boundary of the grid.
- COUNT — The number of features/objects to be made.
- DX — The spacing of grid points in the X direction.
- DY — The spacing of grid points in the Y direction.
- DZ — The normal space between the plane of rotation and the feature/object.
- DELTA X, DELTA Y, DELTA Z — Key in the delta X, Y and Z co-ordinates.
- FILL — Copy the feature/object to each point in the grid.
- FLIP — Reverse the indicated direction, if desired.
- LOCATION — Relocate the array/rotate grid relative to the feature.
- NX — The number of grid points in the X direction.
- NY — The number of grid points in the Y direction.
- ORIGIN — Indicate an origin point.

COPY: Usage Envelope

1. In COPY >> COPY FEATURE, Cimatron always tries to perform a BOOLEAN ADD operation between the copied objects. If this cannot be done, separate solid objects will be created. No Boolean operations will be performed on other existing objects.

❑

CREATE

Create basic solid features.

Main Options:

SELECT OPTION	EXTRUDE
	REVOLVE
	DRIVE
	HOLE
	SHAFT
	SHELL
	RIB
	BOOLEAN
	IMPORT

EXTRUDE Extrude a 2D shape to create a protrusion or cut-out in a solid object.

REVOLVE Create a solid object, or a cut-out in a solid object, by revolving a 2D shape around an axis.

DRIVE Drive a 2D section along a trajectory to create a protrusion or cut-out in a solid object.

HOLE Create shaped or straight hole(s).

SHAFT Create shaped or straight shaft(s).

SHELL Remove material from a solid object to create a shell.

RIB Create a rib between the faces of a concave model.

BOOLEAN Perform operations between objects, surfaces, and datums.

IMPORT Import solid objects from another file into the current file.

CREATE >> EXTRUDE

Create a new solid object; create a protrusion or cut-out in an existing solid object.

SELECT	ADD
	REMOVE
	DIVIDE
	DATUM
	NEW

ADD Create a protrusion by adding material.

REMOVE Create a cut-out by removing material.

DIVIDE Divide solid object into two new objects.

DATUM Create a datum based on a 2D shape.

NEW Begin a new object.

Notes:

- DIVIDE, ADD, and REMOVE will only appear if a solid object has already been created.
- For ADD/REMOVE: If more than one solid object exists, the following prompt will appear:

  PICK ACTIVE OBJECT

EXTRUDE >> All Options

The EXTRUDE options have the same interactions.

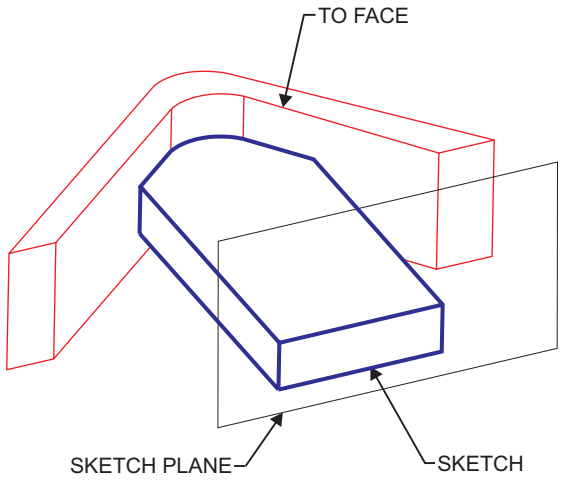


Figure 2-8: EXTRUDE >> TO FACE

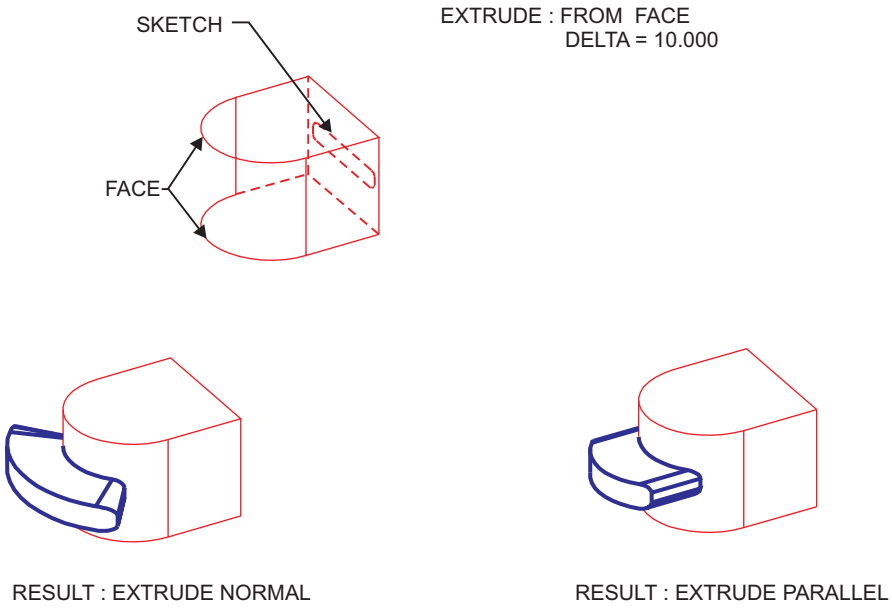


Figure 2-9: EXTRUDE >> FROM FACE WITH DELTA

PICK FACE/PLANE <PICK> a surface on which to perform the extrusion.

You are now in the **Sketcher**.

IND. 1ST POINT	DISPLAY	LINE	CORNER		ERASE	EXTERN	PREVIEW	EXIT

IND. 1ST POINT Indicate the first point of the contour on the FACE / PLANE.

DRAW Sketch and dimension a 2D shape, and then select EXIT. For details on using the Sketcher, see Sketcher Functions, page 1-6.

INDICATE AREA Indicate the area which will be extruded. This prompt is only displayed if an open contour has been sketched.

SELECT OPTION	DELTA	THROUGH	FROM	TO	APPLY

DELTA Enter the depth of the cut-out or protrusion.

"FULL FACE" OPTION If the option ADD was selected, and if the plane selected for sketching is a SOLID FACE (not a REFERENCE FACE), then the closed boundary of the face may be used as the section contour.

Note: • After picking the face as the reference plane, the Sketcher is invoked. Simply exit the Sketcher (by picking EXIT) without doing anything. **Cimatron** will assume you want to extrude the entire face.

THROUGH This applies only in the REMOVE and DIVIDE options, and the following interaction will occur:

<CR> TO CONTINUE	ONE SIDE / BOTH SIDES	FLIP SIDE

FROM Select the starting face, plane or reference surface.

PICK FROM PLANE / FACE	PLANE →

PLANE
FACE
SURFACE

- PLANE — Select a plane.
- FACE — Select a face.
- SURFACE — Select a reference surface.

(The option FROM must be used together with one of the options DELTA or TO).

TO Select the stopping face, plane or reference surface.

PICK TO PLANE / FACE	PLANE →

PLANE
FACE
SURFACE

- PLANE — Select a plane.
- FACE — Select a face.
- SURFACE — Select a reference surface.

APPLY Apply the selections.

When selecting FROM in conjunction with the DELTA option, there are two choices: NORMAL and PARALLEL.

<CR> TO CONTINUE	DELTA=100.00	NORMAL / PARALLEL

CREATE >> REVOLVE

Revolve a 2D contour about an axis to create a new solid object or datum, or create a protrusion, cut-out, or divide in an existing object.

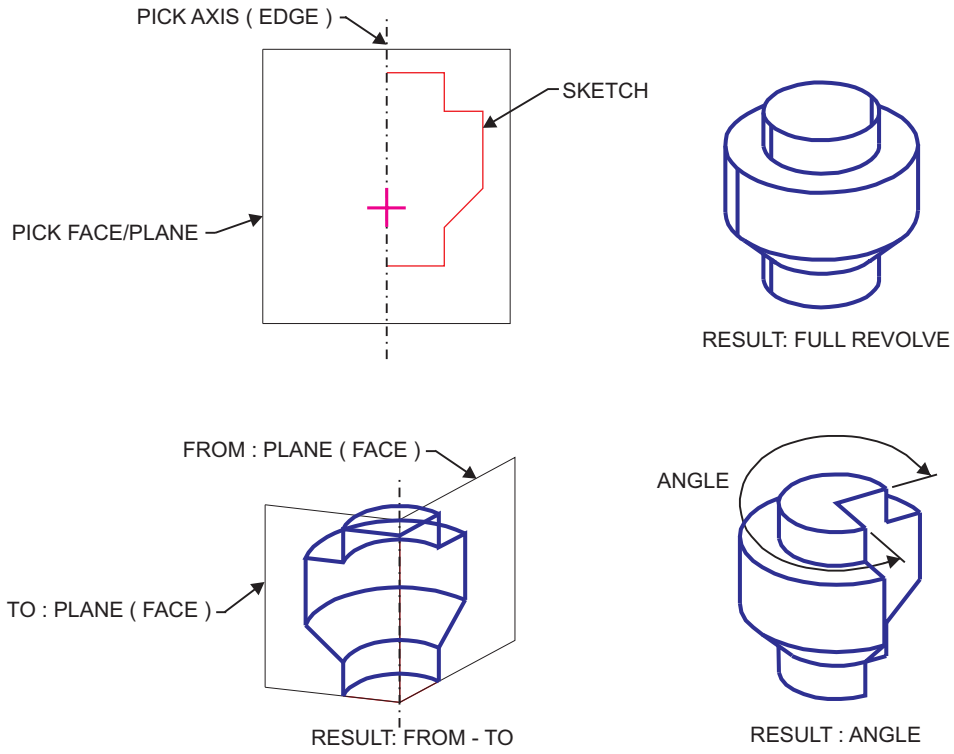


Figure 2-10: REVOLVE

How To:

1. Choose the appropriate option (ADD, REMOVE, DIVIDE, DATUM, NEW). If this is the first feature you are creating, only ADD and DATUM are available.
2. <PICK> the axis of revolution.
3. <PICK> a plane/face on which to sketch a 2D contour. The contour will be revolved about the selected axis.
4. You are now in the **Sketcher**. The selected axis will serve as a reference when sketching.
   Sketch and dimension a 2D feature, then EXIT the **Sketcher**.
5. If the contour is open, indicate which area will be revolved about the axis.
6. Define the extent of the revolution. Either enter an angle or select the faces (planes) between which the protrusion or cut-out will extend.

SELECT	ADD
	REMOVE
	DIVIDE
	DATUM
	NEW

ADD Create a protrusion by adding material.

REMOVE Create a cut-out by removing material.

DIVIDE Divide solid object into two new objects.

DATUM Create a datum based on a 2D shape.

NEW Begin a new object.

Notes:

- DIVIDE, ADD, and REMOVE will only appear if a solid object has already been created.
- For ADD/REMOVE: If more than one solid object exists, the following prompt will appear:

  PICK ACTIVE OBJECT

REVOLVE >> All Options

PICK AXIS/EDGE <PICK> an axis or edge about which the contour will be revolved and confirm your choice.

PICK FACE / PLANE <PICK> the face/plane that will be sketched on.

You are now in the **Sketcher**.

IND. 1ST POINT	DISPLAY	LINE	CORNER		ERASE	EXTERN	PREVIEW	EXIT

IND 1ST POINT Indicate the first point of the contour on the FACE / PLANE.

DRAW Sketch and dimension a 2D shape. The selected axis will serve as a reference for the **Sketcher**. Then pick <EXIT>.

INDICATE AREA Indicate the area which will be revolved. This prompt is only displayed if an open contour has been sketched.

Next select the extent of the revolution:

SELECT OPTION	DELTA	FULL	FROM	TO	APPLY

DELTA Enter the angle of revolution in degrees.

KEY IN ANGLE	ANGLE = 86.000

- ANGLE = 90.000 The extent revolution in degrees.

"FULL FACE" OPTION If the option ADD was selected, and the plane selected for sketching is a SOLID FACE, then the closed boundary of the face may be used as the section contour.

Note: • After picking the face as the reference plane, the Sketcher is invoked. Simply exit the Sketcher (by picking EXIT) without doing anything. **Cimatron** will assume you want to revolve the entire face.

FULL (360 DEG) The 2D shape will be revolved 360 degrees about the selected axis.

If you chose this option, the extent of revolution will no longer be a parameter and you will not be able to change it using the EDIT sub-functions PARAMETERS or RELATION.

FROM Select the starting face, plane, or surface.

PICK FROM PLANE / FACE	PLANE →	PLANE
		FACE
		SURFACE

- PLANE Select a plane.
- FACE Select a face.
- SURFACE Select a reference surface.

TO Select the stopping face, plane or surface.

PICK TO PLANE / FACE	PLANE →	PLANE
		FACE
		SURFACE

- PLANE Select a plane.
- FACE Select a face.
- SURFACE Select a reference surface.

INDICATE DIRECTION Indicate the direction (CW or CCW).

APPLY Apply the selections to the figure. The exhibited figure will reflect those selections.

CREATE >> DRIVE

Create a protrusion or cut-out by driving blended sections along a single/several trajectory contours. This may also result in a new object or reference object.

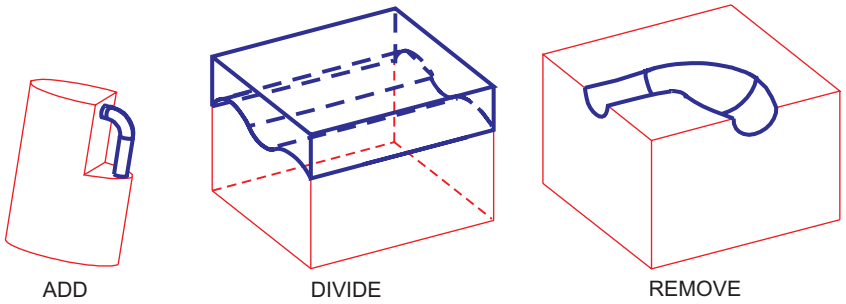


Figure 2-11: DRIVE >> All Options

How To:

1. Define the trajectory along which the section will be driven, normal to the trajectory or parallel to the user defined plane. The definition is performed by picking or sketching the trajectory on the desired plane/face.
2. Define additional trajectories if required.
3. Sketch one/several sections.

If the drive is initiated when a model already exists, the following options appear:

SELECT	ADD
	REMOVE
	DIVIDE
	DATUM
	NEW

ADD Create a solid object or protrusion in an existing object.

REMOVE Create a cut-out in an existing object.

DIVIDE Divide an existing object.

DATUM Create a reference object.

NEW Create a new solid object in addition to the currently existing one.

Notes:

- If this option is used to create the first feature, only the NEW and DATUM options are displayed. After selecting one of these options, the Sketcher is entered allowing you to sketch the trajectory on the active or user defined plane.
- If two or more objects already exit, the following prompt is displayed:
  PICK ACTIVE OBJECT
  Pick the object whose drive will be added, removed or divided.

DRIVE >> All Options

1. Define the trajectory

After selecting an option, the following prompt appears:

SELECT	PICK OR SKETCH TRAJECTORY

Click <SUBMENU> to display the list of options that can be used to define the trajectory.

SINGLE
LOOP
CHAIN
SKETCH

Notes:

- The trajectory can be a line, curve, a spline or a sequence of these.
- The first trajectory picked is the main trajectory. Its endpoints are used to define a temporary UCS to indicate planes for section sketching. After the main trajectory is defined, the prompt MORE TRAJECTORIES? YES/NO appears. Picking YES allows you to define secondary trajectories.
- When working with two or more trajectories, the selected beginning or end planes for the drive must intersect all the trajectories.
- The main trajectory appears in red. The secondary trajectories, if any are present, appear in green.
- If more than one trajectory is defined, or if the trajectory has break points, only one section can be defined.

DRIVE >> All options >> SINGLE

Pick an edge or curve to define the trajectory.

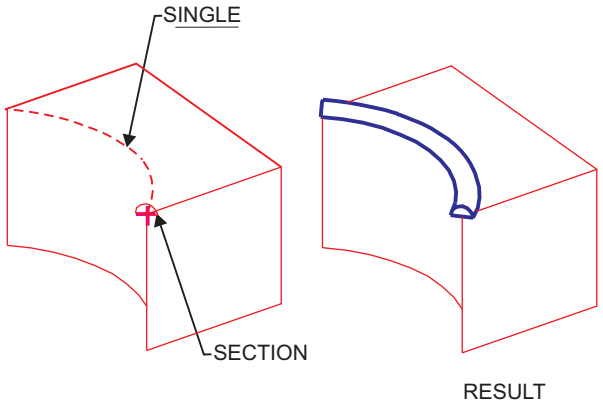


Figure 2-12: DRIVE >> SINGLE

When this option is selected, the following prompt appears:

PICK EDGE/CURVE	PICK OR SKETCH TRAJECTORY - SINGLE

Pick an edge or curve as the trajectory. The temporary UCS will be located at end of the trajectory and normal to it, to define the plane for section sketching.

Click <SUBMENU> to obtain options that can be used to define the new trajectory.

If you only need one trajectory for the drive, press <EXIT> and continue by defining a section as described on page 2-17.

DRIVE >> All options >> LOOP

Pick a loop to define the trajectory.

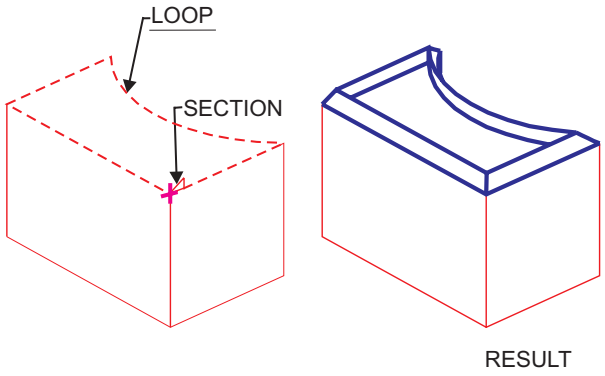


Figure 2-13: DRIVE >> LOOP

When this option is selected, the following prompt appears:

PICK LOOP	PICK OR SKETCH TRAJECTORY - LOOP

Pick a loop, i.e., a closed boundary of face edges. The temporary UCS will be located at end of the trajectory, and normal to it. Continue by defining a section as described on page 2-17 below.

Notes:

- If the loop you selected has breakpoints, only one trajectory drive may be defined. After the drive has been defined, the **Sketcher** is entered.
- If the loop is 3D, the following prompt appears:

PICK PLANE	PICK OR SKETCH TRAJECTORY - TRAJECTORY PLANE

<PICK> the reference plane. The temporary UCS will be located at end of the trajectory normal to the plane. The **Sketcher** is entered.

DRIVE >> All options >> CHAIN

Pick a chain to define the trajectory.

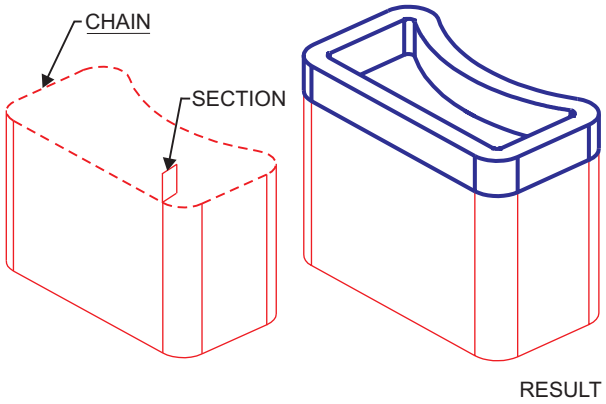


Figure 2-14: DRIVE >> CHAIN

When this option is selected, the following prompt appears:

PICK EDGE	PICK OR SKETCH TRAJECTORY - CHAIN

Pick a chain, i.e., 2D/3D sequence without breakpoints. The edges are displayed in attention mode and the following prompt appears:

PICK PLANE	PICK OR SKETCH TRAJECTORY - TRAJECTORY PLANE

<PICK> the reference plane.

The temporary UCS will be located at end of the trajectory, and normal to it. Add a new trajectory or press <EXIT> to continue by defining a section as described on page 2-17.

DRIVE >> All options >> SKETCH

Pick a face or plane to define the trajectory.

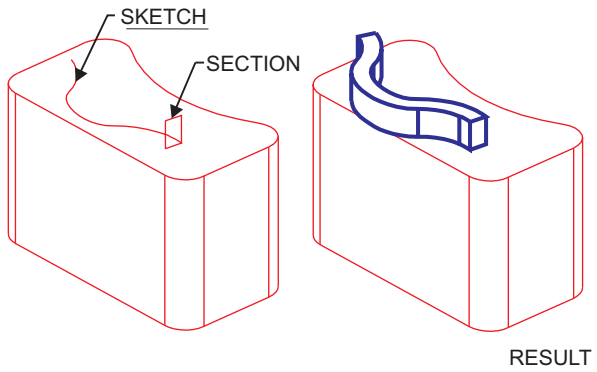


Figure 2-15: DRIVE >> SKETCH

When this option is selected, the following prompt appears:

PICK SKETCH PLANE

<PICK> the face or plane onto which the trajectory will be sketched. The face is displayed in attention mode, and the **Sketcher** is entered. Sketch and dimension the trajectory contour and select <EXIT>.

A temporary UCS indicates the active location and section plane. Add a new trajectory or press <EXIT> to continue by defining a section as described on page 2-22.

2. Define the section

LOC. SECTION	SKETCH SECTION	PARALLEL	MODIFY PLANE
		NORMAL	

LOC. SECTION Locate a section. A section may be located at every trajectory end. Endpoints are highlighted when the cursor is moved toward them.

Note:

- For two or more trajectories or for a non-smooth trajectory, only one section can be defined. The section may be located at the beginning or the end of the trajectory.

For multi-trajectory

After section sketch dimensioning, the following prompt appears:

PICK VARIABLE/EXIT Pick EXIT if the section dimensions must be constant. <PICK> the dimension that must be variable to define its behavior along trajectory. When the dimension is selected, the following prompt appears:

PICK SKETCH PLANE Pick the face or plane on which the graph will be sketched. The face and the first trajectory projection is displayed in attention mode for reference.

Sketch and dimension the graph and select <EXIT>.

For an open trajectory and one section

SELECT OPTION	FROM	TO	APPLY

■ FROM Start from a plane or face.

PICK FROM PLANE	PLANE
	FACE
	SURFACE

PLANE

PICK FROM PLANE <PICK> the planar face from which to start.

FACE

PICK FROM FACE <PICK> the non-planar face from which to start.

SURFACE

PICK FROM SURFACE <PICK> the surface from which to start.

■ TO End at a plane or face.

PICK TO PLANE	PLANE
	FACE
	SURFACE

PLANE

PICK TO PLANE <PICK> the planar face at which to end.

FACE

PICK TO FACE <PICK> the non-planar face at which to end.

SURFACE

PICK TO SURFACE <PICK> the surface at which to end.

For a closed trajectory and a single open section (only applies to DIVIDE operation)

ADD CAPPING FACE	YES
	NO

YES Create a planar face limited by the inner patch of a section contour.

NO Create an intersection between the section contour and the trajectory plane.

For multi-sections

In the case of multi-sections, the sections consist of more than one edge each, the number of edges must be identical in all sections. A red point will indicate the endpoint of every section contour.

IND. END POINT/EXIT The endpoint of every section contour can be redefined by picking it. All section endpoints will be connected by the DRIVE operation. Incorrect selection of section endpoints will lead to a function failure.

Note:

- When working with multi trajectories, the planes chosen as the beginning and end for the drive must intersect the trajectory. They can be picked regardless of the endpoints of the trajectory.

APPLY Perform the DRIVE operation.

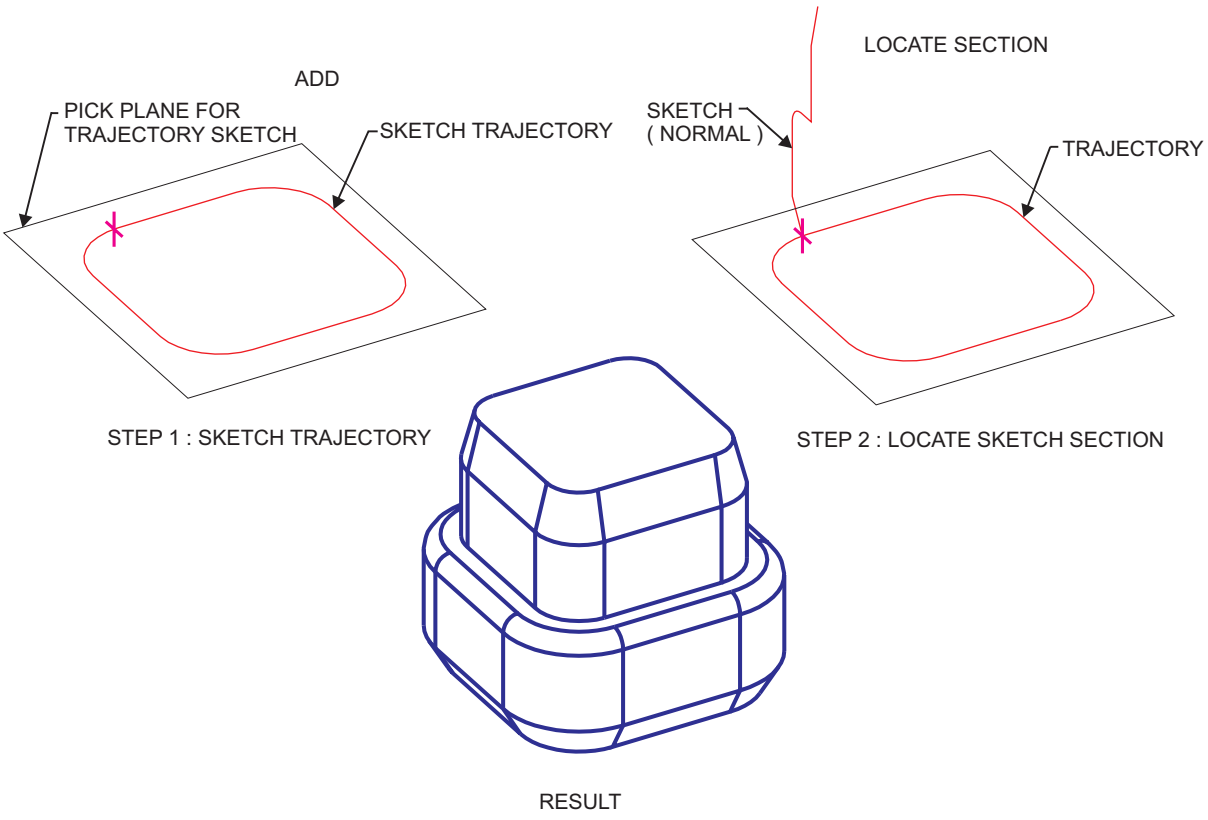


Figure 2-16: Closed Trajectory

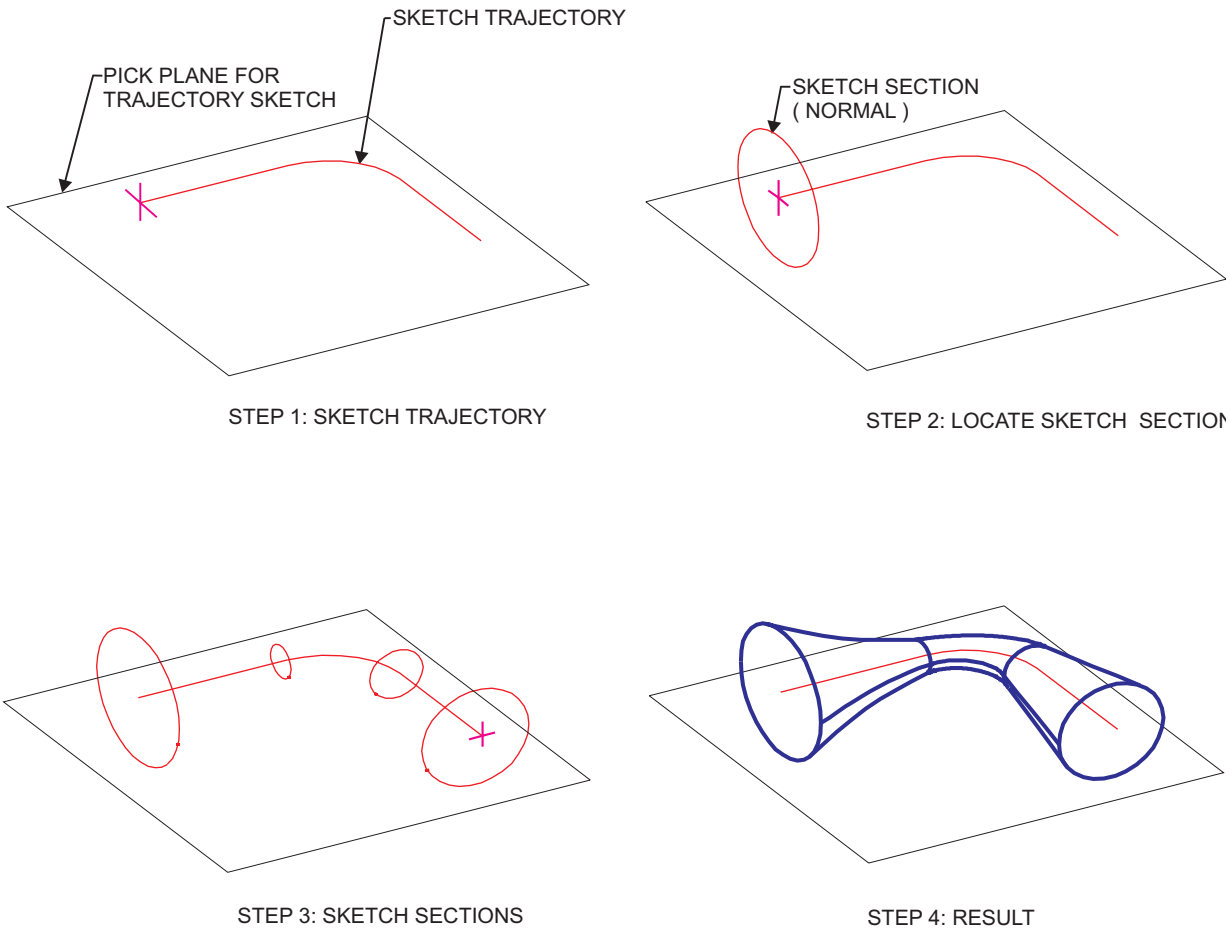


Figure 2-17: DRIVE >> ADD (Open Trajectory)

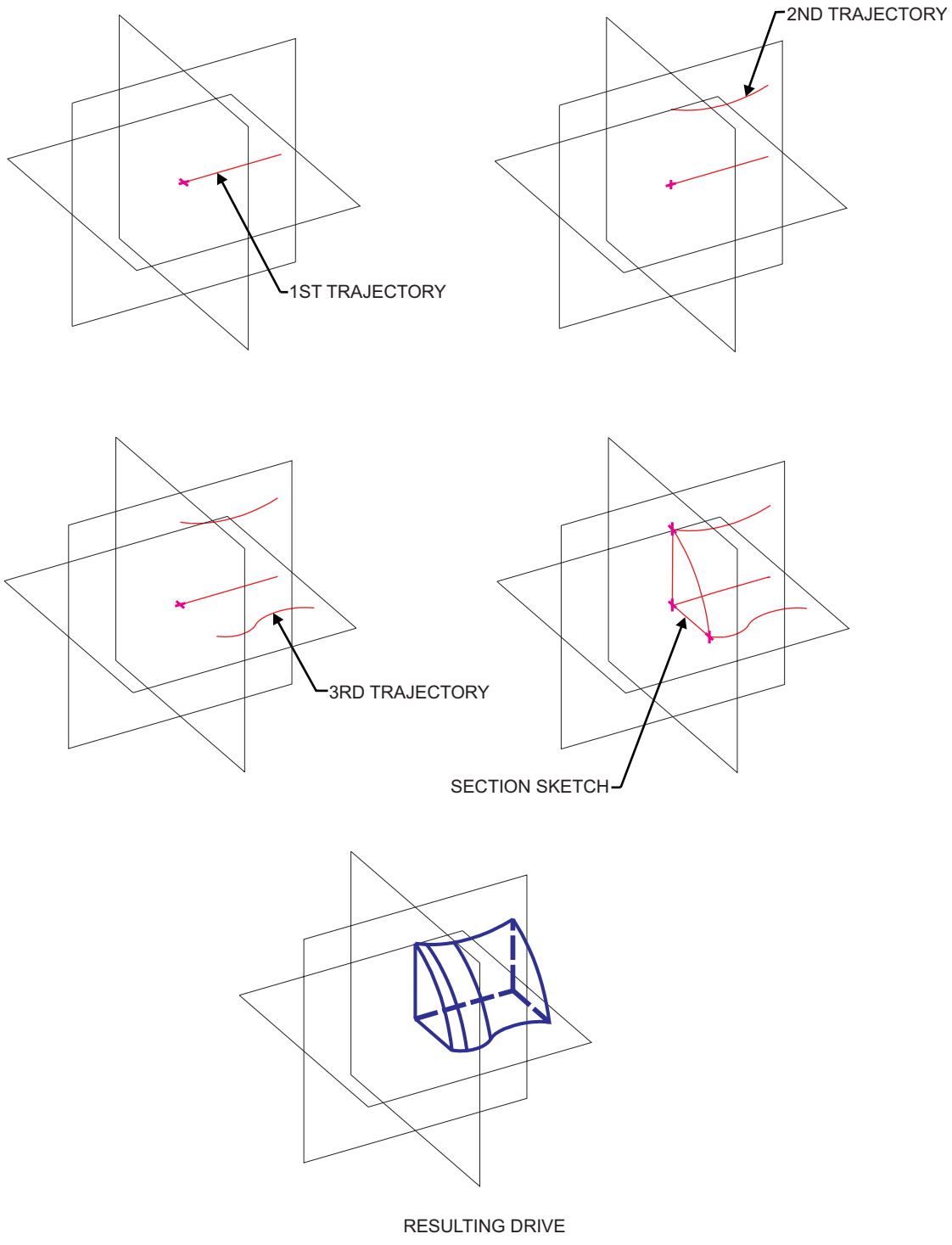


Figure 2-18: Multi Trajectory

CREATE >> HOLE

Create shaped or straight hole(s).

SELECT	SHAPED
	STRAIGHT
	AXIS

SHAPED Create shaped hole(s) by revolving a section contour around a reference axis.

STRAIGHT Create straight hole(s).

AXIS Create straight hole(s) along axis.

Notes:

- Multiple holes with the same section can be created in one operation.
- Edges, datum/axes and datum/curves may be used as axes.
- Multiple holes must not intersect.

HOLE >> SHAPED

Create shaped hole(s).

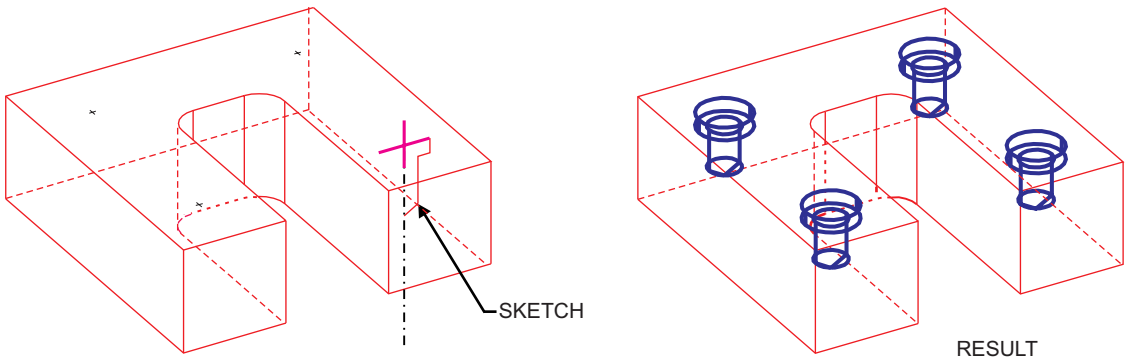


Figure 2-19: HOLE >> SHAPED

PICK FACE / PLANE <PICK> a face or plane upon which you wish to sketch.

You will enter the **Sketcher**.

IND. POINT	DISPLAY	POINT			ERASE	EXTERN	PREVIEW	EXIT

Indicate the location of the center point of each hole. Dimension the locations of these points, then select EXIT.

A "+" symbol appears, indicating the reference point and a plane on which to sketch the section of the hole. A reference axis indicating the centerline of the hole will also appear.

Sketch one half of the hole section with respect to the reference axis, using an open contour. The sketch must begin and end on the reference axis, and either the start or end point of the sketch must lie on the "+" symbol reference point. Dimension the contour with respect to the reference point and axis, then select EXIT.

HOLE >> STRAIGHT

Create straight through or blind hole(s).

PICK FACE / PLANE <PICK> a face or plane upon which you wish to sketch.

You will enter the **Sketcher**.

IND. POINT	DISPLAY	POINT			ERASE	EXTERN	PREVIEW	EXIT

Indicate the location of the center point of each hole. Dimension the locations of these points, then select EXIT.

<CR> TO CONTINUE	DIAMETER=10.0	DELTA / THRU ALL

HOLE >> AXIS

Create straight hole(s) along an axis.

PICK AXIS

- DIAMETER=10.000 — Choose the diameter of the hole.

SELECT OPTION	THROUGH	FROM	TO	- APPLY -

- THROUGH — The hole will pass through the solid object along the axis.
- FROM — Select the plane, face, or surface from which the hole will start.

PICK FROM PLANE / FACE	PLANE →

PLANE
FACE
SURFACE

- TO — Select the plane, face, or surface at which the hole will terminate. This option works in conjunction with the FROM option.

PICK FROM PLANE / FACE	PLANE →

PLANE
FACE
SURFACE

- APPLY — Activate with all the conditions fulfilled.

CREATE >> SHAFT

Create shaped or straight shaft(s).

SELECT	SHAPED
	STRAIGHT

SHAPED Create shaped shaft(s) by revolving a section contour around a reference axis.

STRAIGHT Create straight shaft(s).

Notes:

- Multiple shafts with the same section can be created in one operation.
- Edges, Datum/axes and datum/curves may be used as axes.
- Multiple shafts must not intersect.

SHAFT >> SHAPED

Create shaped shaft(s) by revolving a section contour around an axis.

PICK FACE / PLANE <PICK> a face or plane upon which you wish to create the shaft(s).

You will enter the **Sketcher**.

IND. 1ST POINT	DISPLAY	POINT			ERASE	EXTERN	PREVIEW	EXIT

Indicate the location of the center point of each shaft. Dimension the locations of these points, if you wish, then select EXIT.

A "+" symbol appears, indicating the reference point and a plane on which to sketch the section of the shaft. A reference axis indicating the centerline of the shaft will also appear.

Sketch one half of the shaft section with respect to the reference axis, using an open contour. The sketch must begin and end on the reference axis, and either the start or end point of the sketch must lie on the "+" symbol reference point. Dimension the contour with respect to the reference point and axis, then select EXIT.

SHAFT >> STRAIGHT

Create straight shaft(s).

PICK FACE / PLANE <PICK> a face or plane upon which you wish to create the shaft(s).

You will enter the **Sketcher**.

IND. 1ST POINT	DISPLAY	POINT			ERASE	EXTERN	PREVIEW	EXIT

Indicate the location of the center point of each shaft. Dimension the locations of these points, if you wish, then select EXIT.

<CR> TO CONTINUE	DIAMETER = 1.000

<CR> TO CONTINUE Press <ENTER> to continue.

KEY IN DELTA	DELTA = 10.000

KEY IN DELTA Enter the height of the shaft(s). Press <CR> to execute.

CREATE >> SHELL

Create a cavity in a solid body to form a shell.

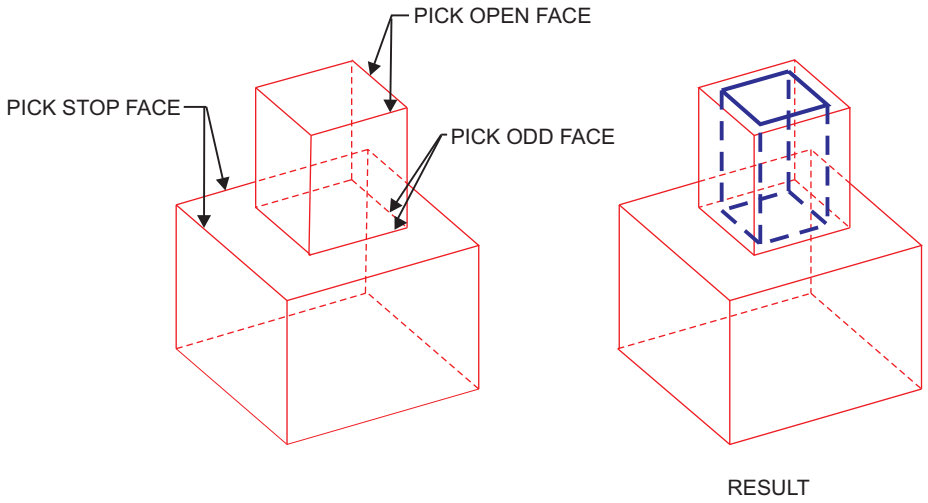


Figure 2-20: CREATE >> SHELL

How To:

1. Pick the object from which you wish to create a shell.
2. Specify the shell direction.
3. Specify the general thickness of the shell.
4. Define which faces are to be open, if any.
5. Define which faces are to be stop faces, if any.
6. <PICK> odd faces (faces which will be assigned a different thickness), if any.
7. Specify the thickness of the odd faces, if any.
8. Press <EXIT> to execute.

PICK OBJECT Pick the object from which you wish to create a shell.

INDICATE DIRECTION Choose the arrowhead indicating the direction of the cavity.

PICK OPEN FACE / EXIT	GENERAL THICK = 10.000

PICK OPEN FACE / EXIT Select open face(s), if any, and exit.

PICK STOP FACE / EXIT	GENERAL THICK = 10.000

PICK STOP FACE <PICK> face(s) which will be the boundary of the cavity, if any, then <EXIT>.

PICK ODD FACE / EXIT	ODD THICK = 20.000

PICK ODD FACE <PICK> face(s), if any, which will have a thickness different than the general thickness, then <EXIT>.

Press <EXIT> to execute.

Note:

- If the SHELL operation cannot be executed due to the self-intersection of the resulting faces, the loops with the self-intersections are displayed by yellow curves. To remove these yellow curves, press CTRL+R.

If no open faces exist;

PICK START FACE	GENERAL THICK=

PICK START FACE Select the face to begin the cavity.

■ GENERAL THICK = Enter a value for the general thickness.

CREATE >> RIB

Create a rib between the faces of a concave model, or a shell.

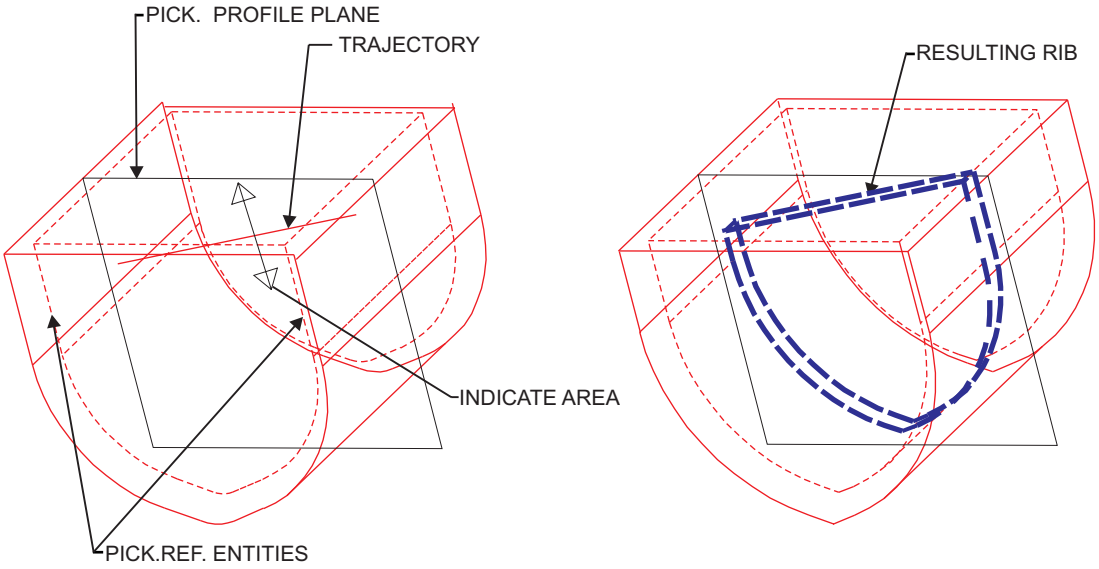


Figure 2-21: CREATE >> RIB

How To:

1. Create a plane on which the rib trajectory will lie, or choose an existing plane, face, or planar surface..
2. Draw the trajectory.
3. Choose draft angle and type of top.
4. Choose reference plane if needed.

PICK PROFILE PLANE <PICK> the plane on which the trajectory will be sketched.

SELECT	PICK REFERENCES	IND. +X AXIS

See “Sketching on a Plane”, page 1-20.

IND. 1ST POINT	DISPLAY	LINE	CORNER		ERASE	EXTERN	PREVIEW	EXIT

Draw the trajectory.

INDICATE AREA Choose direction towards face for rib to fill.

<CR> TO CONTINUE	WIDTH=	DRAFT ANGLE=	FLAT TOP / ROUND TOP

If the draft angle is not set at 0, the following prompt will appear:

PICK REF PLANE/EXIT <PICK> a plane that intersects the trajectory plane. The draft angle will be measured from this plane.

The cross section is constant relative to this plane.

Note:

- The rib will expand to the first face of a shell on a concave model found in the direction of the trajectory. If the extension of the trajectory does not find a face, an error will result.

CREATE >> BOOLEAN

Perform operations between objects, surfaces, and datums/planes.

SELECT	ADD
	REMOVE
	CUT
	DIVIDE

ADD Merge two objects into one object

REMOVE Remove material from a solid object by using a second object to cut it.

CUT Trim a surface by using a cutting object.

DIVIDE Divide one object into two independent objects.

BOOLEAN >> ADD

Merge two objects into one object.

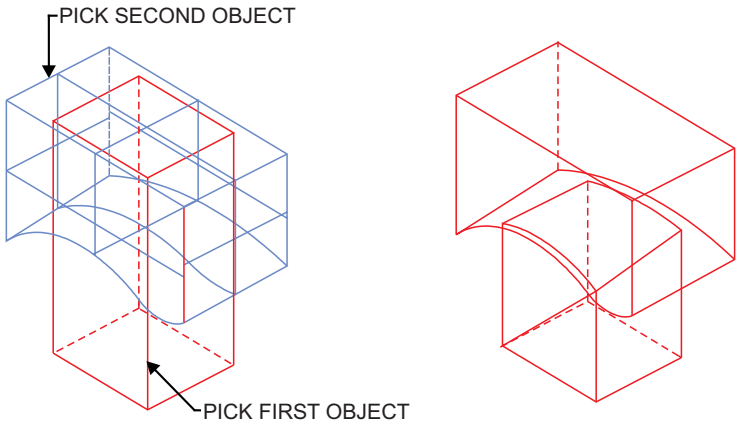


Figure 2-22: BOOLEAN >> ADD

PICK FIRST OBJECT <PICK> the first object to be merged. The object may be an open or closed solid, or a surface. If a surface or an open solid is picked, the following prompt will appear:

DEFINE OUTSIDE DIRECTION Choose the arrowhead indicating the outside direction.

PICK SECOND OBJECT <PICK> the second object to be merged. If a surface or an open solid is picked, you will be asked to define the outside direction as described above.

BOOLEAN >> REMOVE

Remove material from a solid object by using a second object to cut it. The second object may be a plane, surface, or a solid object.

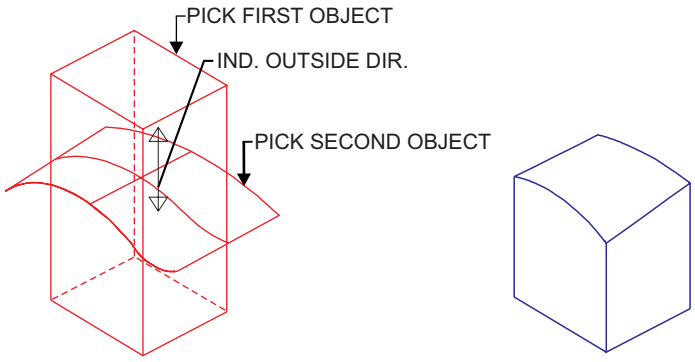


Figure 2-23: BOOLEAN >> REMOVE

PICK FIRST OBJECT

<PICK> a closed or open solid object from which material is to be removed.

PICK PLANE/OBJECT Pick an open or closed solid object, surface, or datum plane that is to be used as the cutting surface.

IND. REMAINING SIDE Choose which side of the first object is to remain.

If a solid object or a surface was chosen as the cutting object, the following prompt will appear:

KEEP THE SECOND OBJECT? YES NO

Choose whether or not the cutting object will remain.

BOOLEAN >> CUT

Trim a surface by using a cutting object.

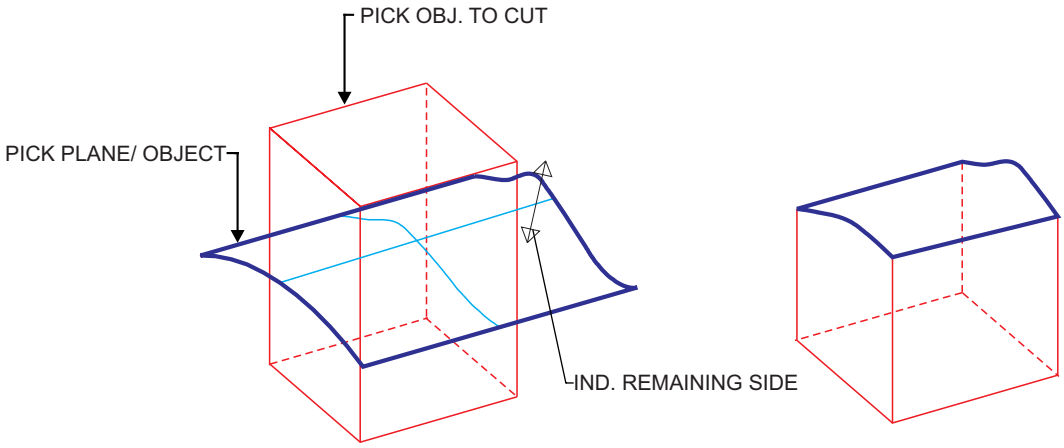


Figure 2-24: BOOLEAN >> CUT

PICK OBJ. TO CUT <PICK> the surface to be cut.

PICK PLANE/OBJECT <PICK> the plane, surface, or solid by which to cut the first object.

IND. REMAINING SIDE Choose the side of the first object that will remain.

Note:

- The intersection between the cut surface and the cutting object must be an open or closed contour. The boundaries of the cut surface must be contained within the cutting object.
- The result of the cut operation is an open object.

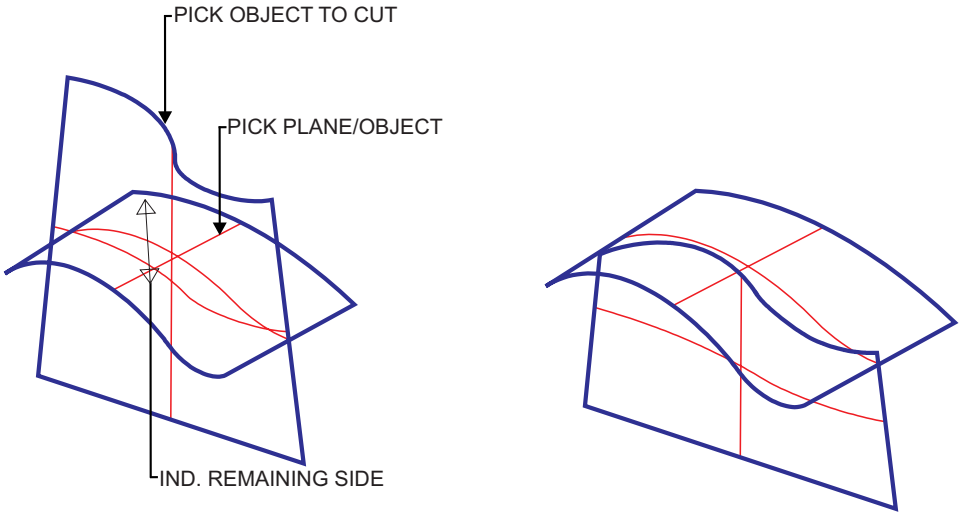


Figure 2-25: BOOLEAN >> CUT (TRMSRF)

BOOLEAN >> DIVIDE

Divide one object into two independent objects. The dividing can be done by a plane, surface, or solid.

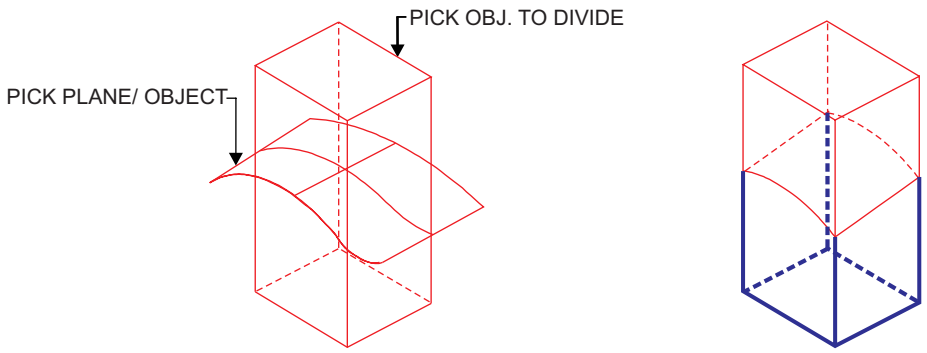


Figure 2-26: BOOLEAN >> DIVIDE

PICK OBJECT
<PICK> an object to be divided.

PICK PLANE/OBJECT
<PICK> the plane, surface, or solid by which to divide the first object.

CREATE >> IMPORT

Import solid objects from another file into the current file.

ENTER FILE NAME
Enter the name of the source file (the file from which you will be importing objects).

IND. ORIGIN/EXIT	FLIP

IND ORIGIN / EXIT
IND. +X AXIS/EXIT
IND. +Y AXIS/EXIT
Indicate the UCS selection.

It is possible to flip the direction as necessary. Press <EXIT> when finished.

POINTS OK? Y/N

YES Accept the points. All objects will be imported from the source file and will be placed at the UCS indicated. The import is, in effect, a copy operation from UCS to UCS.

NO Re-indicate the origin.

CREATE: Modal Parameter Definitions

- BOTH SIDES — Remove / divide material from both sides.
- DELTA = 10.000 — Enter the depth of the hole(s).Press <CR> to execute.
- DELTA = 10.000 — Enter the height of the shaft(s). Press <CR> to execute.
- DIAMETER = 1.000 — Enter the diameter of the hole(s)/shaft(s)

KEY IN DELTA	DELTA = 10.000

- DRAFT ANGLE — Angle between rib sides.
- FLAT TOP — Rib ends with face normal to profile plane.
- FLIP — Reverse the indicated direction, if desired.
- FLIP SIDE — Selects the alternate side.
- GENERAL THICK = 10.000 — Specify the general thickness of the shell.
- MODIFY PLANE — Change the sketching plane for the current section. The temporary UCS will be located parallel to the selected plane.

Note: This option is not available when DRIVE is used to create the first sketch in the active level.

- NORMAL — For NORMAL, the addition extends from the face normal to it.
- NORMAL — All the sections will be normal to the trajectory.
- ODD THICK = 4.000 — Specify the thickness of the odd face, if any.
- ONE SIDE — Remove / divide material from one side.
- PARALLEL — All the sections will be parallel to one another.
- PARALLEL — For PARALLEL, the addition extends are normal to the sketching plane.
- ROUND TOP — Top of rib is rounded off. The radius of curvature is on half of the rib width.
- SKETCH SECTION — Enter the ***Sketcher*** and sketch the section.
- THRU ALL — The hole(s) will extend through the entire object.
- WIDTH — Width of rib. See Figure 2-27, next page.

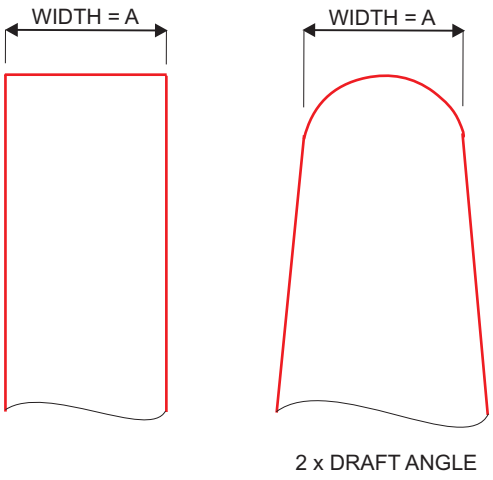


Figure 2-27: Rib Parameters

CREATE: Usage Envelope

1. For CREATE >> EXTRUDE >> ADD and CREATE >> REVOLVE >> ADD: when you want to extrude an entire face, you may use the FULL FACE option. After picking the face as the reference plane, the Sketcher is invoked. Simply exit the Sketcher (by picking EXIT) without doing anything. Cimatron will assume you want to extrude the entire face.

   When using the full-face, only the DELTA option is available.

2. In some cases when using CREATE >> SHELL, the shell will fail due to problems with offsets of ROUND faces. Therefore, it is recommended to try creating the same shell without rounds or before the rounds.

3. In certain rare cases when using CREATE >> DRIVE, the following message will appear: CANNOT SET SKETCHER PLANE. This usually happens when driving along a 3D curve or edge. If you selected a reference plane normal to one of the slopes at the end of the drive curve, the plane is identical to the Sketcher plane at that point. In such a case, the sketch cannot be created.

   You must go back using REJECT, and define a different reference plane.

   ❑

DATUM

Create reference planes, axes, curves, and points

Main Options:

PLANE
AXIS
CURVE
POINT

PLANE Create reference planes.

AXIS Create reference axes.

CURVE Create reference curves.

POINT Create reference points.

DATUM >> PLANE

Create reference planes.

DEFINE PLANE	MAIN
	PARALLEL
	EDGE + POINT
	3 POINTS
	PERP EDGE
	EDGE AT ANG
	2PTS AT ANG
	2PTS PAR EDG
	EDGE PERP PLANE
	REFCRVS

MAIN Create three perpendicular planes according to the model UCS.

PARALLEL Create a plane parallel to a selected face or plane.

EDGE + POINT Create a plane that passes through a selected edge / axes and point.

3 POINTS Create a plane that passes through three selected points.

PERP EDGE Create a plane that is perpendicular to the selected edge/axis and passes through the selected point.

EDGE AT ANG Create a plane that is offset from a selected face or plane by a specific angle through a selected edge/axis.

2PTS AT ANG Create a plane that passes through two selected points and is offset from a selected face or plane by a specified angle.

2PTS PAR EDG Create a plane that passes through two selected points and is parallel to a selected edge/axis.

EDGE PERP PLANE Create a plane that passes through a selected edge/axis and is perpendicular to a selected face or plane.

REFCRVS Create a plane that passes through a 2D reference curve, or two reference lines.

Note: • Once you have created a plane, the following prompt will appear:

RESIZE PLANE <PICK> any corner of the plane and drag it to resize the plane. When you have finished resizing, press <EXIT>. (As the plane is infinite, the displayed lines are only a representation).

PLANE >> MAIN

Three perpendicular planes conforming with the model UCS are created.

PLANE >> PARALLEL

Create a plane parallel to a selected face or plane.

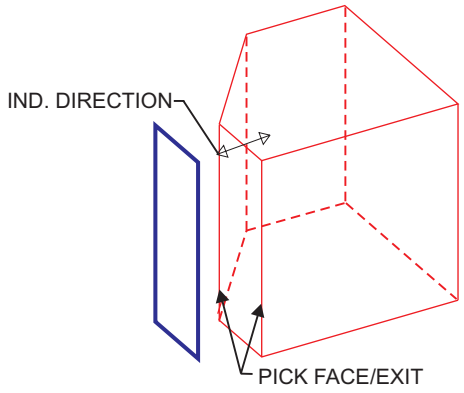


Figure 2-28: PLANE >> PARALLEL

PICK FACE/PLANE	OFFSET = 10.000

PICK FACE/PLANE <PICK> the face or plane from which you wish to offset the new plane.

INDICATE DIRECTION Indicate the direction of the offset.

Note: • If the offset is zero, this prompt will not appear.

PLANE >> EDGE + POINT

Select an edge/axis and a point to define a plane.

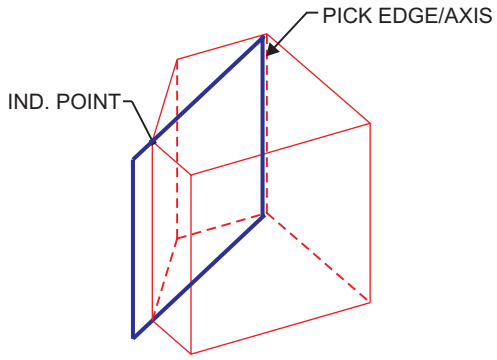


Figure 2-29: PLANE >> EDGE + POINT

PICK EDGE/AXIS

<PICK> the edge/axis which will be used to define the plane.

IND. POINT

Indicate a point to define the plane.

The point indication menu appears in the middle of the screen. The point selection options are explained on page 2-53.

END
MID
CENTER
KEY IN
INTERS
PIERCE
PICK

PLANE >> 3 POINTS

Select three points to define a plane.

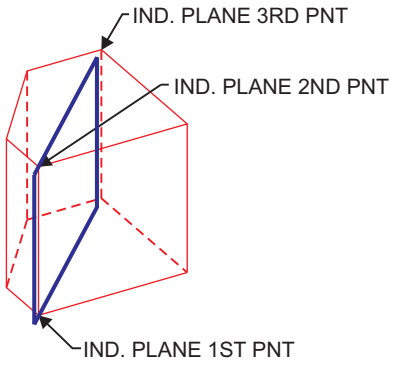


Figure 2-30: PLANE >> 3 POINTS

IND. PLANE 1ST POINT Indicate a point to define the plane.

The point indication menu appears in the middle of the screen. The point selection options are explained on page 2-53.

END
MID
CENTER
KEY IN
INTERS
PIERCE
PICK

IND. PLANE 2ND POINT
IND. PLANE 3RD POINT <PICK> a second and third point, using the point indication menu.

PLANE >> PERP EDGE

Create a plane that is perpendicular to the selected edge/axis and passes through the selected point.

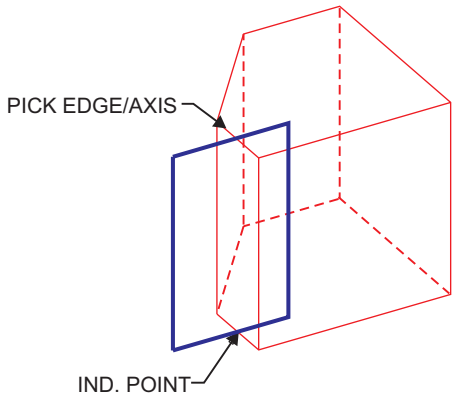


Figure 2-31: PLANE >> PERPENDICULAR EDGE

PICK EDGE/AXIS <PICK> the edge/axis which will be used to define the plane.

IND. POINT Indicate a point through which the plane will pass.

The point indication menu appears in the middle of the screen. The point selection options are explained on page 2-53.

END
MID
CENTER
KEY IN
INTERS
PIERCE
PICK

PLANE >> EDGE AT ANG

Create a plane that is offset from a selected face or plane by a specified angle and passes through a selected edge/axis.

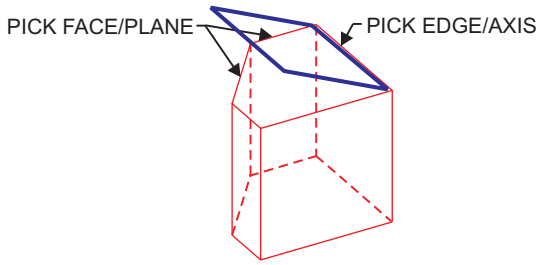


Figure 2-32: PLANE >> EDGE AT ANGLE

PICK EDGE/AXIS	ANGLE = 15.0

PICK EDGE/AXIS <PICK> an edge/axis which the plane will pass through.

PICK FACE/PLANE <PICK> a plane/face from which to rotate the new plane. The selected plane must be parallel to the selected edge/axis.

PLANE >> 2PTS AT ANG

Create a plane that passes through two selected points, and is offset from a selected face or plane by a specified angle.

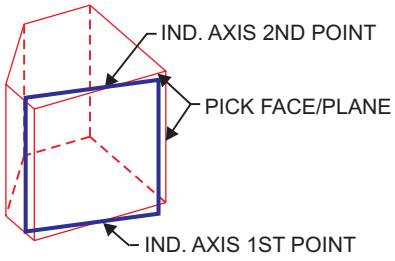


Figure 2-33: PLANE >> 2 POINTS AT AN ANGLE

IND. AXIS 1ST POINT
IND. AXIS 2ND POINT

Indicate two points to create a vector, using the point indication menu that appears in the middle of the screen.

END
MID
CENTER
KEY IN
INTERS
PIERCE
PICK

The new plane will pass through the vector.

The point selection options are explained on page 2-53.

PICK FACE/PLANE	ANGLE=15.0

PICK FACE/PLANE

<PICK> a plane/face from which to offset the new plane. The selected plane must be parallel to the vector you have created.

PLANE >> 2PTS PAR EDG

Create a plane that passes through two selected points and is parallel to a selected edge/axis.

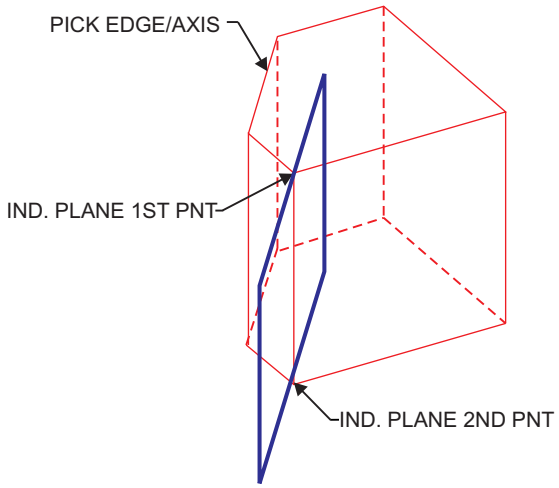


Figure 2-34: PLANE >> 2 POINTS PARALLEL TO AN EDGE

IND. PLANE 1ST POINT
IND. PLANE 2ND POINT

<PICK> two points that the new plane will pass through, using the point indication menu appears in the middle of the screen.

END
MID
CENTER
KEY IN
INTERS
PIERCE
PICK

The point selection options are explained on page 2-53.

PICK EDGE/AXIS

<PICK> an edge/axis to which the plane will be parallel.

PLANE >> EDGE PERP PLANE

Create a plane that passes through a selected edge/axis and is perpendicular to a selected face or plane.

PICK EDGE/AXIS <PICK> the edge/axis through which the plane will pass.

PICK FACE/PLANE <PICK> the plane/face to which the new plane will be perpendicular.

PLANE >> REFCRVS

Create a plane that passes through a 2D reference curve, or two reference lines.

PLANE: PICK REFCRV <PICK> the reference curve.

Note:

- If the reference curve does not define a plane (if the reference curve is a line), the system will prompt:

  PLANE: PICK 2ND REFC

Point Selection Options

The following options are used to select a point:

- END <PICK> the end of an edge.
- MID <PICK> the midpoint of an edge.
- CENTER <PICK> the center of an edge.
- KEY-IN Enter the X, Y and Z coordinates of the point, relative to the model UCS.
- INTERS Select a point at the intersection of two datum curves. The system will prompt you to <PICK> the first and second curves.
- PIERCE Select a point at the intersection of a curve and a plane/surface. The system will prompt you to <PICK> the piercing curve, then the surface/plane.
- PICK <PICK> a reference point. These points are created by the option POINT. For details of this option, see page 2-69.

DATUM >> AXIS

Define a reference axis.

DEFINE AXIS	2 POINTS
	2 PLANES
	PARALLEL
	CYLINDER
	CENTER

2 POINTS Define an axis by indicating two points.

2 PLANES Define an axis by indicating two intersecting planes.

PARALLEL Define an axis by selecting an edge/axis and a plane/face. The axis will be offset a specified distance from the edge. The offset vector will be parallel to the plane/face.

CYLINDER Define an axis by indicating a cylindrical face.

CENTER Define an axis by indicating a circular edge.

Notes:

- If you select AXIS before creating a solid object, a reference axis will be created along the Y axis.
- A reference axis is used, for example, with the command REVOLVE which creates a protrusion or cut-out by revolving a 2D shape about an axis.
- Once you have created an axis using any of these options, the following will appear:

RESIZE AXIS Click and drag either end of the axis to resize it, then <EXIT>. The re-sized axis will be displayed.

AXIS >> 2 POINTS

Define the axis by indicating two points.

IND. AXIS 1ST POINT Indicate the first point which defines the axis.

The point indication menu appears in the middle of the screen. The point selection options are explained on page 2-53.

END
MID
CENTER
KEY IN
INTERS
PIERCE
PICK

IND. AXIS 2ND POINT Indicate the second point which defines the axis, using the point indication menu.

AXIS >> 2 PLANES

Define the axis by indicating two intersecting planes. The axis will be formed by the intersection of the planes.

PICK 1ST PLANE
PICK 2ND PLANE <PICK> two intersecting planes.

AXIS >> PARALLEL

Define an axis by selecting an edge and a plane/face. (If a face is selected, it must be planar).

The axis will be offset a specified distance from the edge. The offset vector will be parallel to the plane/face.

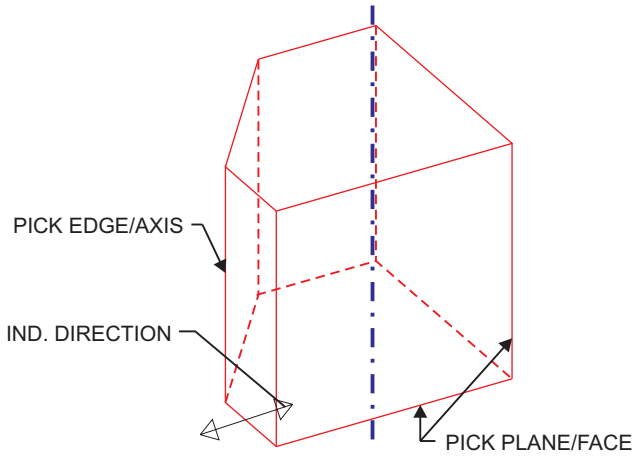


Figure 2-35: AXIS >> PARALLEL

PICK EDGE/AXIS <PICK> the edge of the plane/face.

PICK PLANE/FACE <PICK> a plane/face.

The offset vector will appear. The vector is parallel to the plane/face and perpendicular to the edge.

<CR> TO CONTINUE	DELTA=10.000

INDICATE DIRECTION Indicate the direction in which the axis will be offset from the selected edge.

AXIS >> CYLINDER

Define the axis by picking a cylindrical face.

PICK CYLINDER <PICK> a cylindrical face. The axis will pass through the cylinder.

AXIS >> CENTER

Define the axis by picking a circular edge. The axis will be normal to the plane of the circle and will pass through the circle's center.

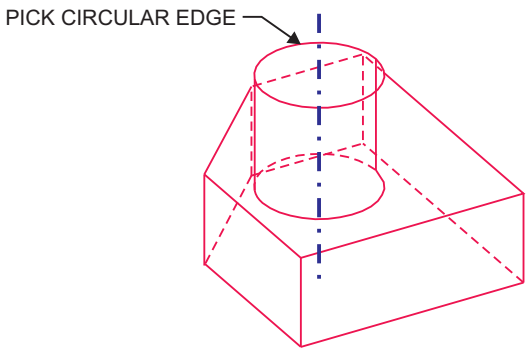


Figure 2-36: AXIS >> CENTER

PICK CIRCULAR EDGE <PICK> a circular edge.

DATUM >> CURVE

Create reference curves. The reference curves are shown by dashed lines.

SKETCH
SRFSEC
EDGE CHAIN
HELIX
SPLINE
LINE NRM
COM CURVE

SKETCH Sketch a curve(s) in the **Sketcher**.

SRFSEC Define a reference curve as a section between two surfaces (faces).

EDGE CHAIN Choose a smooth edge chain to be a reference curve.

HELIX Create a reference curve in the form of a helix.

SPLINE Create a reference curve in the form of a spline.

LINE NRM Create a reference line normal to a given face, plane, edge, curve, or axis.

COM CURVE Join two or more datum curves to form one composite curve.

CURVE >> SKETCH

Sketch a contour in the **Sketcher**.

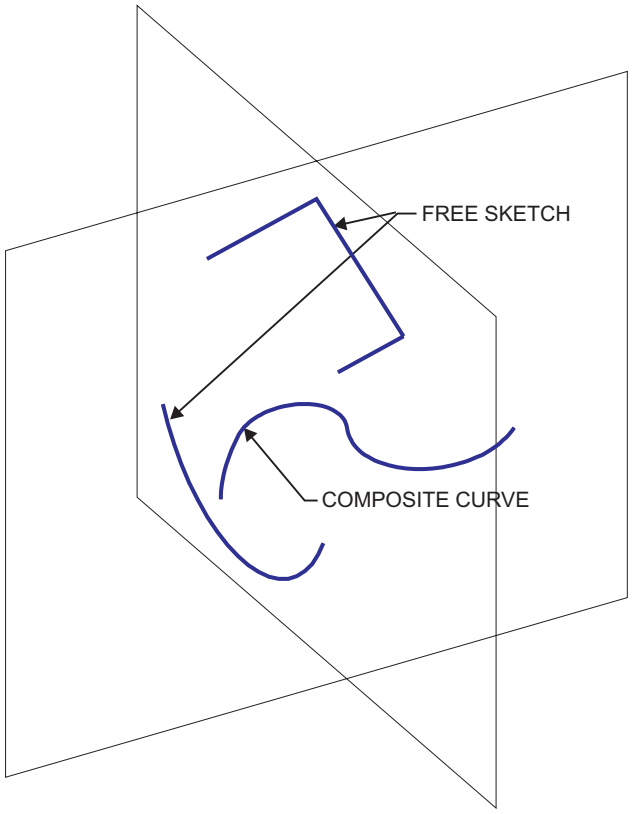


Figure 2-37: CURVE >> SKETCH

SELECT	COMPOSITE CURVE	FREE SKETCH

COMPOSITE CURVE Created reference curves are joined into one composite curve.

FREE SKETCH Each sketched curve is created as a free reference curve.

For details on the limitations when using these options, see the Usage Envelope on page 2-70.

PICK FACE/PLANE Pick a face or plane on the model upon which the reference curve will be sketched. The system will now be in the **Sketcher** mode. After sketching a contour and exiting the **Sketcher**, a contour consisting of reference curves will be created.

CURVE >> SRFSEC

Define a reference curve along the intersection of two surfaces / faces, or the intersection of a surface / face and a plane.

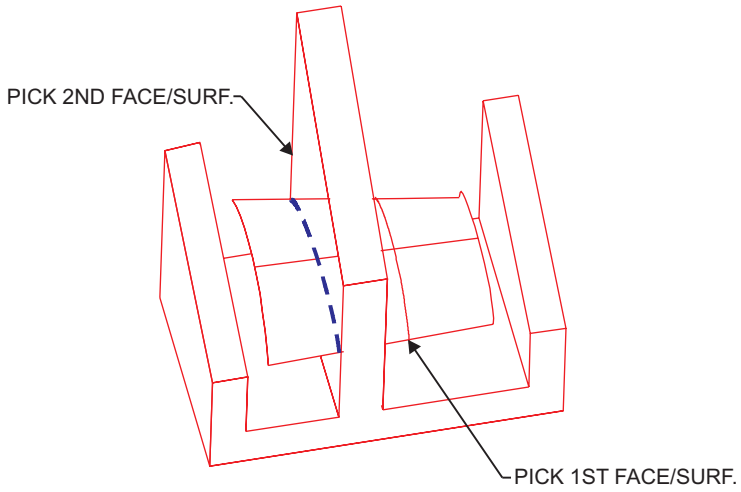


Figure 2-38: CURVE >> SRFSEC

PICK 1ST FACE/SURFACE Pick the first surface / face or plane.

PICK 2ND FACE/SURFACE Pick the second surface / face or plane.

A reference curve will be created.

Note:

- Both of the surfaces / faces cannot be planes.

CURVE >> EDGE CHAIN

Choose an edge or a smooth chain of edges to be a reference curve. As a result, one composite reference curve is created.

SELECT	ALL
	LIMITED
	SINGLE

CURVE >> EDGE CHAIN >> ALL

Create a reference curve from a chain of smooth edges.

PICK EDGE Pick a smooth edge.

CURVE >> EDGE CHAIN >> LIMITED

Create a reference curve from a limited chain of smooth edges.

PICK FIRST EDGE Pick the first edge of the chain.

IND. DIRECTION Choose the direction in which the chain will proceed. The direction will only be shown if the selected edge is a loop.

PICK LAST EDGE Pick the last edge of the chain.

CURVE >> EDGE CHAIN >> SINGLE

Create a reference curve from a single edge.

PICK EDGE Pick the edge that will be used as the reference curve.

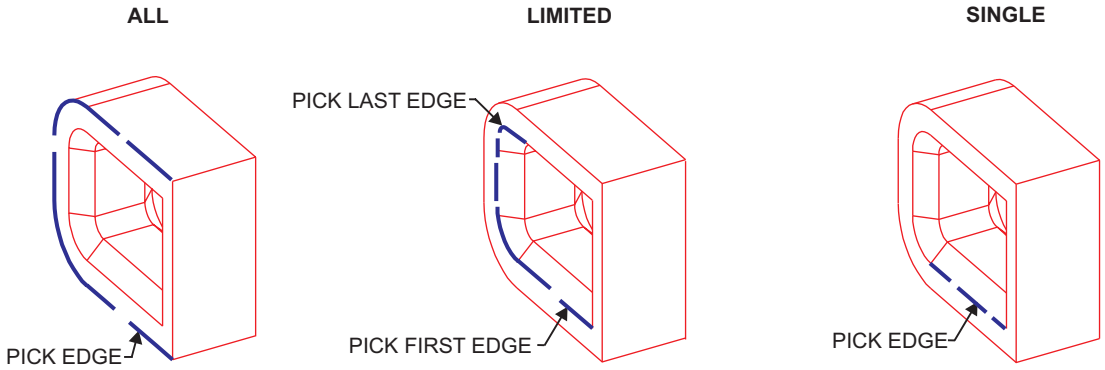


Figure 2-39: CURVE >> EDGE CHAIN

CURVE >> HELIX

Create a reference curve in the form of a helix.

SELECT RADIUS TYPE:	FIXED
	LINEAR
	EXPONENTIAL

CURVE >> HELIX >> FIXED

Create a helix with a fixed radius.

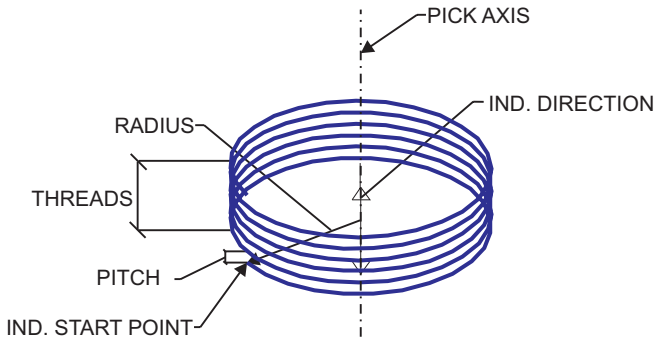


Figure 2-40: CURVE >> HELIX >> FIXED

PICK AXIS	C.C.W / C.W.	THREADS=2.000	PITCH=1.000
	RADIUS=10.000		

PICK AXIS Pick the axis or edge along which the helix will be generated. The cross-section of the helix will be normal to this axis.

- C.C.W / C.W. — Choose whether the helix will proceed in a clockwise or counterclockwise direction.
- THREADS=2.000 — Enter the number of revolutions of the helix.
- PITCH=1.000 — Enter the spacing between revolutions of the helix..
- RADIUS=10.000 — Enter the radius of the helix.

IND. START POINT Choose a point that will indicate the direction normal to the axis where the helix will start. This point cannot be located on the axis itself.

Activate the submenu for the following picking choices:

END
MID
CENTER
KEY-IN
INTERS
PIERCE
PICK

For explanations of the picking choices, see page 2-53.

IND. DIRECTION Choose the direction in which the helix will be generated.

CURVE >> HELIX >> LINEAR

Create a helix with a linearly increasing or decreasing radius.

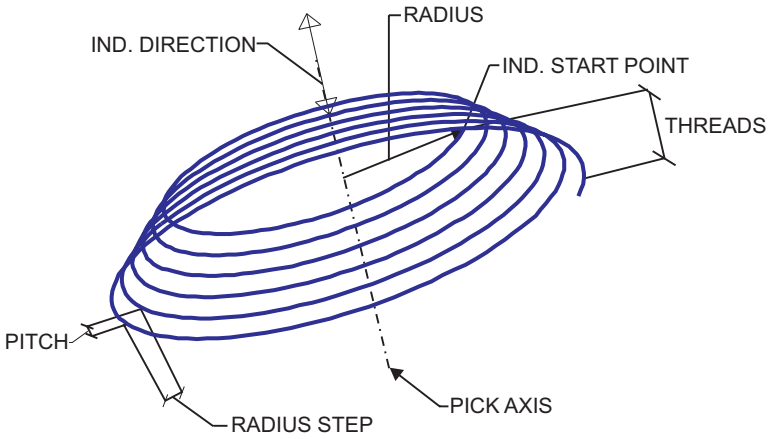


Figure 2-41: CURVE >> HELIX >> LINEAR

PICK AXIS	C.C.W / C.W.	THREADS=2.000	PITCH=1.000
	RADIUS=10.000	RADIUS STEP=1.000	

- RADIUS STEP=1.000 Enter the positive or negative value by which the radius will change per revolution.

This interaction and the remaining modals are identical to those for CURVE >> HELIX >> FIXED.

CURVE >> HELIX >> EXPONENTIAL

Create a helix with an exponentially increasing or decreasing radius.

PICK AXIS	C.C.W / C.W.	THREADS=2.000	PITCH=1.000
	START RADIUS=10.000	END RADIUS=20.000	

- START RADIUS=10.000 Enter the radius at the start of the helix.
- END RADIUS=20.000 Enter the radius at the end of the helix.

This interaction and the remaining modals are identical to those for CURVE >> HELIX >> FIXED.

CURVE >> SPLINE

Create a reference curve in the form of a spline.

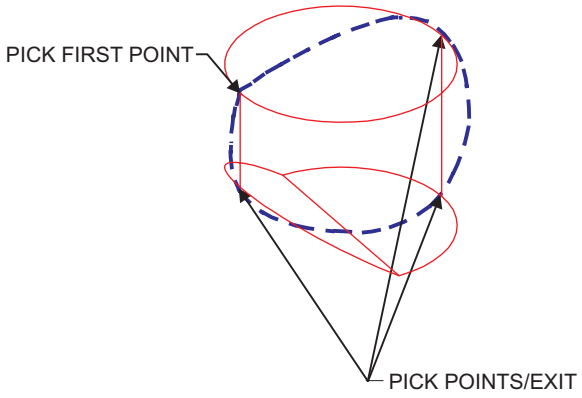


Figure 2-42: CURVE >> SPLINE

PICK POINTS/EXIT

Pick a series of points through which the spline will pass. The first points will be labeled “FIRST”, and the last in the point series will be labeled “LAST.” <EXIT> when finished.

Activate the submenu for the following picking choices:

END
MID
CENTER
KEY-IN
INTERS
PIERCE
PICK

For explanations of the picking choices, see page 2-53.

DEFINE SLOPE/EXIT	FREE SLOPES/ DEFINE SLOPES

DEFINE SLOPE/EXIT

Choose either to define slopes at the ends of the spline or to leave the slopes as they were generated. <EXIT> is equivalent to choosing FREE SLOPES.

- FREE SLOPE — Slopes at the FIRST and LAST point will remain as they are.
- DEFINE SLOPES — Define the slopes at the FIRST and/or LAST spline point. The following prompt will appear:

PICK FIRST/ LAST POINT — Pick the end whose slope you wish to define.

DEFINE DIRECTION	AXIS/EDGE	FLIP	ORIGIN	+ DIRECTION	APPLY

There are two ways to define a slope; steps 2 and 3.

1. Use the slope of an existing edge or axis as a reference slope.
   - *a).* Choose an axis or straight edge which will be used to define the slope. Use the AXIS/EDGE parameter.
   - *b).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *c).* Apply the above slope constraints to modify the spline. Use the APPLY button.
2. Use the slope of a vector defined by two points: an origin and a point indicating the positive direction.
   - *a).* Indicate an origin point. Use the ORIGIN parameter.
   - *b).* Pick a point to indicate the positive direction of the vector. Use the + DIRECTION parameter.
   - *c).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *d).* Apply the above slope constraints to modify the spline. Use the APPLY button.

CURVE >> LINE NRM

Create a reference line normal to a given face, plane, edge, curve, or axis.

SELECT NORMAL:	FROM PNT TO...	AT A PNT ON...

CURVE >> LINE NRM >> FROM PNT TO

Create a line normal to a specified entity, starting from a specified point.

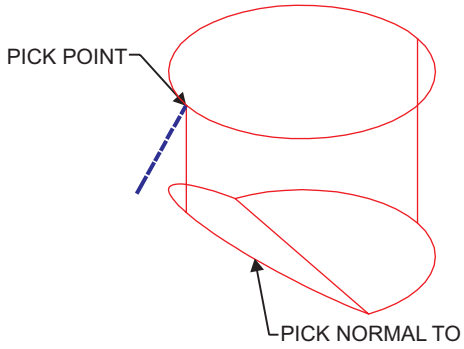


Figure 2-43: CURVE >> LINE NRM >> FROM PNT TO

PICK NORMAL TO

Pick an entity to which the line will be normal. Activate the submenu for the following picking choices:

FACE
PLANE
EDGE
CURVE
AXIS

The default choice is FACE or PLANE.

PICK POINT

Pick the point from which the normal line will start. Activate the submenu for the following picking choices:

END
MID
CENTER
KEY-IN
INTERS
PIERCE
PICK

For explanations of the point picking choices, see page 2-53.

CURVE >> LINE NRM >> AT A PNT ON

Create a line of a specified length normal to a specified entity at a specified point.

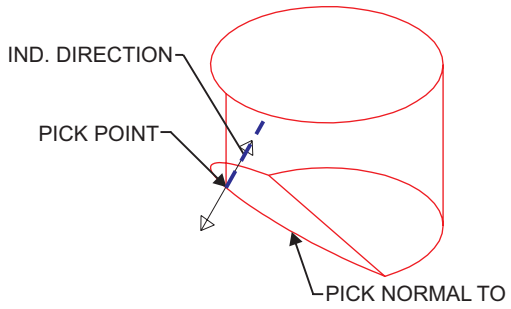


Figure 2-44: CURVE >> LINE NRM >> AT A PNT ON

PICK NORMAL TO	LENGTH=100.000

PICK NORMAL TO Pick an entity to which the line will be normal.

PICK POINT	LENGTH=100.000

PICK POINT Pick the point on the entity from which the normal line will start.

If the selected edge or curve is a straight line, the following prompt appears:

PICK PLANE/FACE Pick a plane/face on which a normal line will be created.

IND. DIRECTION	LENGTH=100.000

IND. DIRECTION Choose the direction of the line.

Notes:

- The length of the line can be entered or changed at any stage of this interaction.
- Picking submenus are identical to those for the previously described option, LINE NRM >> FROM PNT TO.

CURVE >> COM CURVE

Join two or more datum curves to form one composite curve.

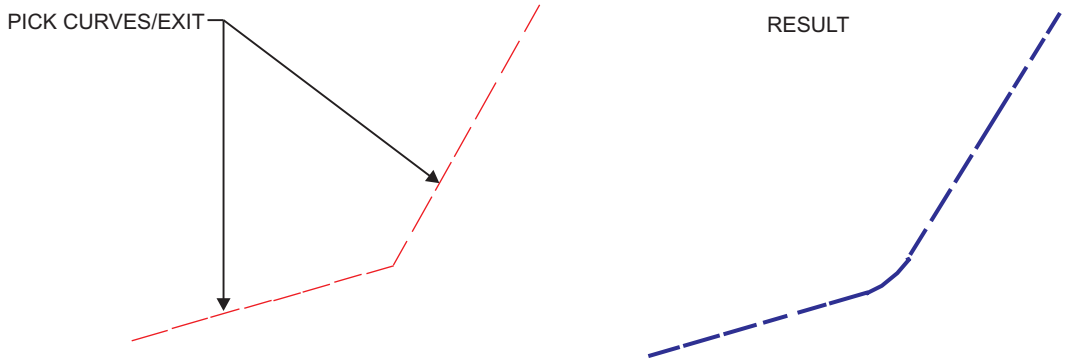


Figure 2-45: CURVE >> COM CURVE

PICK CURVES/EXIT	GAPTOL=0.100	RADIUS=0.100

PICK CURVES/EXIT Pick the datum curves to be joined. <EXIT> when finished.

Note:

- The original curves will be blanked after the composite curve has been created.

DATUM >> POINT

Create datum points.

PICK POINTS/EXIT <PICK> points, <EXIT> when finished. When picked, the points will be represented by an "X". After you <EXIT> the picking session, the points will be represented by a "+".

Activate the submenu for the following picking choices:

END
MID
CENTER
KEY-IN
INTERS
PIERCE
PICK & DELTA
UNPICK

For explanations of the point picking choices (except PICK & DELTA and UNPICK), see page 2-53.

PICK & DELTA Create a point at a specified distance from a reference point. The reference point must have been created in a previous picking session, i.e. the point must appear as a "+".

<CR> TO CONTINUE	DELTA X=0.000	DELTA Y=0.000	DELTA Z=0.000

Enter the X, Y, and Z distances from the reference point. Press <CR> when finished.

UNPICK Unpick a point. This point must have been created in the current picking session, .e., the point must appear as an "X".

DATUM: Modal Parameter Definitions

- + DIRECTION — Pick a point to indicate the positive direction of the vector.
- ANGLE = 15.0 — Enter the offset angle.
- APPLY — Apply the above slope constraints to generate a new spline.
- AXIS/EDGE — Choose an axis or straight edge which will be used to define the slope.
- DELTA — Enter the offset between the axis and the selected edge.
- FLIP — Reverse the indicated direction, if desired.
- GAPTOL=0.100 — Enter the maximum gap tolerance between curves.
- LENGTH=100.000 — Enter the length of the normal line.
- OFFSET = 10.000 — Specify an offset value.
- ORIGIN — Indicate an origin point.
- RADIUS=0.100 — Enter the rounding radius by which the curves will be joined.

DATUM: Usage Envelope

CURVE >> SKETCH

1. In the COMPOSITE CURVE option, the open or closed contour must be created. This means that the end point of the previous curve in the chain coincides with the end point of the following curve.
2. Only one contour can be created during one interaction in the option COMPOSITE CURVE.

❑

DETAIL

Add detail features to solid objects.

Main Options:

SELECT OPTION
ROUND
ROUND EDGE-FACE
CHAMFER
EDGE-DRIVE
DRAFT
REPLACE FACE
DELETE FACE
STITCH

ROUND Fillet selected edge(s) of the object with a fixed or variable radius.

ROUND EDGE-FACE Create a fillet with a fixed or variable radius that starts at an edge and is tangential to a face.

CHAMFER Create a chamfer along selected edge(s).

EDGE-DRIVE Create a protrusion or cut-out by driving a section along a selected loop of edges in an existing feature.

DRAFT Angle selected faces of an object. Faces are angled to facilitate the release of the part from a mold.

REPLACE FACE Replace a face by a non-planar surface.

DELETE FACE Delete the faces of a solid object.

STITCH Merge surfaces or open objects together.

DETAIL >> ROUND

Fillet selected edge(s) of an object with a fixed or variable radius.

How To:

1. Select whether to use a fixed radius or a variable radius.
2. For a fixed radius, enter the value of the radius.
   <PICK> the edge you wish to round: either a single edge or a smooth sequence of edges that includes the selected edge.
3. Choose between an arc or conic edge.
4. For a variable radius, pick the edge(s) you wish to round: either a single edge or a smooth sequence of edges that includes the selected edge(s).
   Select an end of an edge and enter the radius. Repeat for other ends as necessary.
   Press <EXIT> to execute.
5. Enter the radii as prompted.
6. Press <EXIT> to execute.

SELECT	FIXED RADIUS
	VARIABLE RADIUS

FIXED RADIUS Fillet selected edge(s) with a fixed radius.

VARIABLE RADIUS Fillet selected edge(s) with a variable radius.

For a number of the ROUND options, when asked to pick an edge the submenu gives the following options:

SET PICK MODE	SINGLE	CHAIN

SET PICK MODE Set the mode for <PICK>.

- SINGLE — Fillet each edge separately.
- CHAIN — Fillet a smooth sequence of edges that includes the selected edge.

Note: • The default is CHAIN.

ROUND >> FIXED RADIUS

Fillet selected edge(s) with a fixed radius.

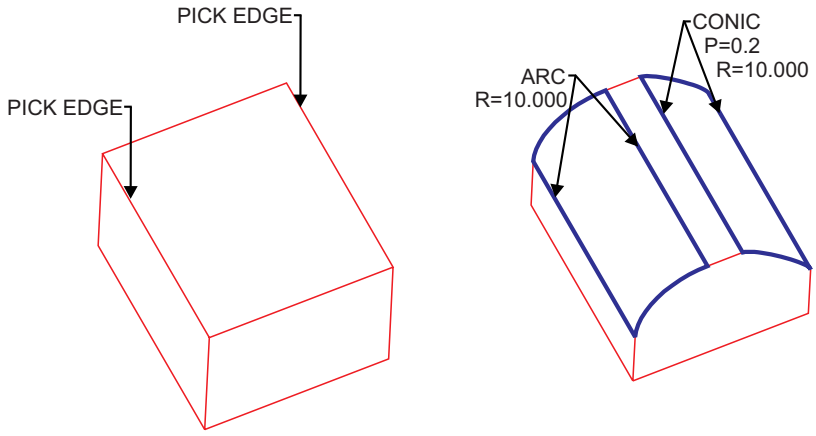


Figure 2-46: DETAIL >> ROUND >> FIXED RADIUS

Interaction:

PICK EDGE	RADIUS =10.000	ARC / CONIC	P=1.000

PICK EDGE <PICK> an edge to round.

The submenu displays the SINGLE and CHAIN options. See page 2-66.

The following interaction will occur in a situation involving a corner where more than three edges meet.

A gap will be created in the model when four or more edges meet and at least three of them are not smooth. When more than three edges enter into a corner, the program will leave a gap in the body until all the non-tangential edges are rounded. If the program can close the solid, the following interaction will occur.

The system highlights the gap. When this occurs, the following prompt appears:

SELECT	CORNER BLEND	SINGLE EDGE

SELECT Select one of the following options:

- CORNER BLEND — The corner appears in white outline indicating that it is open, and all edges in contact with this corner must be rounded by the user. At the end the system will create a blend between all the rounds.
- SINGLE EDGE — The single edge will merge with the gap with one of it's neighbor's faces.

ROUND >> VARIABLE RADIUS

Fillet selected edge(s) with a variable radius.

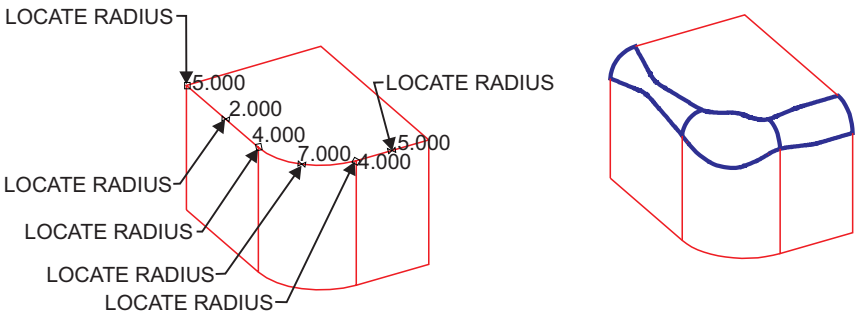


Figure 2-47: DETAIL >> ROUND >> VARIABLE RADIUS

How To:

1. Choose the kind of variable round fillet edge you want to create; tangent arc or conic curve.

   For an arc, decide whether you want linear or non-linear changes of the radius value between every pair of given points.

   For a conic curve, enter the value of the P parameter which defines the relative weight of the mid-point through which the conic is created.

2. Pick the edge to round.

3. When the LOCATE RADIUS prompt is displayed, you may enter variable radius values at each end point and at the midpoint of the edge.

Interaction:

PICK EDGE	ARC	LINEAR
		NON LINEAR
	CONIC	P = 1.000

PICK EDGE

<PICK> an edge to round.

The submenu displays the SINGLE and CHAIN options. See page 2-66.

LOCATE RADIUS	RADIUS = 10.000

You may enter variable radius values at each end point and at the midpoint of the edge.

Press <EXIT> when finished.

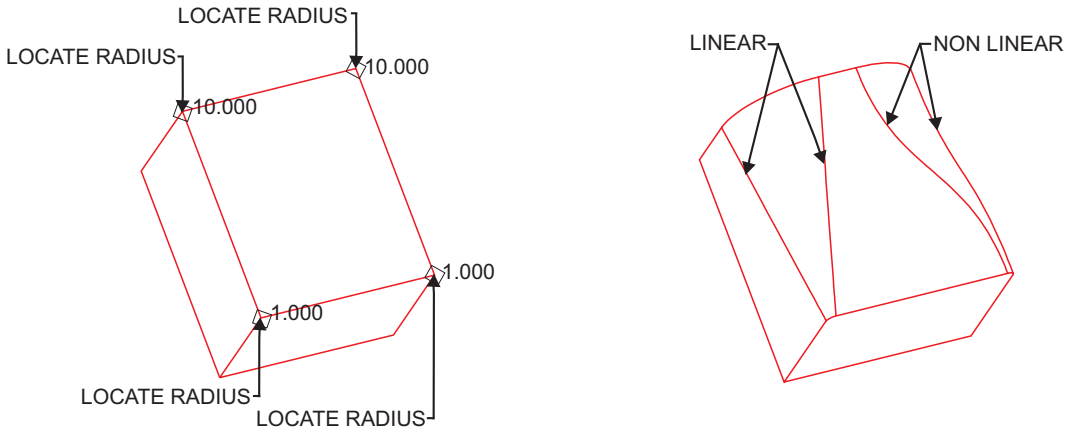


Figure 2-48: DETAIL >> ROUND >> VARIABLE RADIUS (LINEAR, NON-LINEAR)

DETAIL >> ROUND EDGE-FACE

Creates a fillet with a fixed or variable radius that starts at an edge and is tangential to a face.

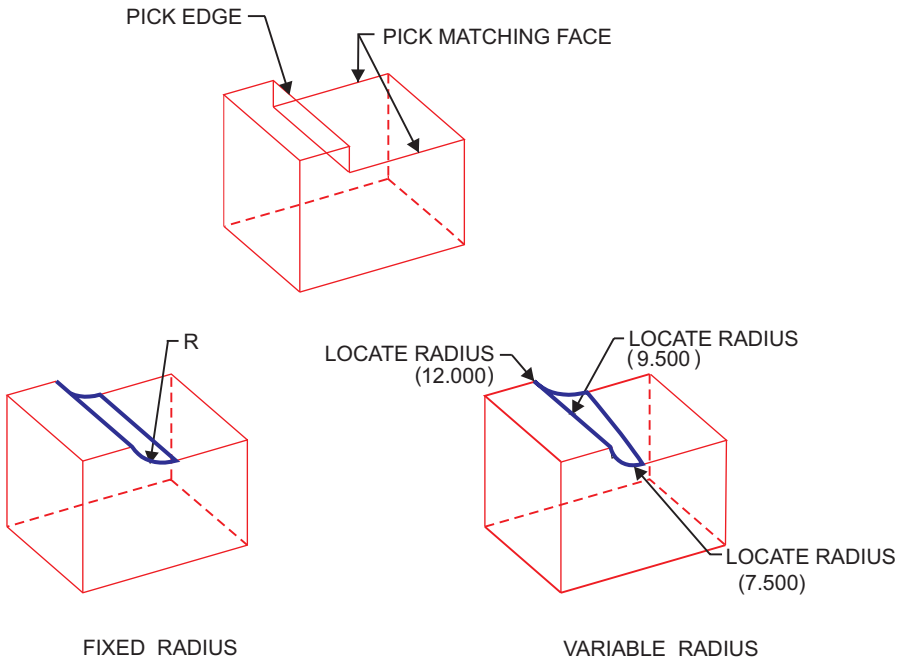


Figure 2-49: DETAIL >> ROUND EDGE-FACE (FIXED, VARIABLE RADIUS)

How To:

1. Pick type of fillet (fixed or variable) and set its value.
2. Pick edge.
3. Pick face.

Interaction:

SELECT FUNCTION	FIXED RADIUS
	VARIABLE RADIUS

FIXED RADIUS Fillet selected edge(s) with a fixed radius.

VARIABLE RADIUS Fillet selected edge(s) with a variable radius.

ROUND EDGE-FACE >> FIXED RADIUS

Fillet selected edge(s) with a fixed radius (see Figure 2-49).

PICK EDGE	RADIUS = 10.000

Set the radius and pick the edge

PICK MATCHING FACE <PICK> the face to which the fillet will be tangent.

Note: • This option allows a fillet to be constructed when the radius is too large to allow an arc to be tangent to both faces.

ROUND EDGE-FACE >> VARIABLE RADIUS

Fillet selected edge(s) with a variable radius (see Figure 2-49).

PICK EDGE Pick the edge to be filleted.

PICK MATCHING FACE <PICK> the face to which the fillet will be tangent.

LOCATE RADIUS	RADIUS=10. 000

Enter variable radius values at end points and at midpoints of the edge.

Press <EXIT> when finished.

DETAIL >> CHAMFER

Create a chamfer along selected edge(s). If the edges are straight edges the chamfer produced will be at 45° along the selected edge.

PICK EDGE	D = 10.00

■ D = 10.00 Enter the chamfer size.

PICK EDGE <PICK> an edge to be chamfered.

The submenu gives the following options when picking an edge:

SET PICK MODE	SINGLE	CHAIN

SINGLE Chamfer each edge separately.

CHAIN Chamfer a smooth sequence of edges that includes the selected edge (the default pick mode).

DETAIL >> EDGE DRIVE

Create a protrusion or cut-out by driving a section along an edge or a selected loop of edges.

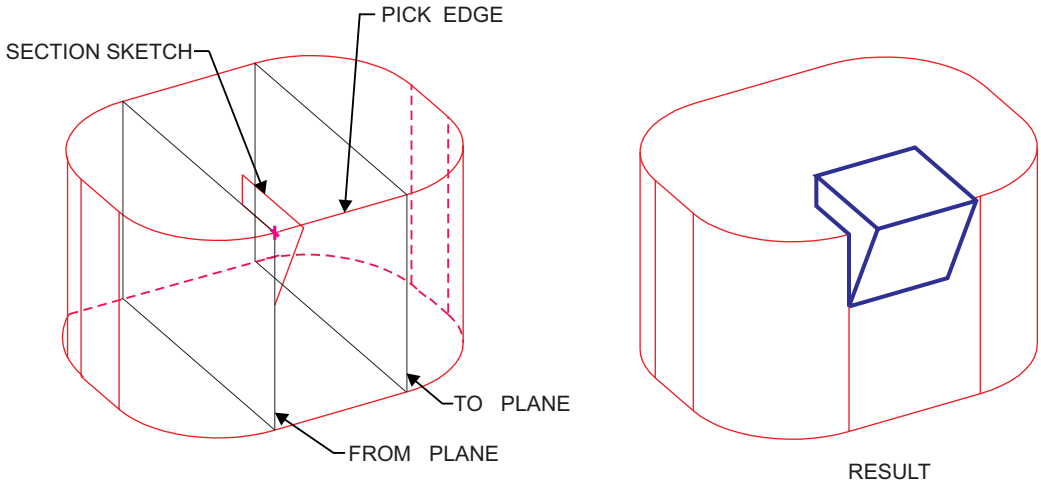


Figure 2-50: DETAIL >> EDGE DRIVE

Interaction:

SELECT	ADD
	REMOVE
	DIVIDE
	DATUM
	NEW

ADD Add a protrusion or cut-out.

REMOVE Remove a protrusion or cut-out.

DIVIDE Divide a solid object into two new objects.

DATUM Create a datum based on a 2D shape.

NEW Begin a new object.

EDGE-DRIVE >> ADD

Add a protrusion or cut-out.

PICK ACTIVE OBJECT Pick the object to which you want to add.

PICK CHAIN <PICK> a smooth chain of edges which will be used as a trajectory.

or: Press <SUBMENU> to select the trajectory type.

SET TRAJECTORY TYPE	EDGE	LOOP	CHAIN	CURVE

- EDGE — Drive a closed section along an edge.

  PICK EDGE & REF FACE Pick the edge which will be used as a trajectory, and the face to which the trajectory will refer.

- LOOP — Drive a section along a closed loop of edges belonging to one SOLID feature. This loop can be 3 dimensional.

  PICK LOOP Pick the loop which will be used as a trajectory.

- CHAIN — Drive a closed section along a chain of smooth curves.

  PICK CHAIN Pick the chain of curves which will be used as a trajectory.

- CURVE — Drive along a reference curve.

  You will enter the **Sketcher**.

  The cross that appears on the screen represents the sketching plane and the pierce point of the trajectory.

  The section contour will be driven normal to the trajectory.

  Sketch and dimension the section contour with respect to the cross center. Then select <EXIT>.

EDGE-DRIVE >> REMOVE

Remove a protrusion or cut-out

See EDGE-DRIVE >> ADD on page 2-79 for details of the interaction.

EDGE-DRIVE >> DIVIDE

Divide a solid object into two new objects.

See EDGE-DRIVE >> ADD on page 2-79 for details of the interaction.

EDGE-DRIVE >> DATUM

Create a datum based on a 2D shape.

See EDGE-DRIVE >> ADD on page 2-79 for details of the interaction.

EDGE-DRIVE >> NEW

Begin a new object.

See EDGE-DRIVE >> ADD on page 2-79 for details of the interaction.

DETAIL >> DRAFT

Angle selected faces of an object. Faces are angled to facilitate the release of the part from a mold.

Note: • The maximum draft angle allowed is 45°.

PICK REF. PLANE/FACE	ANGLE = 10.000	FACE / LOOP	SPLIT / NO SPLIT

PICK REF. PLANE <PICK> the reference plane or face for the angle.

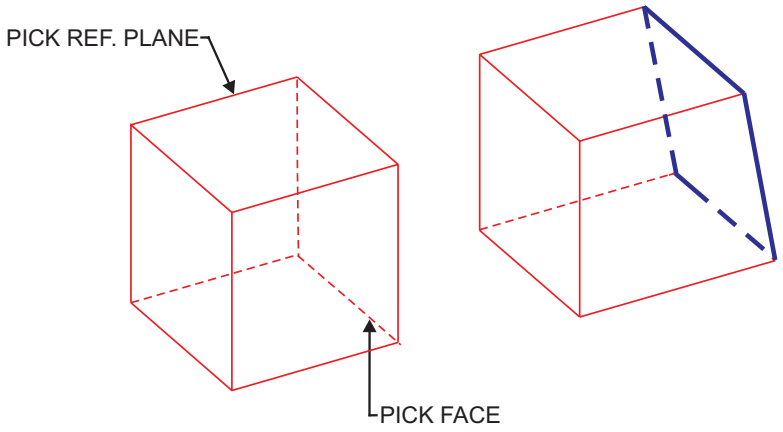


Figure 2-51: DETAIL >> DRAFT (ANGLE = 10.000)

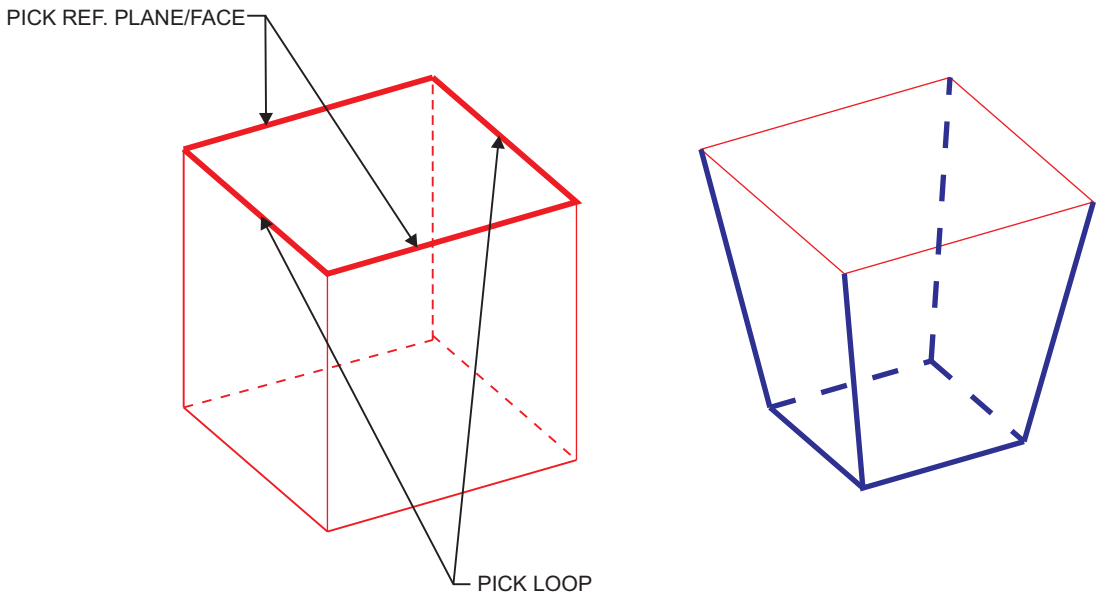


Figure 2-52: DETAIL >> DRAFT (LOOP)

If SPLIT is chosen, the following prompt appears:

<CR> TO CONTINUE	ONE SIDE / BOTH SIDES	FLIP SIDE

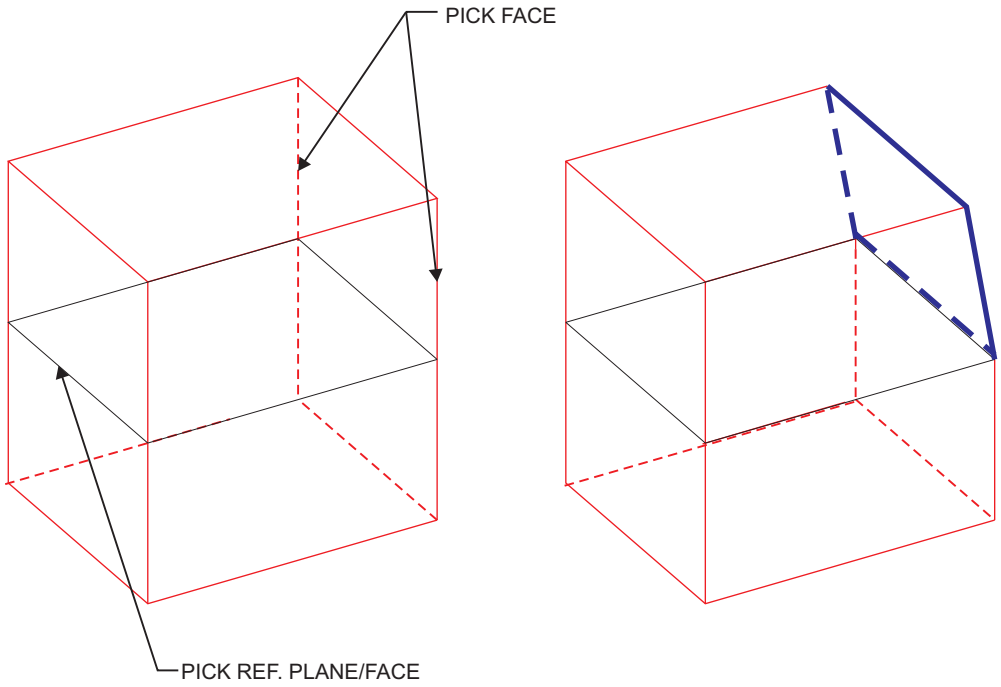


Figure 2-53: Draft Split

DETAIL >> REPLACE FACE

Replace a face by a non-planar surface.

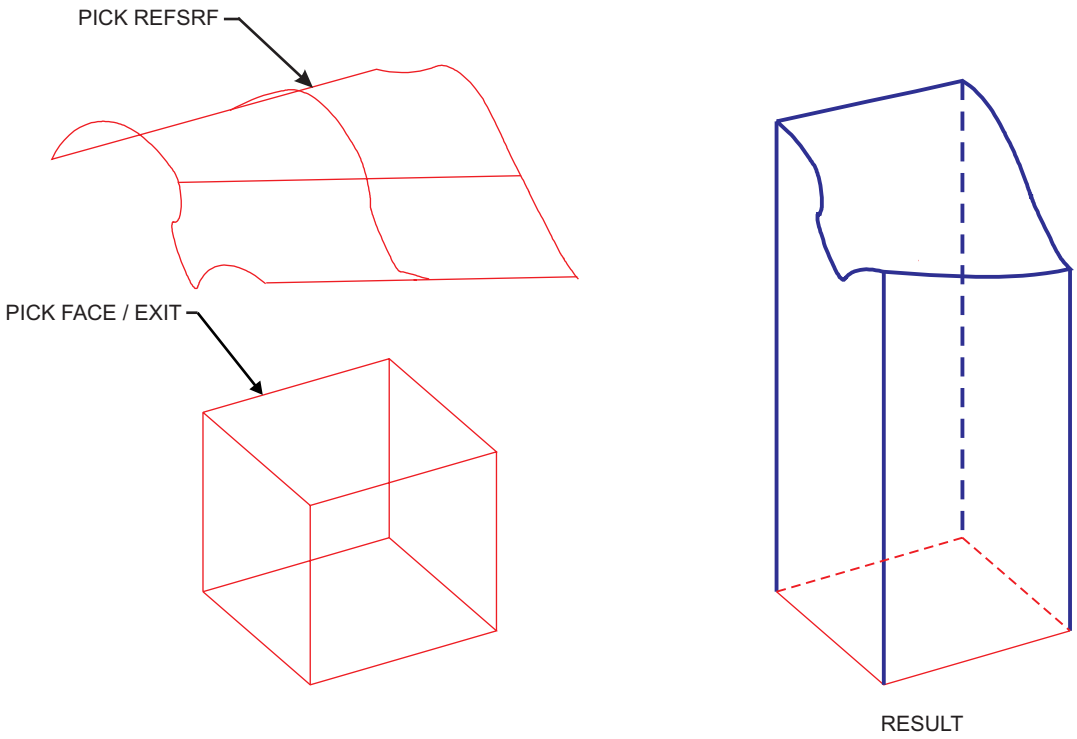


Figure 2-54: DETAIL >> REPLACE FACE

Interaction:

<PICK> FACES & EXIT Pick the faces that will be replaced by a surface and press <EXIT>.

Note: • In the case of several faces, there must be no common edge.

<PICK> SURFACE Pick the reference surface that will replace the face(s).

The system will replace the faces. If the surface is lower or higher than the face(s), the system will extend the boundary faces of the replaced face(s). See Figure 2-54.

Note: • The result of the exchange must not change the solid topology.

DETAIL >> DELETE FACE

Delete a face(s) of a solid object.

PICK FACES/EXIT Pick the face(s) to be deleted and <EXIT>.

DETAIL >> STITCH

Merge surfaces or open objects together.

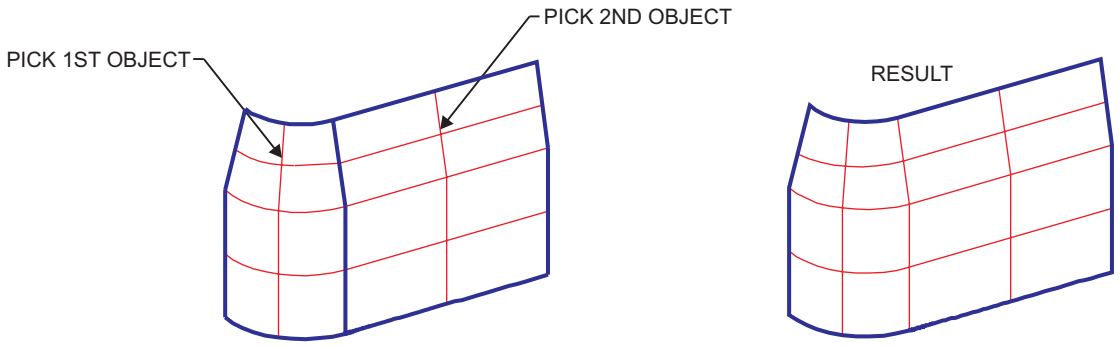


Figure 2-55: DETAIL >> STITCH

How To:

1. Pick the first surface (object).
2. Pick other surfaces (objects) to be stitched to the initial surface (object).

   Pick <EXIT> when finished.

Interaction:

PICK 1ST OBJECT	Pick one of the stitched surfaces (objects).
PICK ENTITIES & EXIT	Pick surfaces (objects) to stitch with the first object. Multipick options are available.

DETAIL: Modal Parameter Definitions

- ANGLE = 10.000 — Enter a positive or negative draft angle. A positive value increases the angle between the selected face and the reference plane.
- ARC — A tangent arc will be created between the faces.
- BOTH SIDES — Two draft faces will be created, originating from the reference plane/face.
- CONIC — A conic curve will be created between the faces.
- FACE — Angle a selected face(s) with the specified angle.

  PICK FACE/EXIT — <PICK> the faces to be angled and press <EXIT>.
- FLIP SIDE — Choose the other direction as indicated by the arrow on screen.
- LINEAR — Linear changes of the radius value between every pair of given points.
- LOOP — Angle all faces in a closed loop with the specified angle.

  PICK LOOP — <PICK> an edge of a closed loop.
- NO SPLIT — Draft faces begin from picked face/plane.
- NON LINEAR — Non-linear changes of the radius value between every pair of given points.
- ONE SIDE — One draft face will be created, originating from the reference plane/face.
- P = — The parameter ‘P’ controls the relative weight of the mid point through which the arc is created. As P is larger the mid point becomes more dominant and the curve blends into a corner. As P is smaller the mid point becomes less dominant and the curve blends into a chamfer. When P=1, the result is the same as when choosing an arc.

Note

- The value of P cannot be edited in the EDIT >> PARAMETERS mode.

- RADIUS = 10.000 — Enter the radius of curvature.

  Locate the cursor on the end or middle of the edge. The system will create a diamond symbol at both ends and at the midpoint. Pick the points where the radius will be defined. The radius may be changed at each point.
- SPLIT — Create draft faces from a reference plane that intersects the faces.

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EDIT

Modify parameters, change the dimensioning scheme, set relations between dimensions, replay work steps, delete features, or suppress them.

Main Options

PARAMETERS
SKETCH
RELATION
DELETE
SUPPRESS
RENAME
UPDATE
REPLAY
TRIM

PARAMETERS Modify dimension values and update the features.

SKETCH Change the dimensioning scheme, or add dimensions to a feature that was created without dimensions, or redesign the sketch.

RELATION Define a mathematical relationship between dimensions or enter a value for a dimension.

DELETE Delete a feature.

SUPPRESS Temporarily remove features from the regeneration list.

RENAME Change the feature's name.

UPDATE Update imported objects.

REPLAY Restore the previous/next step and insert steps.

TRIM Appears only in INSERT mode. Trims off all consecutive additions to model and exits the INSERT mode.

Note:

- Picking objects can be done by pointing at them or with the help of the feature listing accessible through the submenu.

Failure Manager

Every time **Cimatron** is unable to regenerate a feature, an error message appears. On the upper part of the screen the failure manager appears:

SELECT OPTION	GO	STEP	FREEZE	DELETE	CANCEL

GO The program will jump to the last step.

STEP The program will go to the next step.

FREEZE Keeps the feature frozen.

DELETE Deletes feature.

CANCEL Cancels the operation that caused the failure.

Note:

- When the Failure Manager is activated, the colors of the model will change according to the following system:
  - White - Model before modification
  - Red - Feature that caused failure
  - Blue - Child of the failed feature
  - Yellow - Modified model as it looked until the failed feature

EDIT >> PARAMETERS

Modify dimensions and update the features.

SELECT FEATURE/EXIT <PICK> the feature whose dimensions you wish to modify.

The system will display all relevant dimensions of the feature.

The dimension will appear in green and can be dragged to different positions.

SELECT FEATURE/EXIT	VALUE=10.00

SELECT FEATURE/EXIT Enter a dimension value for the specific dimension and Modify another dimension or press <EXIT>.

<PICK> another feature or press <EXIT> to terminate the function.

The feature is updated.

Press <EXIT> to continue. After pressing <EXIT>, the color of the dimension will be modified to white.

Notes:

- You cannot modify a dimension that is the result of an expression (as defined using the option RELATION). If you try to do this, you will receive an error message.
- When modifying round features that were created in one step, the following menu appears:

IND DIM POS/EXIT	VALUE=10.00	ALL / SINGLE

- When modifying a copied feature, the following menu appears:

<CR> TO CONTINUE	PREV	NEXT

PREV / NEXT — Allows the user to pick every step of the copy hierarchy, including parent features.

Notes:

- Features to be modified can be selected by either of the following two ways:
  - <PICK> the feature directly.
  - Activate the submenu. A list of all features appears; select the desired feature.
- Features may be temporarily invisible, due to one of the following reasons:
  - The feature was frozen when the Failure Manager was activated.
  - The feature was suppressed by the EDIT >> SUPPRESS option.
  - You are in INSERT mode.

EDIT >> SKETCH

Edit a sketch.

SELECT FEATURE/EXIT — <PICK> the feature whose sketch you wish to modify.

You will enter the **Sketcher**. The DIMENS module is activated as a default, but picking it will allow you to access all options within the Sketcher. **When** all modifications have been made, press <EXIT>.

This option is particularly useful for changing dimensions, or adding dimensions to sketch components that were insufficiently dimensioned, or for sketch redesign.

PICK FEATURE — <PICK> another feature, or press <EXIT> to terminate the function.

EXECUTING — The feature is updated.

Note:

- Splines cannot be edited in this fashion.
- All the constraints that are not affected by modifications will be kept.
- Dimensions might have to be deleted to allow for modified geometry.
- All the dimensions affected by modifications will be automatically updated, where possible.

EDIT >> RELATION

Define a mathematical relationship between dimensions, or assign a value to a dimension.

SELECT FEATURE / EXIT <PICK> the features whose dimensions will be mathematically related.

The dimensions are displayed with the system names automatically assigned.

Lines are assigned a number with an L prefix, e.g. L30. Radii are assigned an R prefix, and angles are assigned an A prefix.

PICK DIM FOR UPDATE <PICK> a specific dimension to be mathematically related.

IND DIM POS/EXIT	NAME=L39	L39 =
	VALUE =30.000	DESC=

A dimension may be changed either by assigning it an expression which relates it to another dimension, or by entering a value.

Press <EXIT> once to pick additional features.

Press <EXIT> twice to finish the relation setting and to regenerate the part.

Notes:

- Enter all expression descriptions in upper case.
- Assume the standard precedence of operators when writing the mathematical relation. If needed, add parentheses for clarity.
- If the relationship between the two dimensions is linear, for example L2=L3*2+5, the relationship is defined as “bi-directional”. In a bi-directional relationship, you can change either dimension, and the other will be updated.

  If the relationship is not one-to-one, then the relationship is only in one direction, for example L2=L3+L4, and you cannot modify L2.

EDIT >> DELETE

Delete a single feature.

SELECT FEATURE <PICK> a feature to be deleted.

FEATURE OK? YES NO

YES Feature is deleted.

NO <PICK> another feature.

If the feature has children, the children will be highlighted in blue, and the following prompt will appear:

DELETE CHILDREN?	YES	NO

YES Feature is deleted.

NO Cancel operation.

If the feature's dimensions are related to those of other features, the following warning message will appear:

RELATIONS WILL BE DISCONNECTED.

The following prompt will ask:

CONTINUE	YES	NO

YES The relation is disconnected.

NO The DELETE function is aborted.

Notes:

- Features that are deleted in this fashion cannot be retrieved with the UNDO function. The only way to undo the result of the delete action is to exit the file without saving it and reentering it.
- The system will try to break the father-child relationship if possible. In that case, the rerouted features will be highlighted in red and you restrain them using EDIT >> SKETCH function.

EDIT >> SUPPRESS

Temporarily remove features from the features list.

SELECT OPTION	SUPPRESS	UNSUPPRESS

SUPPRESS >> SUPPRESS

This feature allows you to suppress features using two different methods.

SELECT / EXIT	PICK	TYPE

SUPPRESS >> UNSUPPRESS

This feature allows you to display suppressed features.

SELECT / EXIT	ALL	LAST	TYPE	FROM LIST

EDIT >> RENAME

Change the name of a feature.

ENTER NEW NAME	NAME = EXTRUDE 7	SET DEFAULT

NAME The new name of the feature.

SET DEFAULT Restores the name of the feature given by the system. The default feature names are given by the system and include prefixes and serial numbers which indicate the way they were created. For example, EXTRUDE7 designates the seventh feature created, and it was created by using the extrude feature.

EDIT >> UPDATE

Update imported objects.

The full path name of the exported object is displayed to the file owner. If, during updating, the reference file cannot be found, the system displays the message "File <filename> does not exist" and asks if you want to update the full path name.

EDIT >> REPLAY

Restore previous/next steps and insert steps.

SELECT OR EXIT	|<—	<—	—>	—>|	JUMP

Notes:

- When entering the REPLAY function, the number of steps used to create the model and the step currently shown, are given in the upper right hand corner.
- Jumping back to the creating stage of any feature will start the INSERT mode. Any change to the feature will be incorporated into the model. If the changes cause conflicts in the model, an error message will be given and the changes will be ignored.
- Saving the file in INSERT mode will automatically replay the model.

EDIT >> TRIM

Trims off all consecutive additions to model and exits the INSERT mode. This function appears only in the insert mode. The steps of model creation are deleted from the step this function was chosen in the INSERT mode

Note

- If a save is executed after this action, the information deleted cannot be retrieved.

EDIT: Modal Parameter Definitions

- |<— Restore to start.
- <— Restore previous step.
- —> Restore next step.
- —>| Restore to last step.
- ALL Modify dimensions of all round features.

  SINGLE Modify dimensions of only one feature.
- ALL Unsuppress all the suppressed features.

  UNSUPPRESS ALL? YES / NO

  YES Unsuppress all.

  NO Return without unsuppressing.
- DESC= Enter a textual description.
- FROM LIST Pick features to be unsuppressed from the feature list.
- JUMP Jump to the step that the picked object was created in.
- L39= Enter an expression that defines the dimension. For example, for dimension L39, enter L36*2, which means that L39 is twice the length of L36.
- LAST Unsuppress only the feature that was last suppressed.
- NAME=L39 The name of the selected dimension, for example, L39.
- PICK Pick the feature to be suppressed.
- TYPE Choose the feature to be suppressed by type./Unsuppress features according to their type.

  The following options will appear:

ROUND	CHAMFER	HOLE	SHAFT	SHELL

  Select the type of feature.
- VALUE = 10.00 Enter a dimension value for the specific dimension and press <CR> to confirm, then press <CR> again. The dimension is displayed on the screen, but the feature is not updated.

❑

GROUP

Create, place, explode and export sets of parametric features.

How To:

1. To create a group, pick a series of features to be included in a group or pick an existing group.
2. To place a group, pick the target face or plane and place the group.
3. To explode a group, choose explode and then pick a group.
4. To export objects to another file, enter the target file name, select the entities to be exported and select the UCS. If the target file already exists, decide whether to overwrite the existing file or to export to a different file.
5. Press <EXIT> to execute.

Interaction:

Main Options:

CREATE
PLACE
EXPLODE
EXPORT

CREATE Create a group of features.

PLACE Place a group.

EXPLODE Explode a group into its components.

EXPORT Export objects to another file.

Notes:

- The group function is not applicable to open contour features.
- The group base must be sketch based, i.e. it cannot be a SHELL feature.

GROUP >> CREATE

Create a group of features. A PFM file will be created which can be opened in **Cimatron** and viewed, but not edited.

PICK FEATURE Pick the feature that will be the basis for the group. All features created from this feature (i.e., EXTRUDE >> ADD) will be included in the group.

PICK DIMENSION/EXIT Pick the dimensions that will be editable. The parametric dimensions are marked green, other dimensions (fixed dimensions) are colored white The fixed dimensions cannot be edited by the EDIT >> PARAMETERS option. Press <SUBMENU> to choose all dimensions to be editable.

ENTER FILE NAME Assign a filename to the group.

GROUP >> PLACE

Place a group

SELECT MASTER	EXTERNAL	<Group filename 1>	<Group filename 2>	<Group filename n>

Select a group by filename, or choose EXTERNAL to import a group. If no groups are currently in use in the model, EXTERNAL is the default. You will see the following prompt:

ENTER FILENAME Enter filename of the group to be inserted.

IND. LOCATION/EXIT Indicate the general positioning of the group. When you are satisfied with the general location of the group, press <EXIT>.

PICK SKETCH PLANE Pick the face or plane on which to locate the group. You are now in the **Sketcher**.

PLACE SKETCH	DISPLAY	PLACE				EXTERN		

PLACE SKETCH Place the sketch in the desired location. (For explanations of the table options above, see **The Sketcher Functions** on page 1-6.

To move and/or rotate the sketch, see **Moving the Sketch** on page 1-14.

Notes:

- Using the MIRROR option when placing the sketch will cause the group to be mirrored relative to the face/place on which it is located.
- If the group was placed once in the model, it cannot be called in again by EXTERNAL. To place the same group more than once, it must be picked from the "SELECT MASTER" menu that appears when PLACE is invoked.

GROUP >> EXPLODE

Exploding the GROUP allows editing those dimensions that were not fixed. When a GROUP is exploded, it ceases to be a single object, and its individual parts can be modified. For example, if the GROUP consisted of several features, each feature can be deleted. When a group is exploded, the feature list (when in the EDIT mode) is updated, and the features appear in the list as if they were created in the current model.

GROUP >> EXPORT

Export objects from one file to another.

How To:

1. Enter the target file name.
2. Select the entities to be exported.
3. Select the UCS.
4. If the target file already exists, decide whether to overwrite the existing file or to export to a different file.

ENTER FILE NAME — Enter the name of the target file (the file to which you will be exporting objects).

If the target file already exists, decide whether to overwrite the existing file or to export to a different file.

PICK ENTITIES & EXIT — Select the entities to be exported.

IND. ORIGIN/ EXIT	FLIP

Select the UCS origin point and define the positive X and Y directions.

POINTS OK Y/N

Y Accept the UCS points.

N Re-select the UCS origin point.

EXECUTING — The objects are exported to the newly created file.

GROUP: Modal Parameter Definitions

■ FLIP — Flip the Y and Z axes 180 degrees around the X axis.

❑

MODIFY

Modify parametric surfaces.

Main Options:

SLOPE
FAIR
SCALE
INVERT
BREAK EDGE
SPLIT
ADJOIN FACE

SLOPE Redefine the slopes of surfaces.

FAIR Fair surfaces: reduce waviness, eliminate breakpoints.

SCALE Change scale factors of surfaces or solid objects.

INVERT Display and change the direction of the external normal for an open object.

BREAK EDGE Break a solid edge into 2 edges.

SPLIT Split faces by a plane, surface, or a silhouette line, for creating additional edges to create parting line(s).

Note: • A parting line is based on edges. If there are not enough edges in the model, use the SPLIT option to define additional edges and then define the parting line using SEPARATE >> PARTING LINE.

ADJOIN FACE Close gaps in open solid objects to adjoin faces.

MODIFY >> SLOPE

Redefine the slopes of surfaces.

PICK FACE/SURFACE Pick the surface for which slopes will be redefined.

PICK EDGE Pick the edge for slope to be redefined.

Choose one of the following options for defining the slope:

CONSTANT
LINEAR
KEEP NORMAL
KEEP SLOPES

SLOPE >> CONSTANT

Define a constant slope. Define the slope at one end of the curve/edge.

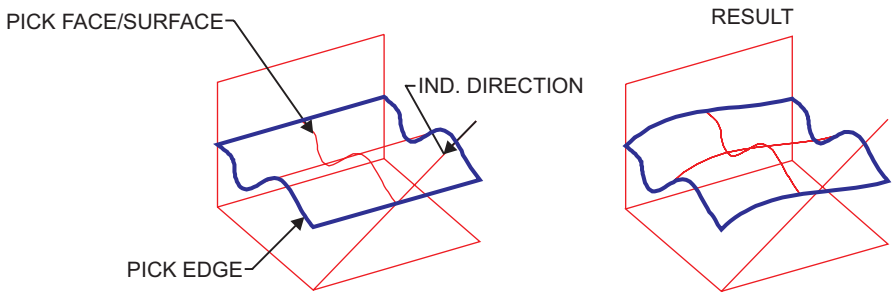

Figure 2-56: SLOPE >> CONSTANT

How To:

There are two ways to define a slope.

1. Use the slope of an existing edge or axis as a reference slope.
   - *a).* Choose an axis or straight edge which will be used to define the slope. Use the AXIS/EDGE parameter.
   - *b).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *c).* Apply the above slope constraints to the picked curve / edge. Use the APPLY button
2. Use the slope of a vector defined by two points: an origin and a point indicating the positive direction.
   - *a).* Indicate an origin point. Use the ORIGIN parameter.
   - *b).* Pick a point to indicate the positive direction of the vector. Use the + DIRECTION parameter.
   - *c).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *d).* Apply the above slope constraints to the picked curve / edge. Use the APPLY button

Interaction:

DEFINE DIRECTION	AXIS/EDGE	FLIP	ORIGIN	+ DIRECTION	APPLY

DEFINE DIRECTION Define the slope at one end of the curve/edge.

There are two ways to define a slope (see How To, above).

SLOPE >> LINEAR

Define a linearly changing slope. Define the slopes at both ends of the curve/edge.

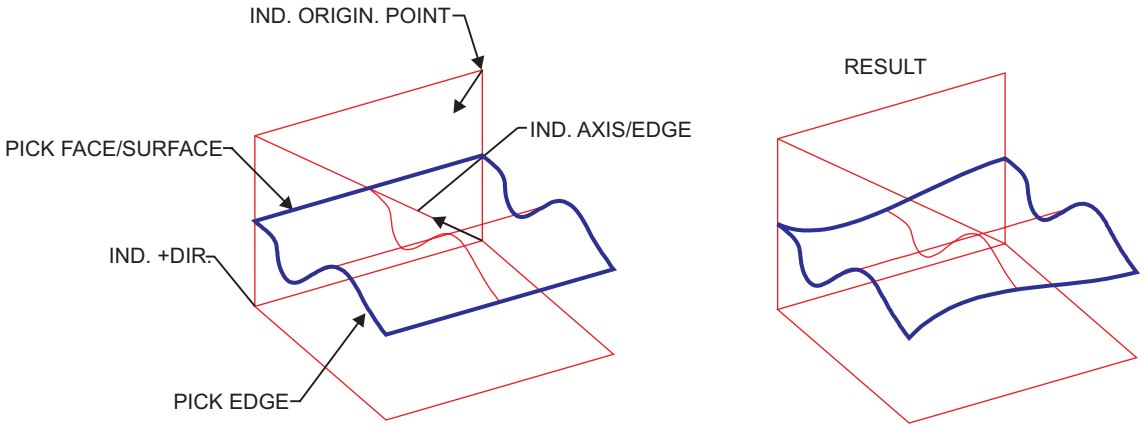

Figure 2-57: SLOPE >> LINEAR

How To:

There are two ways to define a slope.

1. Use the slope of an existing edge or axis as a reference slope.
   - *a).* Choose an axis or straight edge which will be used to define the slope. Use the AXIS/EDGE parameter.
   - *b).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *c).* Apply the above slope constraints to the picked curve / edge. Use the APPLY button
2. Use the slope of a vector defined by two points: an origin and a point indicating the positive direction.
   - *a).* Indicate an origin point. Use the ORIGIN parameter.
   - *b).* Pick a point to indicate the positive direction of the vector. Use the + DIRECTION parameter.
   - *c).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *d).* Apply the above slope constraints to the picked curve / edge. Use the APPLY button

Interaction:

DEFINE 1ST DIRECTION	AXIS/EDGE	FLIP	ORIGIN	+ DIRECTION	APPLY

DEFINE 1ST DIRECTION Define the slopes at both ends of the curve/edge.
DEFINE 2ND DIRECTION

There are two ways to define a slope (see How To, above).

SLOPE >> KEEP NORMAL

Define the slope such that all normals are collinear to the normals on the edge of a reference plane/face.

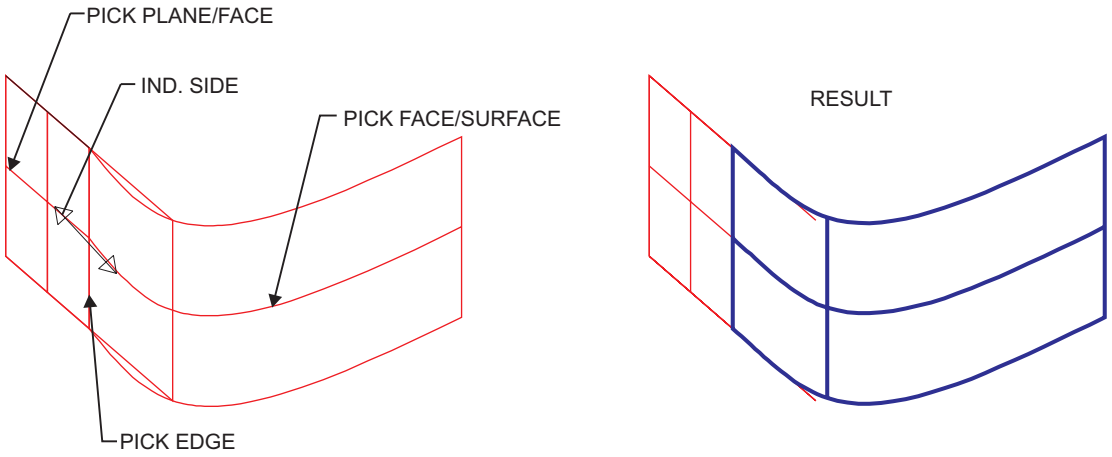

Figure 2-58: SLOPE >> KEEP NORMAL

How To:

1. Pick the reference plane or face.
2. Indicate the direction of the slope.

Interaction:

PICK PLANE/FACE Pick the reference plane or face.

INDICATE DIRECTION Indicate the direction of the slope.

SLOPE >> KEEP SLOPES

Define the slope such that it is tangent to the slope on the edge of a reference plane (face).

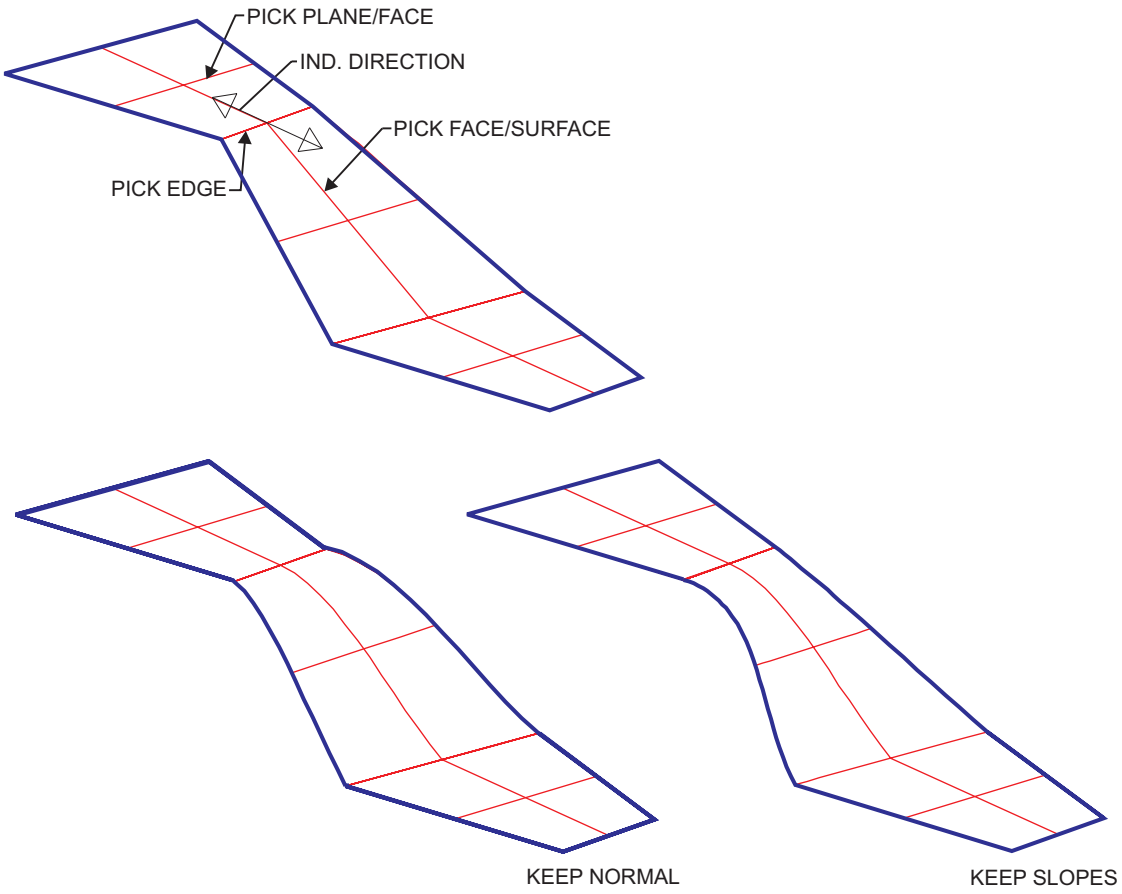

Figure 2-59: SLOPE >> KEEP NORMAL and KEEP SLOPES

How To:

1. Pick the reference plane or face.
2. Indicate the direction of the slope.

Interaction:

PICK PLANE/FACE	Pick the reference plane or face.
INDICATE DIRECTION	Indicate the direction of the slope.

MODIFY >> FAIR

Fair surfaces/faces or curves: reduce waviness, eliminate breakpoints.

If there are no reference curves in the file, the following options are displayed:

SELECT	CURVES
	SURFACES

FAIR >> CURVES

Reduce waviness and eliminate breakpoints of reference curves.

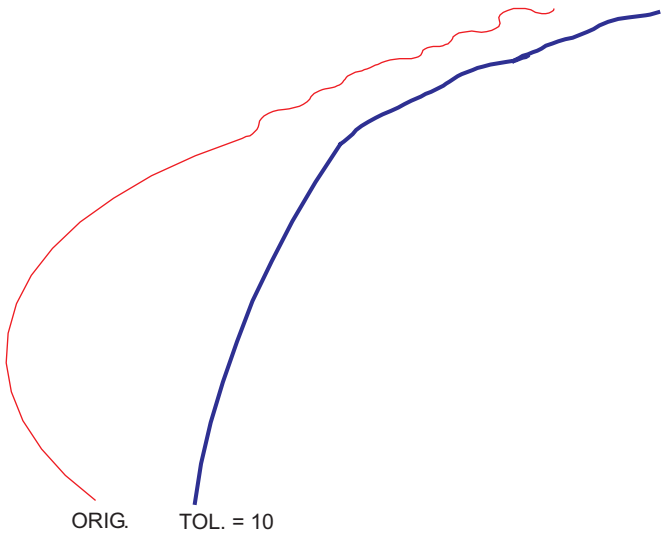

Figure 2-60: MODIFY >> FAIR >> CURVES

How To:

1. Set the parameters.
2. Select the curve(s) to be faired.
3. Press <EXIT> when finished.

Interaction:

PICK CURVES/EXIT	TOLERANCE = 0.0010	FREE SLOPES

PICK CURVES/EXIT Pick curves to be faired. <EXIT> when finished.

FAIR >> SURFACES

Reduce waviness and eliminate breakpoints of reference surfaces.

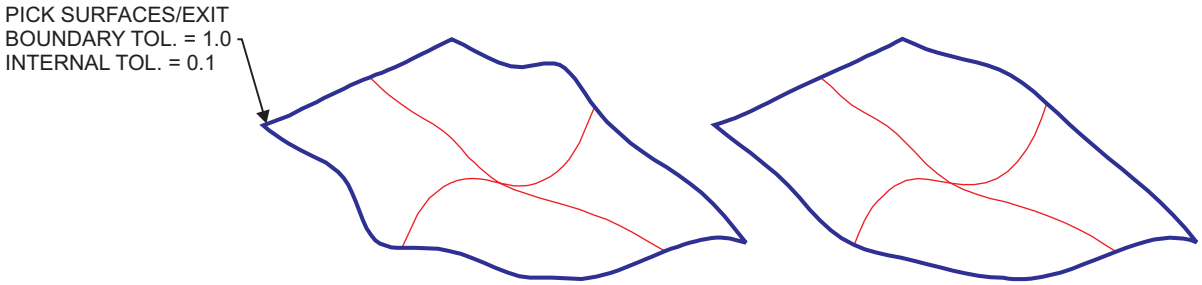

Figure 2-61: FAIR >> SURFACES

How To:

1. Set the parameters.
2. Select the surface(s) to be faired.
3. Press <EXIT> when finished.

Interaction:

PICK SURFACES/EXIT	BOUNDARY TOL = 0.001	INTERNAL TOL = 0.001
	FREE/KEEP SLOPES	

PICK SURFACES/EXIT Pick surfaces to be faired. <EXIT> when finished.

MODIFY >> SCALE

Change the scale factors of surfaces or solid objects using either uniform or non-uniform scaling.

SELECT	UNIFORM
	NON-UNIFORM

SCALE >> UNIFORM

Scale the model using uniform scaling. Enter one scale value to scale the entities.

How To:

1. Indicate a reference point from which all calculations will be performed.
2. Enter a scaling value.
3. Pick the entities to be scaled.
4. Press <EXIT> to scale.

IND. REF. POINT Indicate a reference point from which all calculations will be performed.

PICK ENTITIES & EXIT	SCALE = 1.010

PICK ENTITIES & EXIT Pick the entities to be scaled.

Press <EXIT>.

SCALE >> NON-UNIFORM

Scale the model using non-uniform scaling.

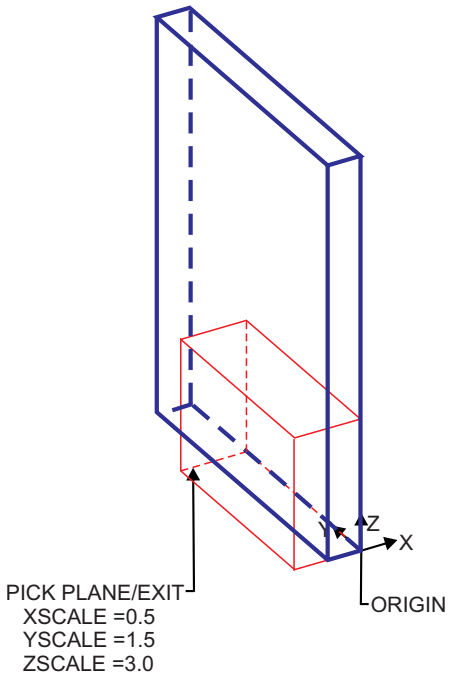

Figure 2-62: MODIFY >> SCALE

How To:

1. Define the reference coordinate system and select APPLY.
2. Set the X, Y and Z scale factors.
3. Pick the surface or solid to be scaled.
4. Press <EXIT> to scale.

Interaction:

REFERENCE SYSTEM	XY PLANE	ORIGIN	+X AXIS	FLIP	APPLY

REFERENCE SYSTEM Define the reference coordinate system and select APPLY.

PICK ENTITIES & EXIT	X SCALE = 0.500	Y SCALE = 0.800	Z SCALE = 0.600

PICK ENTITIES & EXIT Pick the surface or solid to be scaled.
Press <EXIT> to scale.

MODIFY >> INVERT

Display the direction of an external normal to an open object; change the direction to invert the object.

PICK OBJECT Pick the open object. The arrow appears external directed to the object.

INVERT OBJECT? YES NO

YES The object inverts and the arrow changes direction accordingly.

NO Cancel the operation.

Notes:

- Only one object can be inverted. If the picked object is not open, the following message is displayed:

  OBJECT IS NOT OPEN

- This option may be used after using the function TRANSL >> SURF -> SOLID.

MODIFY >> BREAK EDGE

Break a solid edge into 2 edges.

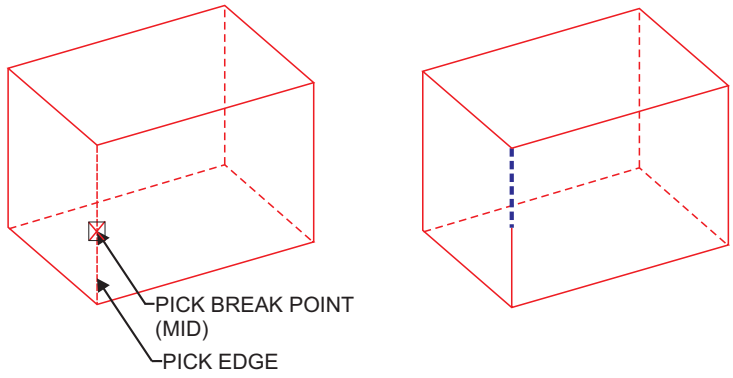

Figure 2-63: MODIFY >> BREAK EDGE

How To:

1. Pick the solid edge to be broken into 2 edges.
2. Pick the break point.
3. Confirm the break point.

Interaction:

PICK EDGE	Pick the solid edge to be broken into 2 edges.
PICK BREAK POINT	Pick the break point.
POINT OK? YES NO	Confirm the break point.

MODIFY >> SPLIT

Split a mold by a plane, surface, or a silhouette line to create additional edges before defining a parting line.

A parting line is based on edges. If there are not enough edges in the model, use the SPLIT option to define additional edges and then define the parting line using SEPARATE >> PARTING LINE (see page 6-12).

When **creating** parting lines, only existing edges may be used. However, parting lines can be **merged** either with other parting lines or with datum curves.

BY PLANE
BY SURFACE
BY SIL. LINE

SPLIT >> BY PLANE

Split faces by a plane to create additional edges before defining a parting line.

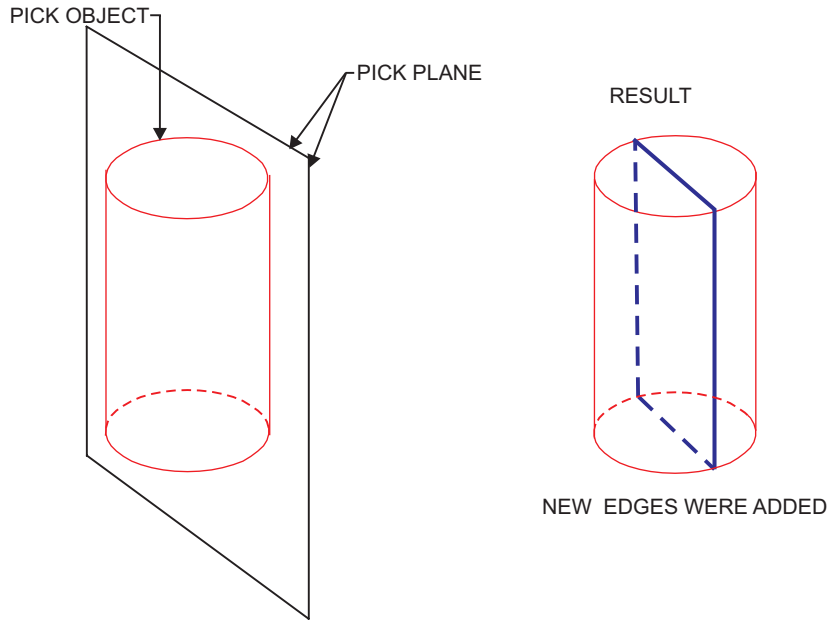

Figure 2-64: SPLIT >> BY PLANE

How To:

1. Pick the splitting plane.
2. Pick the face(s), loop of faces or whole object to be split.
3. Press APPLY.

Interaction:

PICK PLANE Pick the splitting plane.

PICK	FACE	LOOP	OBJECT	APPLY

Note:

- After picking a loop of faces or a whole object, a face(s) can be removed (subtracted) from the selection by picking the FACE modal, and then picking the face(s) to be removed a second time.

SPLIT >> BY SURFACE

Split faces by a surface to create additional edges before defining a parting line.

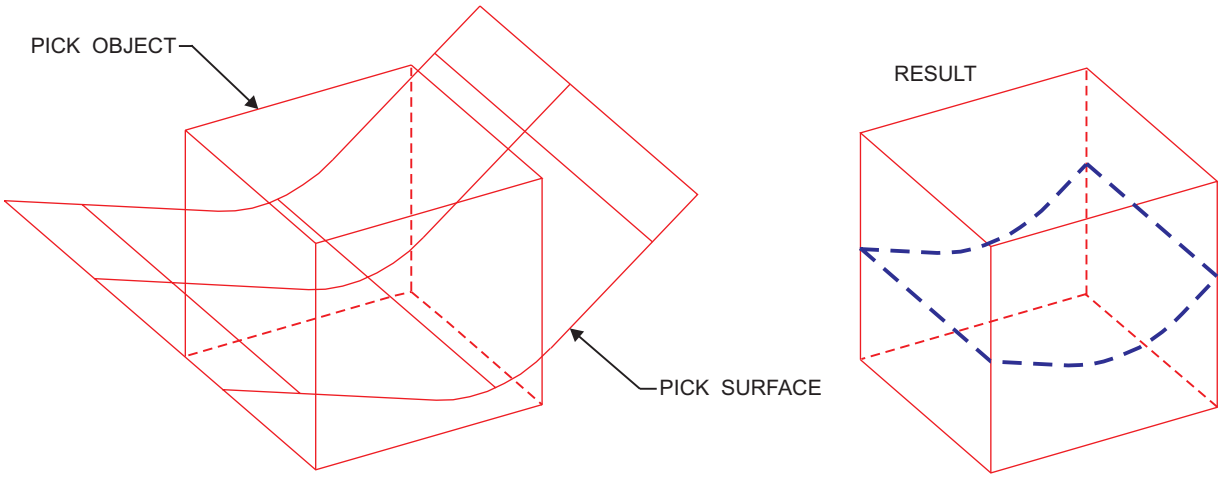

Figure 2-65: SPLIT >> BY SURFACE

How To:

1. Pick the splitting surface.
2. Pick the face(s), loop of faces or whole object to be split.
3. Press APPLY.

Interaction:

PICK SURFACE Pick the splitting surface.

PICK	FACE	LOOP	OBJECT	APPLY

SPLIT >> BY SIL. LINE

Split a mold by a silhouette line to create additional edges before defining a parting line.

MAIN DIRECTION Define the main direction for silhouette line calculation.

The following submenu is displayed:

AXIS
CURVE-END
CURVE-MID
2 POINTS
UCS-X
UCS-Y
UCS-Z
NORMAL

SPLIT >> BY SIL. LINE >> AXIS

Define a direction along an axis or edge.

PICK AXIS/EDGE <PICK> the axis/edge to be used to define the parting line.

DEFINE DIRECTION Select the direction.

PICK	FACE	LOOP	OBJECT	APPLY

SPLIT >> BY SIL. LINE >> CURVE-END

Define a direction from the end of a curve.

PICK CURVE <PICK> the curve whose endpoint is to be used to define the parting line.

DEFINE DIRECTION Select the direction.

PICK	FACE	LOOP	OBJECT	APPLY

SPLIT >> BY SIL. LINE >> CURVE-MID

Define a direction from the midpoint of a curve.

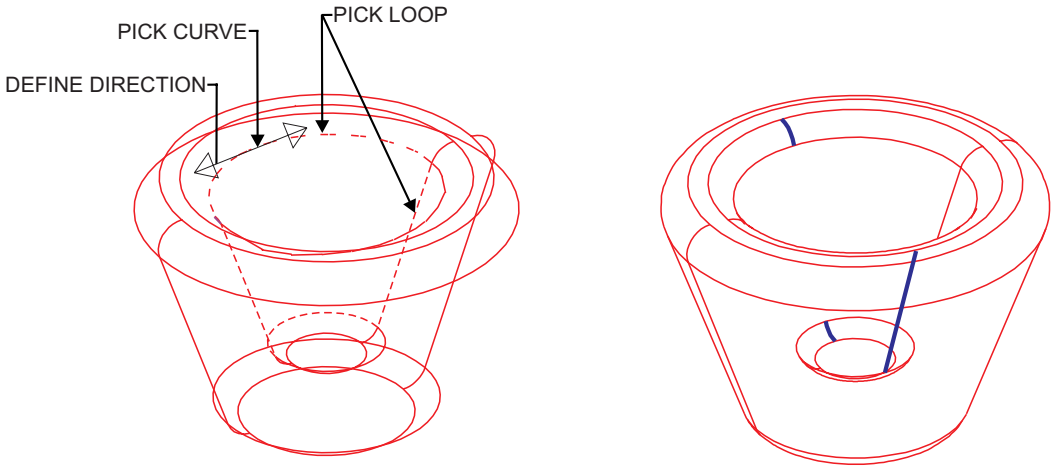

Figure 2-66: SPLIT >> BY SIL. LINE >> CURVE-MID

How To:

1. <PICK> the curve whose midpoint is to be used to define the parting line.
2. Select the direction.
3. Pick the face(s), loop of faces or whole object to be split.
4. Press APPLY.

Interaction:

PICK CURVE <PICK> the curve whose midpoint is to be used to define the parting line.

DEFINE DIRECTION Select the direction.

PICK	FACE	LOOP	OBJECT	APPLY

SPLIT >> BY SIL. LINE >> 2 POINTS

Define a direction using two points.

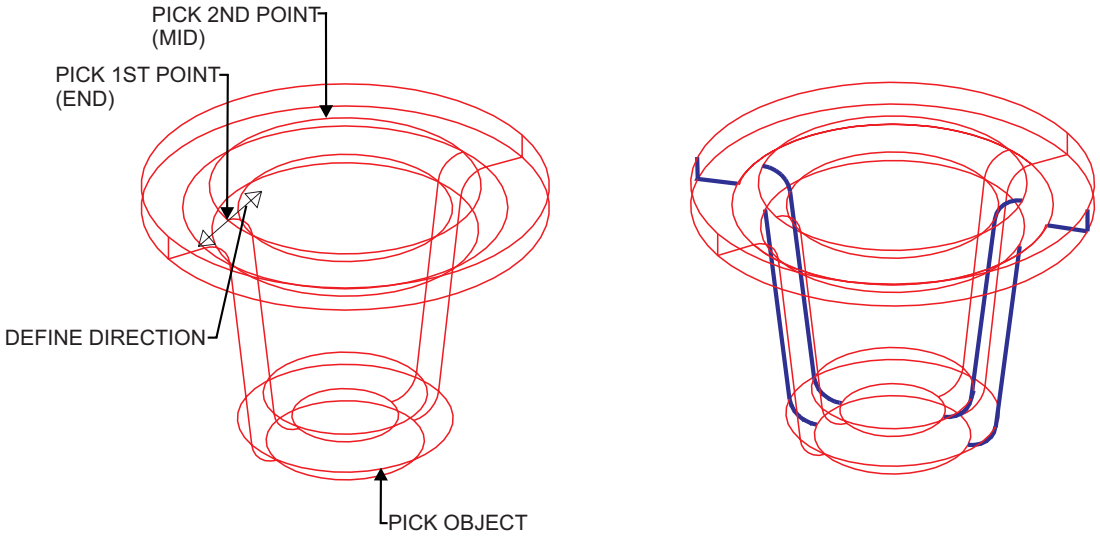

Figure 2-67: SPLIT >> BY SIL. LINE >> 2 POINTS

How To:

1. <PICK> the two points to be used to define the parting line.
2. Select the direction.
3. Pick the face(s), loop of faces or whole object to be split.
4. Press APPLY.

Interaction:

PICK 1ST POINT <PICK> the first point to be used to define the parting line.

PICK 2ND POINT <PICK> the second point to be used to define the parting line.

DEFINE DIRECTION Select the direction.

PICK	FACE	LOOP	OBJECT	APPLY

Note: • The point submenu is available when selecting the points.

SPLIT >> BY SIL. LINE >> UCS-X / UCS-Y / UCS-Z

Define a direction using a user-coordinate system.

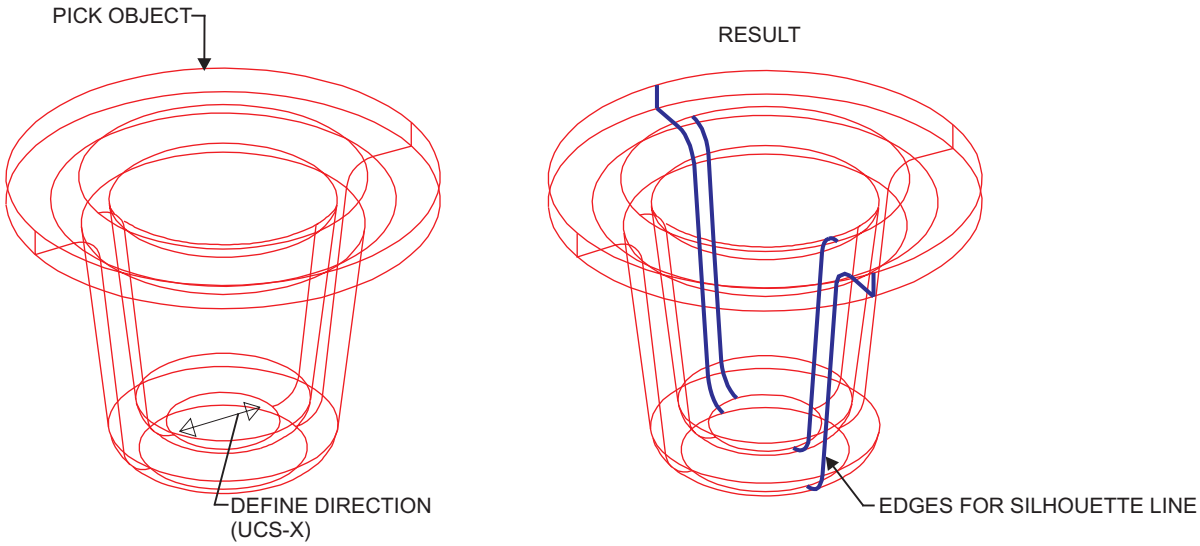

Figure 2-68: SPLIT >> BY SIL. LINE >> UCS-X

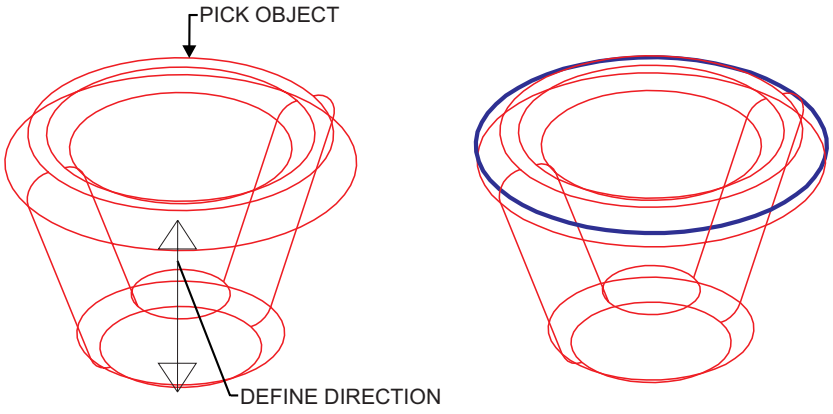

Figure 2-69: SPLIT >> BY SIL. LINE >> UCS-Z

How To:

1. Select the direction to be used to define the parting line.
2. Pick the face(s), loop of faces or whole object to be split.
3. Press APPLY.

Interaction:

DEFINE DIRECTION Select the direction.

PICK	FACE	LOOP	OBJECT	APPLY

SPLIT >> BY SIL. LINE >> NORMAL

Define a direction using the normal.

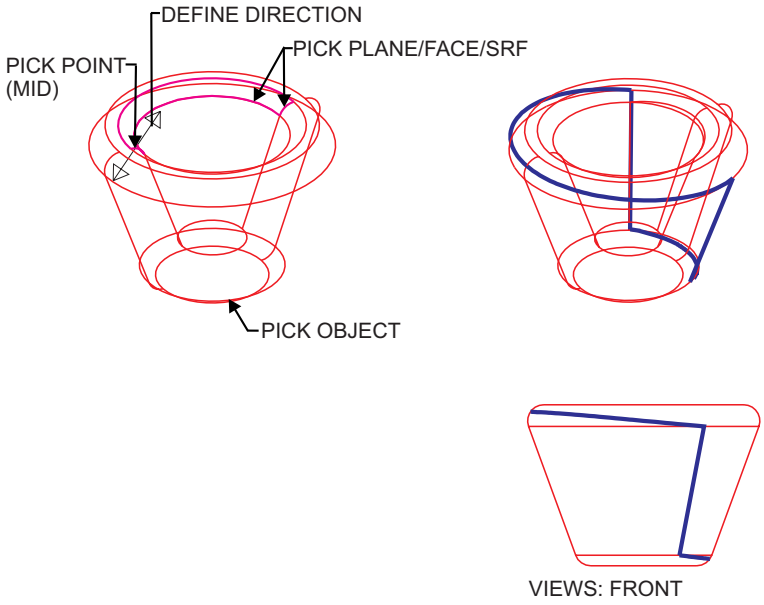

Figure 2-70: SPLIT >> BY SIL. LINE >> NORMAL

How To:

1. <PICK> the plane, face or surface to be used to define the parting line.
2. Select the direction.
3. Pick the face(s), loop of faces or whole object to be split.
4. Press APPLY.

Interaction:

PICK PLANE/FACE/SRF <PICK> the plane, face or surface to be used to define the parting line.

DEFINE DIRECTION Select the direction.

PICK	FACE	LOOP	OBJECT	APPLY

MODIFY >> ADJOIN FACE

Close gaps in open solid objects to adjoin faces.

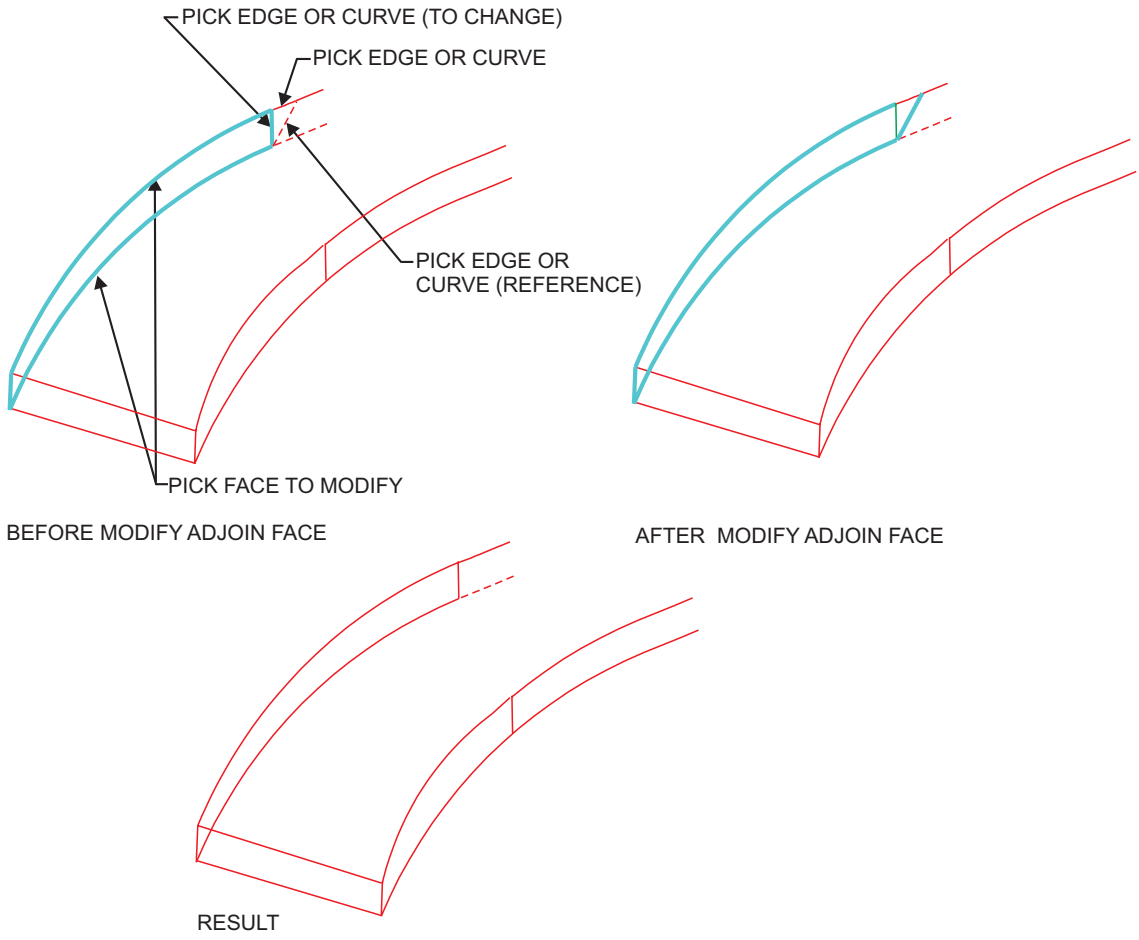

Figure 2-71: SPLIT >> BY SIL. LINE >> NORMAL

How To:

1. Pick the face to be modified. The loops defining the face are displayed in red.
2. Confirm the selected face. If you accept the face, the loops defining the face are displayed in attention mode (colored cyan).
3. If required, edit the selected face.

   You may select REMOVE LOOP to remove the displayed curve(s). You can now add or remove curves as required to close the gaps between adjoining faces.

   You may select RESET LOOP to reset the loop to its initial state.
4. When the message LOOP IS CLOSED NOW is displayed, press APPLY.

Interaction:

PICK FACE TO MODIFY Pick the face to be modified. The face is colored red.

FACE OK? Y/N

Y Accept the selected face. The loops defining the face are displayed in attention mode (colored cyan).

N Pick another face.

PICK EDGE OR CURVE	REMOVE LOOP	RESET LOOP	APPLY

If you select REMOVE LOOP:

OK TO REMOVE? Y/N

Y Remove the displayed curve(s) (now displayed in a different color).

N Do not remove the curve(s).

If you select RESET LOOP:

OK TO RESET? Y/N

Y Reset the loop to its initial state (displayed in cyan).

N Do not reset the loop.

Remove and add curves as necessary.

When the message LOOP IS CLOSED NOW is displayed, press APPLY.

MODIFY: Modal Parameter Definitions

- + DIRECTION — Pick a point to indicate the positive direction of the vector.
- +X AXIS — Define the X axis direction.
- APPLY — Apply the current definitions.
- AXIS — Choose an axis which will be used to define the slope.
- BOUNDARY TOL = 0.001 — Tolerance of the boundary to be used for the fair operation.
- EDGE — Choose a straight edge which will be used to define the slope.
- FACE — Pick the face(s) to be split and select APPLY.
- FLIP — Reverse the indicated direction, if desired.
- FREE SLOPES — Slopes will be changed.
- INTERNAL TOL = 0.001 — Tolerance of the surface to be used for the fair operation.
- KEEP SLOPES — Keep existing slopes.
- LOOP — Pick a loop of faces to be split and select APPLY.
- OBJECT — Pick a whole object to be split and select APPLY.
- ORIGIN — Change/indicate the location of the coordinate system's origin point.
- XY PLANE — <PICK> the plane/face to be the XY plane of the reference coordinate system.

  The reference default coordinate system will be shown.
- REMOVE LOOP — Remove the displayed loop (now displayed in a different color).
- RESET LOOP — Reset the loop to its initial state (displayed in cyan).
- X SCALE
Y SCALE
Z SCALE — Enter the X,Y and Z scale factors.

❑

SURFACE

Create parametric surfaces.

Main Options:

DRIVE
BLEND
OFFSET
MESH
REGION

DRIVE Create a surface by driving sections.

BLEND Create a blend surface between two or more curves/edges.

OFFSET Create a surface at an offset to an existing surface or face of a solid object.

MESH Create a surface with lateral and longitudinal cross sections. Cross sections can be curves, edges, or points.

REGION Create a surface within a closed contour.

SURFACE >> DRIVE

Create a surface by driving sections. The following options are available:

PARALLEL SEC
SPINE
SPINE & PLANE
PIPE

PARALLEL SEC Create a surface by driving a 2D/3D section along a curve/edge. All sections of the surface are parallel to the original driven section.

SPINE Create a surface by driving a 2D/3D section along a contour. The orientation of the surface sections is constant to the driving contour.

SPINE & PLANE Create a surface by moving a 2D/3D section along a contour under the following conditions:

- The orientation of the section to the tangent of the 2D/3D spine at any point is constant.
- The orientation of the section curves to the defined face/plane is constant.

PIPE Create a parametric surface by driving a circle section of fixed or variable radius along a 3D trajectory.

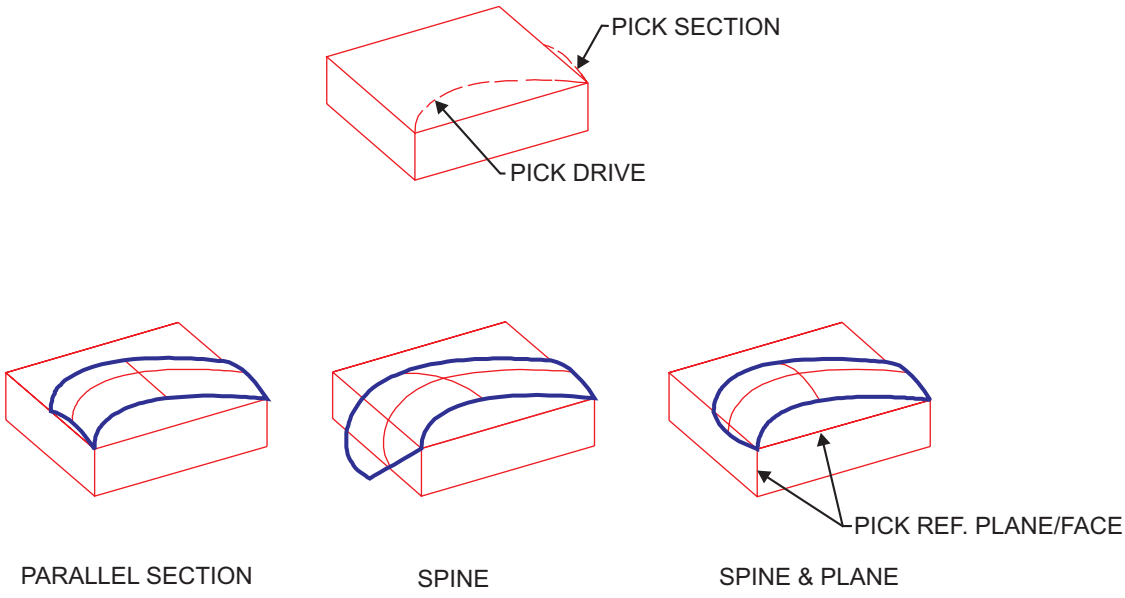

Figure 2-72: SURFACE >> DRIVE (PARALLEL SECTION, SPINE, SPINE & PLANE)

DRIVE >> PARALLEL SEC

Create a surface by driving a 2D/3D section along a curve/edge. All sections of the surface are parallel to the original driven section.

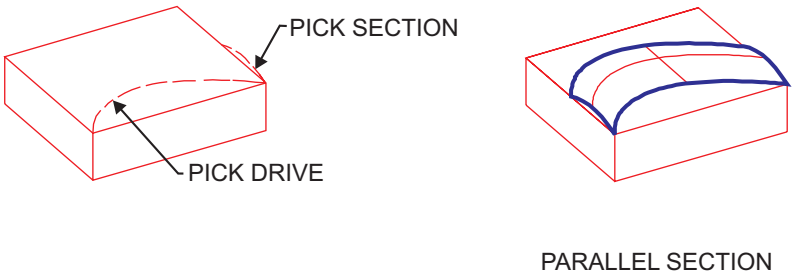

Figure 2-73: SURFACE >> DRIVE (PARALLEL SECTION)

PICK DRIVE — Pick a driving curve or edge.

PICK SECTION — Pick section to be driven.

CONTINUE? YES NO

DRIVE >> SPINE

Create a surface by driving a 2D/3D section along a curve/edge. All sections of the surface are normal to the driving contour.

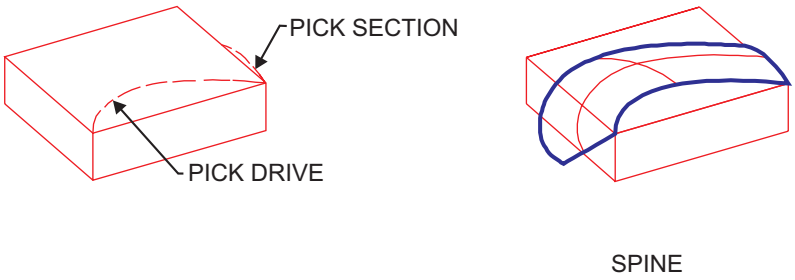

Figure 2-74: SURFACE >> DRIVE (SPINE)

PICK DRIVE

Pick a spine to be the driving curve or edge.

PICK SECTION

Pick section to be driven along the spine.

Press <SUBMENU> to choose an edge or curve:

EDGE
CURVE

CONTINUE? YES NO

DRIVE >> SPINE & PLANE

Create a surface by moving a 2D/3D section along a contour under the following conditions:

- The orientation of the section to the tangent of the 2D/3D spine at any point is constant.
- The orientation of the section to the defined face/plane is constant.

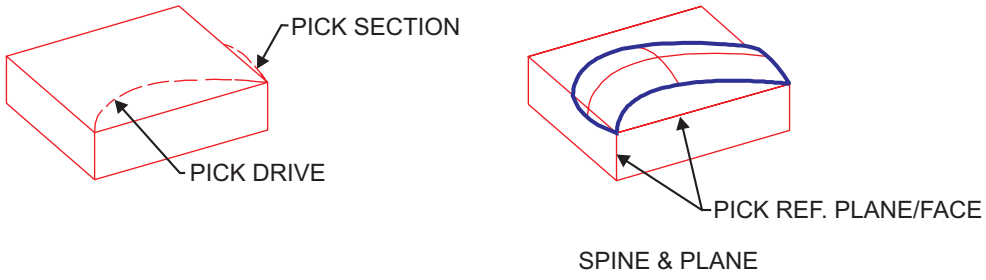

Figure 2-75: SURFACE >> DRIVE (SPINE & PLANE)

PICK REF. PLANE/.FACE Pick the reference plane/face.

PICK DRIVE Pick the drive curve/edge of the drive contour.

PICK SECTION Pick section to be driven.

Press <SUBMENU> to choose an edge or curve:

EDGE
CURVE

DRIVE >> PIPE

Create a parametric surface by driving a circle section of fixed or variable radius along a 3D trajectory.

SELECT	FIXED RADIUS
	VARIABLE RADIUS

PIPE >> FIXED RADIUS

Create a parametric surface by driving a circle section of fixed radius along a 3D trajectory.

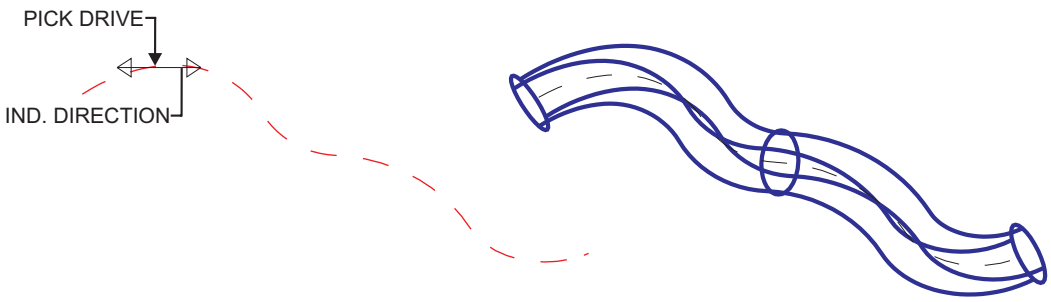

Figure 2-76: SURFACE >> DRIVE >> PIPE (FIXED RADIUS)

How To:

1. Set the Radius of the pipe.
2. Pick the drive trajectory along which the pipe will be created.
3. Indicate the drive direction.
4. Confirm your selections.

Interaction:

PICK DRIVE	RADIUS = 10.000

PICK DRIVE — Pick the drive trajectory along which the pipe will be created.

INDICATE DIRECTION — Indicate the drive direction

CONTINUE? Y/N

Y Create the pipe.

N Re-define the fixed radius.

EXECUTING — The pipe is being created.

PIPE >> VARIABLE RADIUS

Create a parametric surface by driving a circle section of variable radius along a 3D trajectory.

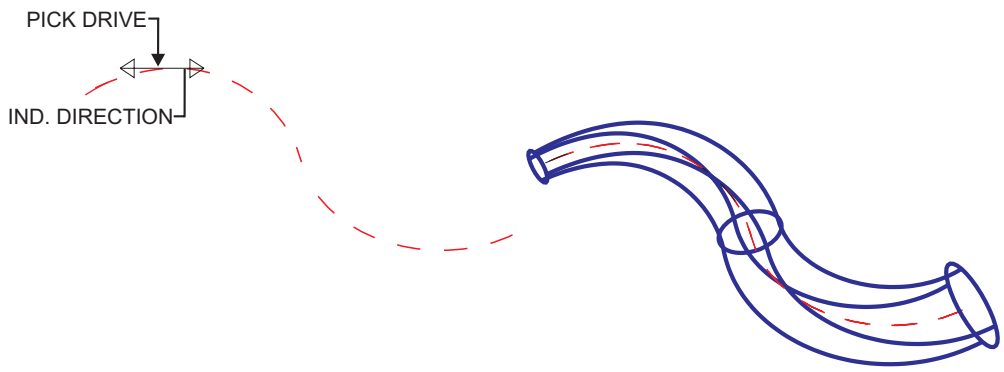

Figure 2-77: SURFACE >> DRIVE >> PIPE (VARIABLE RADIUS)

How To:

1. Set the Start and End Radii of the pipe.
2. Pick the drive trajectory along which the pipe will be created.
3. Indicate the drive direction.
4. Confirm your selections.

Interaction:

PICK DRIVE	START RADIUS = 10.000	END RADIUS = 50.00

PICK DRIVE — Pick the drive trajectory along which the pipe will be created.

INDICATE DIRECTION — Indicate the drive direction

CONTINUE? Y/N

Y Create the pipe.

N Re-define the fixed radius.

EXECUTING — The pipe is being created.

SURFACE >> BLEND

Create a blend surface between two or more curves/edges.

How To:

1. <PICK> curves, edges or points through which the surface will be created. The system will label the FIRST and LAST entities picked. The last section can be a point.
2. If you wish to define the slope of the first or last curve/edge, choose DEFINE SLOPES. Otherwise press <EXIT>.

   If you choose DEFINE SLOPES, first pick an edge (the first or last edge that defines the surface), then define the slope using one of the following methods: FREE, CONSTANT, LINEAR, KEEP NORMAL, KEEP SLOPES.
3. Define side boundaries, if any.
4. Press <EXIT>.

Interaction:

PICK SECTIONS/EXIT <PICK> curves, edges or points which will be the boundaries of the surface. The system will label the FIRST and LAST entities picked. The last section can be a point.

Note: • It is important that the location where you pick each curve is in the same relative location, i.e. closer to the top or side of the model. Picking inconsistently will lead to a "twisted" surface.

DEFINE SLOPES/EXIT	DEFINE SLOPES

DEFINE SLOPES/EXIT If you wish to define the slope of the first or last curve/edge, choose DEFINE SLOPES. Otherwise press <EXIT>.

Press <SUBMENU> to choose an edge, curve or point:

EDGE
CURVE
POINT

If you choose not to define slopes, you will be asked to define side boundaries, if any.

PICK 1ST BOUNDARY/EXIT
PICK 2ND BOUNDARY/EXIT Pick boundaries for the sides of the surface, if any.

If DEFINE SLOPES is chosen, you will receive the following prompt: (After slopes are defined you will receive the PICK 1ST BOUNDARY prompt as described above.)

PICK EDGE Pick the first or last curve/edge that defines the surface.

Choose one of the following options for defining the slope:

FREE
CONSTANT
LINEAR
KEEP NORMAL
KEEP SLOPES

BLEND >> FREE

Define a free slope.

Note:

- This option only appears if CURVE is selected from the DEFINE SLOPES submenu.

Figure 2-78: SURFACE >> BLEND >> FREE

BLEND >> CONSTANT

Define a constant slope.

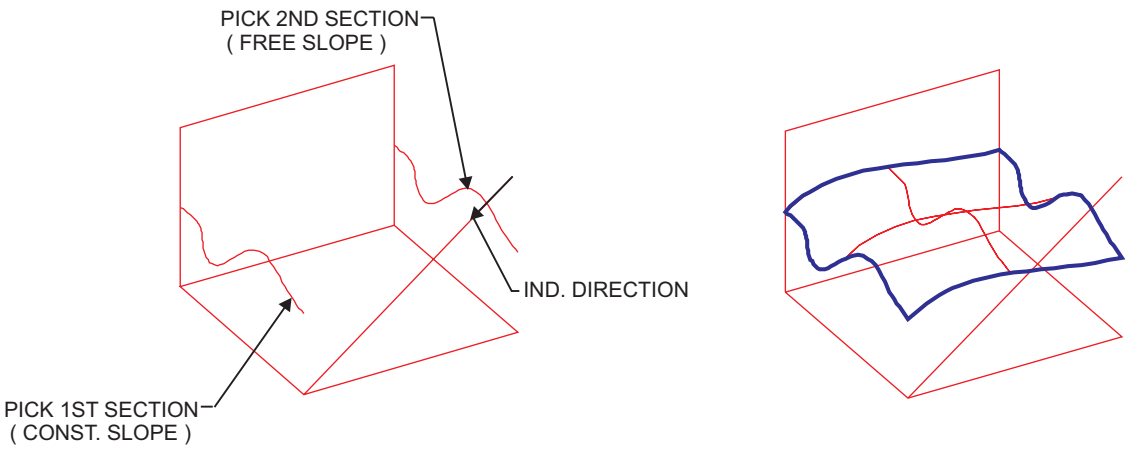

Figure 2-79: SURFACE >> BLEND >> CONSTANT

How To:

There are two ways to define a slope.

1. Use the slope of an existing edge or axis as a reference slope.
   - *a).* Choose an axis or straight edge which will be used to define the slope. Use the AXIS/EDGE parameter.
   - *b).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *c).* Apply the above slope constraints to the picked curve / edge. Use the APPLY button
2. Use the slope of a vector defined by two points: an origin and a point indicating the positive direction.
   - *a).* Indicate an origin point. Use the ORIGIN parameter.
   - *b).* Pick a point to indicate the positive direction of the vector. Use the + DIRECTION parameter.
   - *c).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *d).* Apply the above slope constraints to the picked curve / edge. Use the APPLY button

Interaction:

DEFINE DIRECTION	AXIS/EDGE	FLIP	ORIGIN	+ DIRECTION	APPLY

DEFINE DIRECTION There are two ways to define a slope (see How To, above).

BLEND >> LINEAR

Define a linearly changing slope. Define the slopes at both ends of the curve/edge.

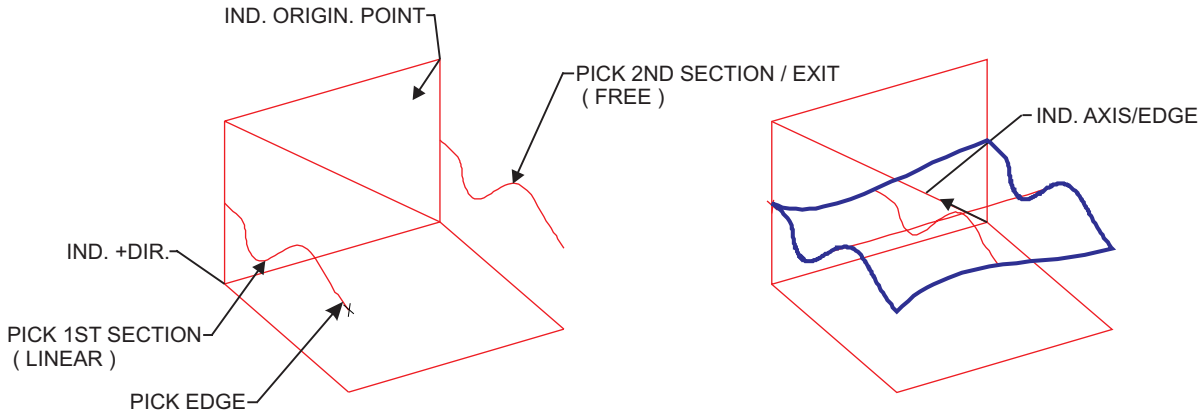

Figure 2-80: SURFACE >> BLEND >> LINEAR

How To:

There are two ways to define a slope.

1. Use the slope of an existing edge or axis as a reference slope.
   - *a).* Choose an axis or straight edge which will be used to define the slope. Use the AXIS/EDGE parameter.
   - *b).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *c).* Apply the above slope constraints to the picked curve / edge. Use the APPLY button
2. Use the slope of a vector defined by two points: an origin and a point indicating the positive direction.
   - *a).* Indicate an origin point. Use the ORIGIN parameter.
   - *b).* Pick a point to indicate the positive direction of the vector. Use the + DIRECTION parameter.
   - *c).* Reverse the indicated direction, if desired. Use the FLIP parameter.
   - *d).* Apply the above slope constraints to the picked curve / edge. Use the APPLY button

Interaction:

DEFINE 1ST DIRECTION	AXIS/EDGE	FLIP	ORIGIN	+ DIRECTION	APPLY

DEFINE 1ST DIRECTION Define the slopes at both ends of the curve/edge.
DEFINE 2ND DIRECTION

There are two ways to define a slope (see How To, above).

BLEND >> KEEP NORMAL

Define the slope such that all normals are collinear to the normals on the edge of a reference plane/face.

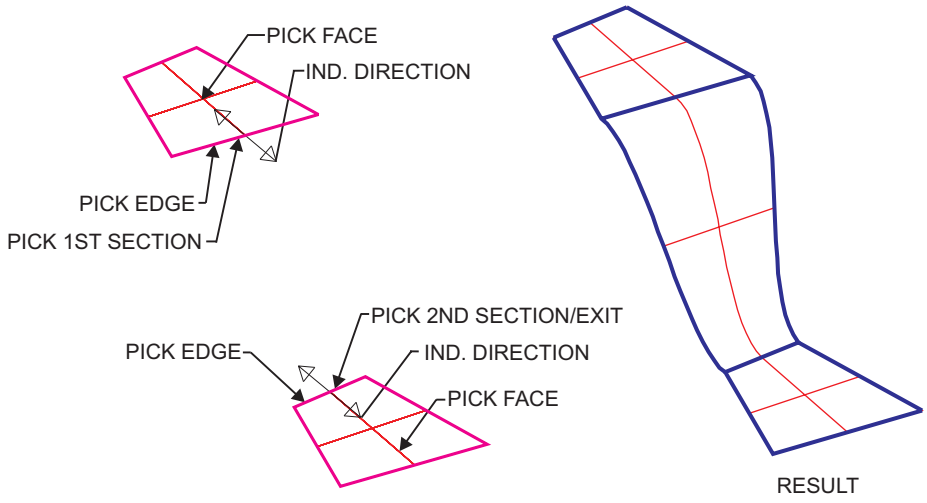

Figure 2-81: SURFACE >> BLEND >> KEEP NORMAL

How To:

1. Pick the reference plane or face.
2. Indicate the direction of the slope.

Interaction:

PICK PLANE/FACE Pick the reference plane or face.

INDICATE DIRECTION Indicate the direction of the slope.

BLEND >> KEEP SLOPES

Define the slope such that it is tangent to the slope on the edge of a reference plane (face).

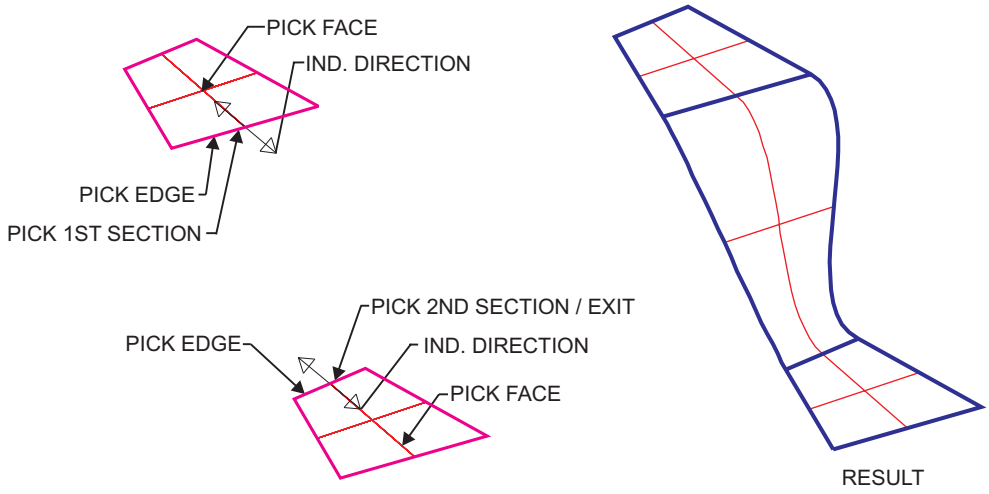

Figure 2-82: SURFACE >> BLEND >> KEEP SLOPES

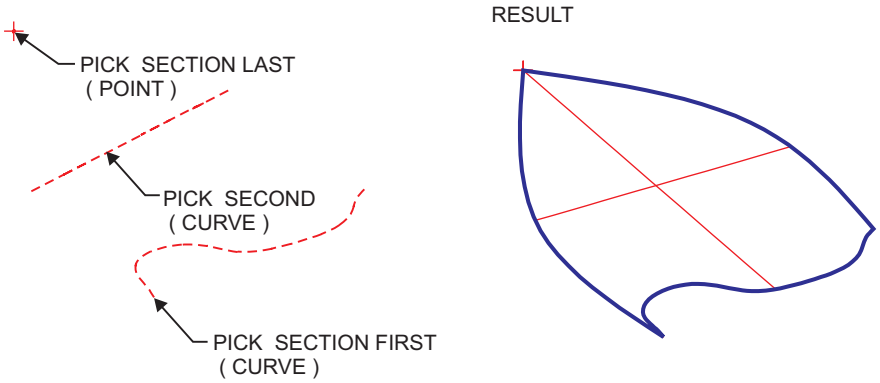

Figure 2-83: SURFACE >> BLEND >> CURVE, POINT

How To:

1. Pick the reference plane or face.
2. Indicate the direction of the slope.

Interaction:

PICK PLANE/FACE	Pick the reference plane or face.
INDICATE DIRECTION	Indicate the direction of the slope.

SURFACE >> OFFSET

Create a surface at an offset to an existing surface, or solid face.

SELECT	SINGLE
	QUILT

SINGLE Only one of the included surfaces/faces of a quilt will be offset.

QUILT All surfaces which are smoothly connected to the selected surface/face will be offset.

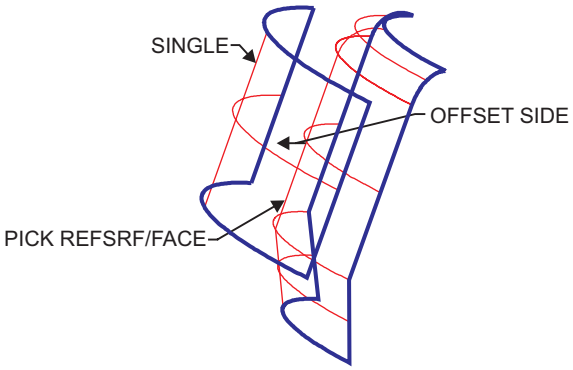

Figure 2-84: SURFACE >> OFFSET >> SINGLE

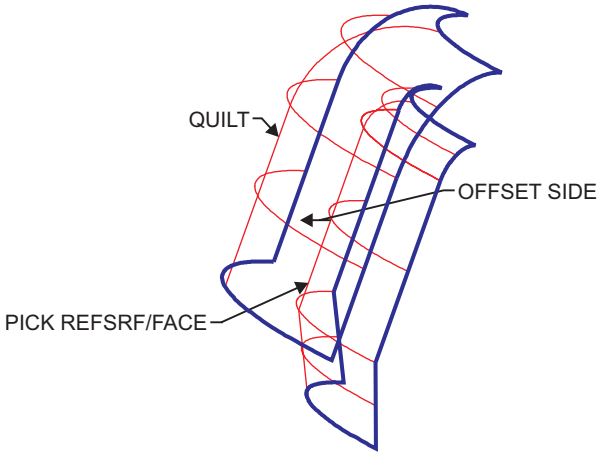

Figure 2-85: SURFACE >> OFFSET >> QUILT

How To:

1. Pick a reference surface or face from which the new surface will be offset.
2. Define the offset value and direction.

Interaction:

PICK REFSRF/FACE Pick a reference surface or face from which the new surface will be offset.

<CR> TO CONTINUE	OFFSET = 10.000	FLIP SIDE

SURFACE >> MESH

Create a surface with lateral and longitudinal cross sections. Cross sections can be curves, edges, or points.

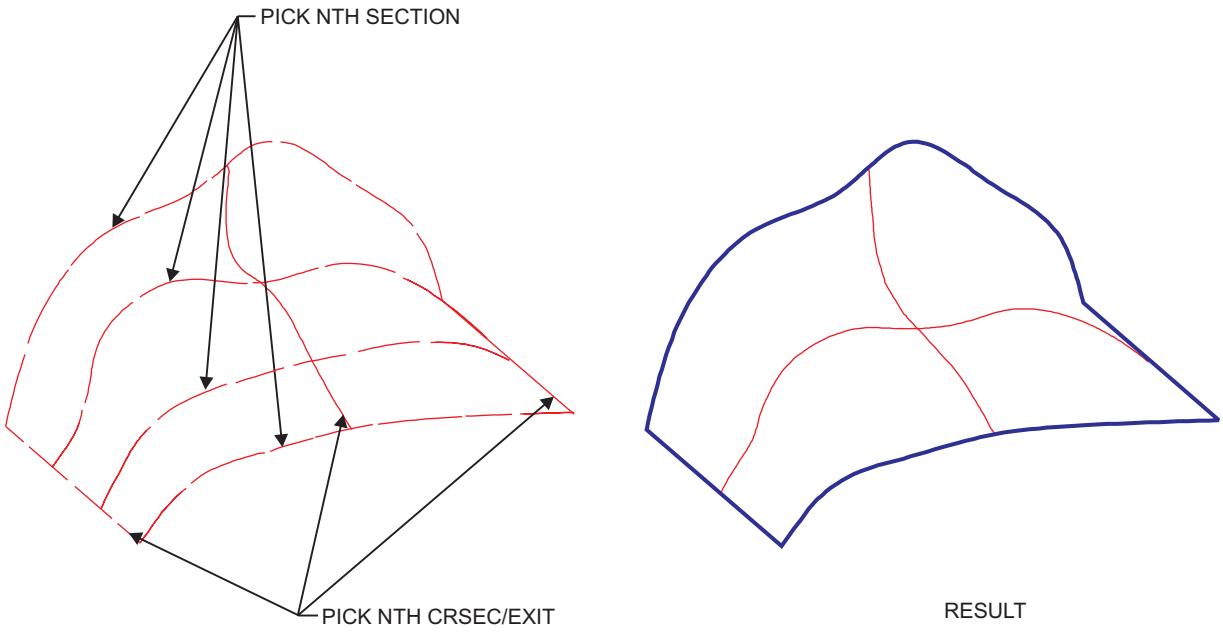

Figure 2-86: SURFACE >> MESH

How To:

1. Pick a series of sections that will define the surface in one direction. Activate the submenu for various picking choices.
2. Press <EXIT> when finished
3. Pick a series of cross-sections that will define the surface in the second direction.
4. Press <EXIT> when finished

Interaction:

PICK 1ST SEC/EXIT
PICK 2ND SEC/EXIT

Pick a series of sections that will define the surface in one direction. Press <EXIT> when finished. Activate the submenu for the following picking choices:

EDGE
CURVE
POINT

PICK 1ST CRSEC/EXIT
PICK 2ND CRSEC/EXIT

Pick a series of cross-sections that will define the surface in the second direction.

SURFACE >> REGION

Create a surface within a closed contour. The surface can be constrained to pass through specified entities.

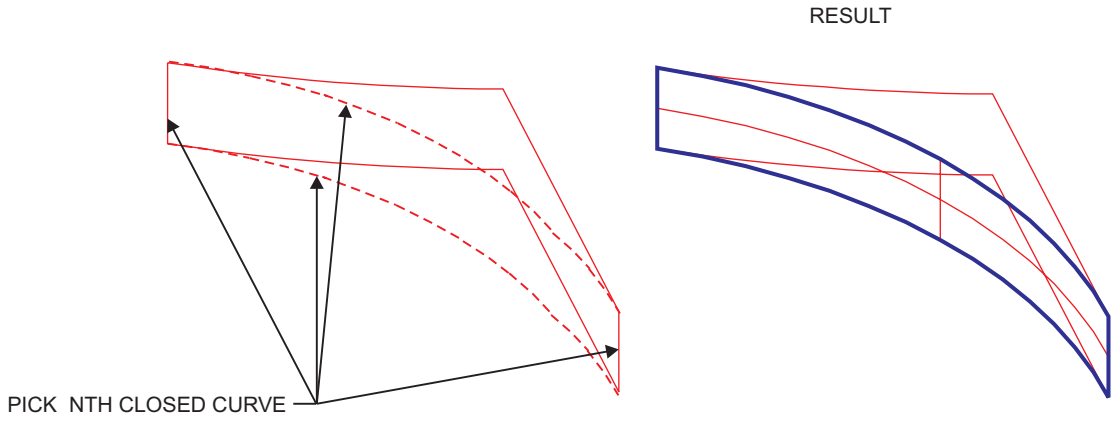

Figure 2-87: SURFACE >> REGION

How To:

1. Pick a series of edges or curves that form a closed contour. Activate the submenu for various picking choices.

   If a closed loop is detected, you will be asked to confirm the curves.

2. If required, define a reference plane or reference entities.

   The surface can be constrained to pass through reference entities. A reference plane can be used for creating the region surface. If no constraints are to be specified, press <EXIT>.

Interaction:

PICK CLOSED CONTOUR Pick a series of edges or curves that form a closed contour. Activate the submenu for the following picking choices:

LOOP
EDGE
CURVE

After the first curve/edge is picked, the following prompt is displayed:

PICK EDGE/CURVE

Note: • If the LOOP option was selected, the created surfaces will be stitched automatically to the surrounding faces.

When the system detects a closed loop you will be asked to confirm the curves:

CURVES OK? YES NO Confirm curves or pick again.

DEFINE REFERENCES/EXIT	REF PLANE	REF ENTITIES

The surface can be constrained to pass through reference entities. A reference plane can be used for creating the region surface. If no constraints are to be specified, press <EXIT>.

SURFACE: Modal Parameter Definitions

- **+ DIRECTION** — Pick a point to indicate the positive direction of the vector.
- **APPLY** — Apply the above slope constraints to generate a new spline.
- **AXIS** — Choose an axis which will be used to define the slope.
- **EDGE** — Choose a straight edge which will be used to define the slope.
- **END RADIUS** — For variable radius options, define the end radius of the circle section.
- **OFFSET = 10.000** — Enter the offset from the chosen surface.
- **ORIGIN** — Indicate an origin point.
- **FLIP** — Reverse the indicated direction, if desired.
- **FLIP SIDE** — Take offset to the other side of the surface.
- **RADIUS** — Define the radius of the circle section.
- **REF PLANE** — Specify the plane.

  PICK PLANE/EXIT — Pick plane/face.
- **REF ENTITIES** — Specify entities though which the surface will pass.

  PICK ENTITIES/EXIT — Pick edges, curves, and/or points. Activate the submenu for help in picking.

EDGE
CURVE
POINT

- **START RADIUS** — For variable radius options, define the start radius of the circle section.

SURFACE: Usage Envelope

1. In SURFACE >> REGION, when closing holes and open faces in the solid object, it is recommended to try the LOOP option first (pick LOOP from the submenu). If the surface can be created, it will be automatically stitched to the object.

❑

TRANSL

Translate entities between wireframe and solid.

Main Options:

WF -> DATUM
SURF -> SOLID
SOLID -> WF

WF -> DATUM Create a datum from wireframe entities.

SURF -> SOLID Convert surfaces and planar faces into solid objects.

SOLID -> WF Create wireframe entities from a solid geometry.

TRANSL >> WF -> DATUM

Create a solid geometry from wireframe entities.

How To:

1. Pick the entities (surfaces, lines, curves, points, etc.) will be created as datum entities.
2. Press <EXIT> to execute.

Interaction:

PICK ENTITIES & EXIT Pick the entities (surfaces, lines, curves, points, etc.) will be created as datum entities.

TRANSL >> SURF -> SOLID

Converts surfaces and planar faces into solid objects. All selected surfaces and planar faces are converted into open solid objects. Automatic stitching is performed between these open objects.

It is possible to create more than one open or closed object.

SELECT	SURFACES TO SOLID
	FIX TRIMMED SURF
	DELETE DUPLICATED

SURFACES TO SOLID Convert the surfaces and planar faces into solid objects.

FIX TRIMMED SURF Verify and fix bad trimmed surfaces (surfaces with incorrect contours, self-intersected contours, etc.). See TRMSRF >> FIX BOUNDARY.

DELETE DUPLICATED Verify and delete exactly coinciding surfaces. See USER >> GEOMETRY >> DELDUP.

SURF -> SOLID >> SURFACES TO SOLID

Convert the surfaces and planar faces into solid objects.

PICK SURFACES & EXIT	OBJECT TOL.=0.001	KEEP SURFACES / DELETE SURFACES	WITH RECOMMEND. TOL / WITHOUT RECOMMEND TOL
	AUTO STITCH / WITHOUT STITCH		

Execution Messages

The following messages appear during the SURFACES TO SOLID conversion process:

1. UPDATE BOUNDARIES – trimmed surfaces are checked and prepared for conversion.

2. SURF. TO FACES XX% - conversion operation. Every surface is converted into a solid object.

3. VERTICES REVISION – solid vertices and edges are recalculated in order to obtain a more accurate solid model.

4. BREAK EDGES – solid edges are divided as needed for stitching purposes.

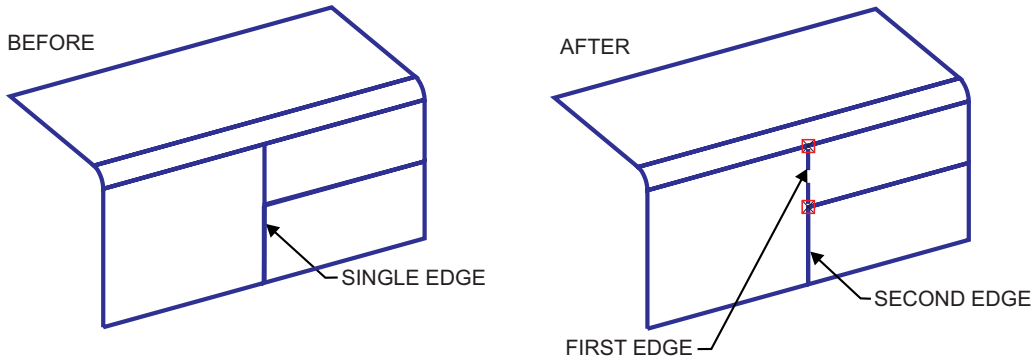

Figure 2-89: Break Edges

5. RECOMMEND. TOLERANCE – the recommended tolerance is calculated for the given surfaces model.

6. FACES MERGING XX% - solid objects created in Step 2 are stitched.

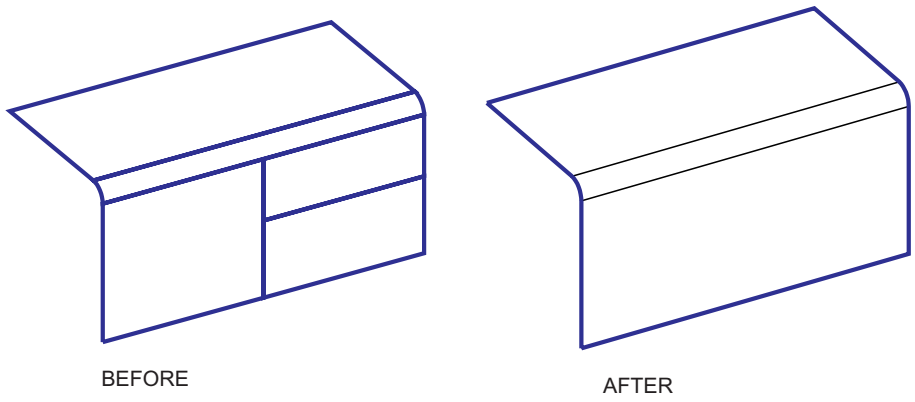

Figure 2-88: Stitched Objects

Interrupting the Conversion Process

The SURFACES TO SOLID execution can be interrupted at any stage by pressing the space bar. The following options appear:

SELECT	CONTINUE	PARTIAL RESULTS	CANCEL

CONTINUE	Continue the conversion process.
PARTIAL RESULTS	Stop the conversion process, but keep all results that were obtained before the process was interrupted.
CANCEL	Cancel the conversion process.

Results of the Conversion Process

The resulting solid objects are displayed and the following message appears:

CREATED OBJECTS: CLOSED –1; OPEN –2

BAD SURFACES: PROBLEMATIC –2; SMALL –5

Problematic surfaces are marked in red with Pen 3. These are surfaces that were not successfully converted into solid faces.

Problematic Surfaces

The following situations will cause a problematic surface to be created:

1. The surface is not smooth. Fair the surface using MODIFY >> FAIR.

2. The surface boundary contains too many points. In this case, the surface can be divided by parameter.

3. The distance is too small between two neighboring points on the boundary of the trimmed surface.

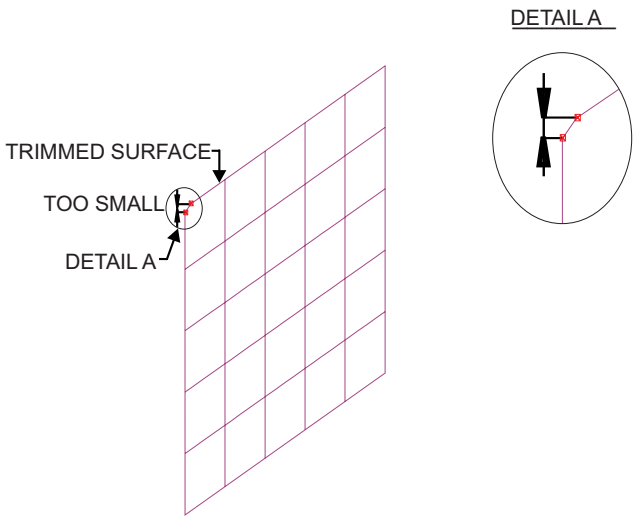

Figure 2-90: Boundary Distance Too Small

4. The surface is too small, nearly degenerated.

5. The trimmed surface has self-intersecting boundaries and/or incorrect contours.

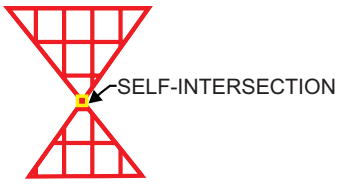

Figure 2-92: Self-Intersecting Boundary

6. The trimmed surface consists of disconnected parts.

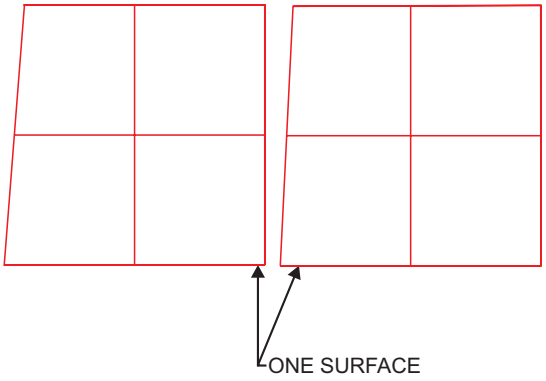

Figure 2-91: Disconnected Parts

Note: • For details on repairing problematic surfaces, see page 2-147.

Small Surfaces

Surfaces are considered small when their width is less than 10 x tolerance. Small surfaces are marked in green with Pen 3.

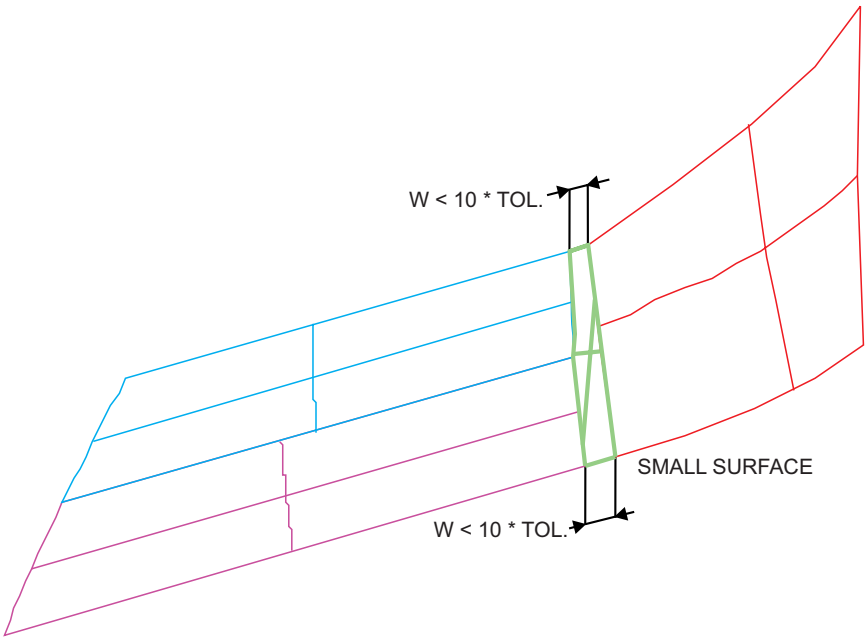

Figure 2-93: Surface Too Small

Checking Resultant Solid Geometry and Topology

To check the results of SURFACES TO SOLID conversions, use UTILITY >> CHECK SOLID or VERIFY >> SOLID >> OBJECT. Entities are shown in various colors to reflect their status. See page 2-148 for details.

Healing Tools

The following solid functions can be used to repair open solid object(s):

SURFACE >> REGION

SURFACE >> BLEND or MESH

DELETE FACE (then create replacement surface)

DETAIL >> STITCH

MODIFY >> BREAK EDGE

MODIFY >> ADJOIN FACE

MODIFY >> SPLIT FACE

TRMSRF, ADJOIN or other Modeling functions.

Usage Recommendations

1. Prepare the surface model for conversion

If you receive a model created in IGES, VDA or other DI format, do the following:

- In Read IGES, use the option EQUAL in TRIM CURVE PREFERENCE.
- In all Read DI functions, use the option VERIFY in the CHECK TRIM SURFACE.

If you use only the VERIFY option (not FIX) in Read DI functions, you will see these trimmed surfaces later when you open the converted pfm file.

Bad trimmed surfaces that can be fixed are shown in magenta. Trimmed surfaces that cannot be fixed are shown in red.

Check and fix trimmed surfaces by using the FIX TRIMMED SURF option of TRANSL >> SURFACES TO SOLID. You can view the bad trimmed surfaces, and set proper LOOP SIZE parameters for fixing self-intersecting boundaries. You will have to choose either to divide trimmed surface at the intersection point (smaller LOOP SIZE parameter), or to remove unnecessary loop (larger LOOP SIZE).

Surface that are fixed before conversion will not be considered problematic during conversion.

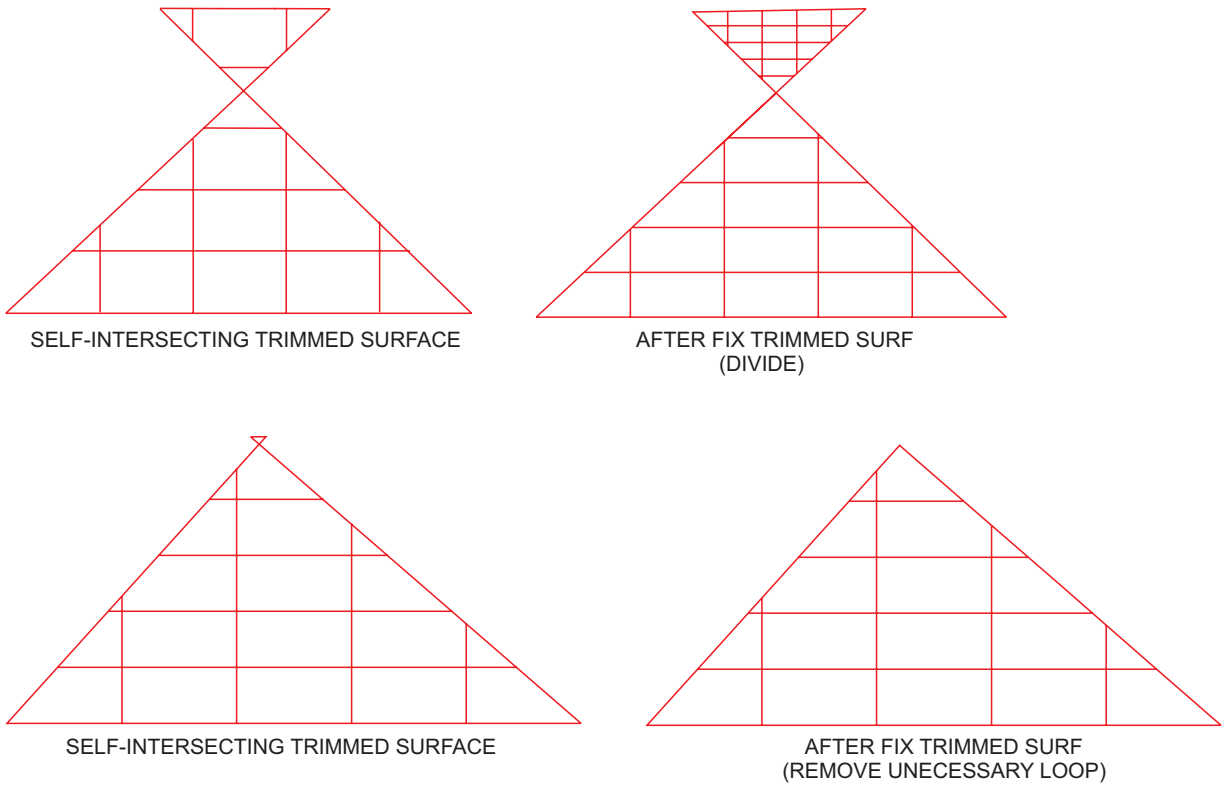

Figure 2-94: Using FIX TRIMMED SURF

Check for duplicate surfaces by using the DELETE DUPLICATED option of TRANSL >> SURFACES TO SOLID. By deleting duplicated surfaces you can eliminate the creation of additional and unnecessary solid objects during the conversion.

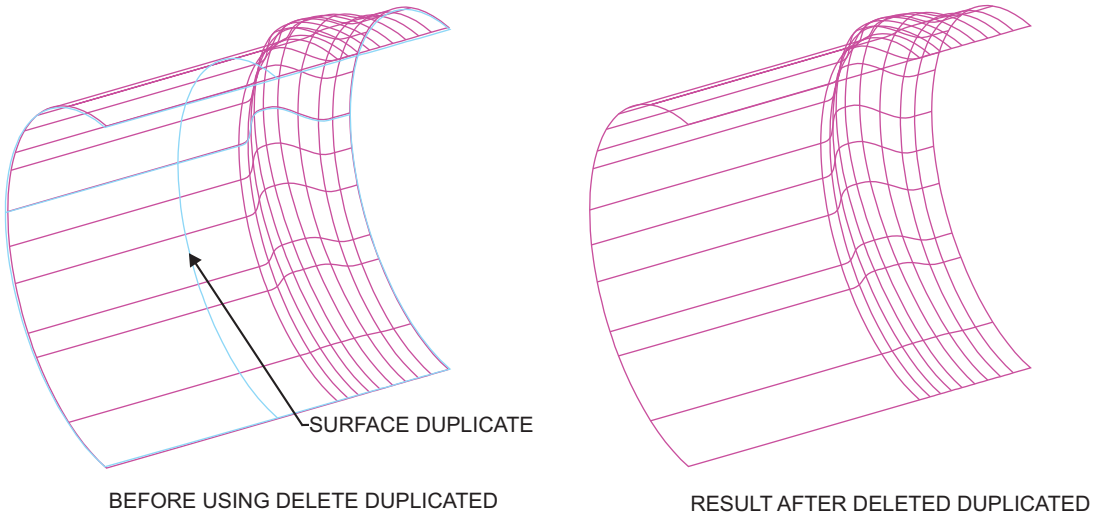

Figure 2-95: Delete Duplicate Surfaces

Detect and fix overlaps between neighboring surfaces. Overlaps may cause extra objects to be created, or may cause incorrect stitching.

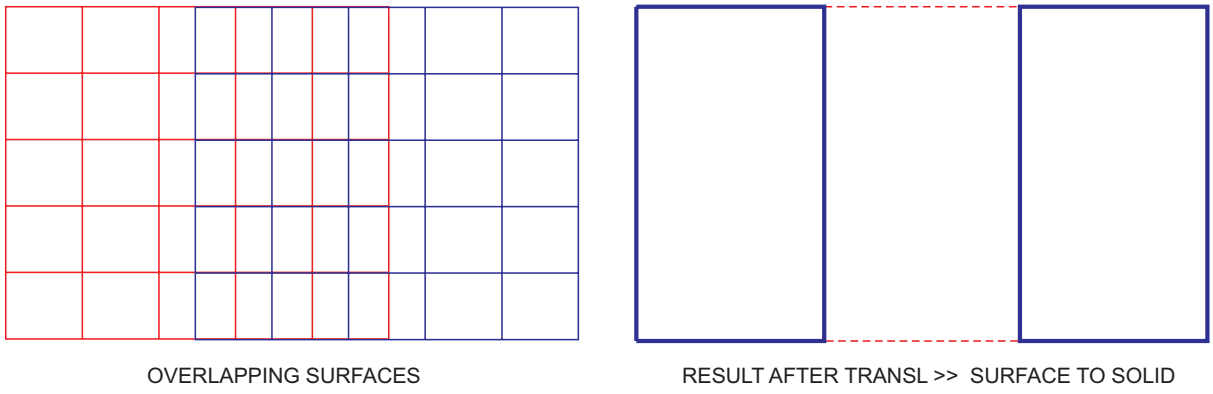

Figure 2-96: Overlapping Surfaces and Incorrect Stitching

To detect overlaps, use USER >> GEOMETRY >> MFILET. Select the VERIFY GEOMETRY option, and pick some or all surfaces. If overlaps are found, the following message will appear: TWO SURFACES OVERLAP EACH OTHER. Overlaps are displayed by thick green lines.

The functions MODIFY >> FAIR and USER >> GEOMETRY >> APPBEZ can also be used to repair the surfaces. These two functions are recommended for models received from Surfaces systems, as opposed to Solid systems. Use a smaller tolerance for approximation in order to avoid gaps in the results. It is also possible to fair only part of the model, rather than the whole model.

Because solid conversion for large models (over 2000 surfaces) is very time consuming, it is recommended to divide such models into several groups of surfaces. Each group of surfaces should be placed in a separate level. Perform SURFACE TO SOLID for each group of surfaces, displaying only the relevant levels, then stitch all objects using DETAIL >> STITCH. This approach saves calculation time, and the temporary created objects are easy to correct and modify.

Create an active level in which to create solid objects. You can then easily switch between the solid and surfaces models. Use the option KEEP SURFACES in the SURFACES TO SOLID conversion. You can use these surfaces later if gaps are created in the solid model which cannot be repaired using standard repair tools.

2. Choose the proper object tolerance

If possible, remember the tolerance used while creating the surface model. Use a smaller tolerance value in the original system in which the model was created.

When using SURFACES TO SOLID, start with the default tolerance. Pick a small group of surfaces (for example, at the corner or in the middle) and check the results of the calculation.

Based of tests performed on several different models, the following tolerances are recommended for first iterations:

- Small parts (1-10 m) - 0.001 mm
- Medium parts (10 - 500 mm) - 0.01 mm
- Large parts (500 - 1000 mm) - 0.1 mm

After the conversion, you will receive the following message:

SMALL SURFACE: RECOMMENDED TOLERANCE LESS THAN 0.002375

SHORT EDGES: RECOMMENDED TOLERANCE LESS THAN 0.001386

CREATED OBJECTS: CLOSED – 0, OPEN – 2,

BAD SURFACES: PROBLEMATIC – 1, SMALL –2.

In the above case, “small” surfaces (shown in green) are ignored and their boundaries are considered as one vertex. Because these surfaces are to be included in the solid model, you will

need a tighter tolerance. Use the recommended tolerance. (Another way to avoid “small” surfaces is to rebuild these surfaces and perhaps their neighbors.)

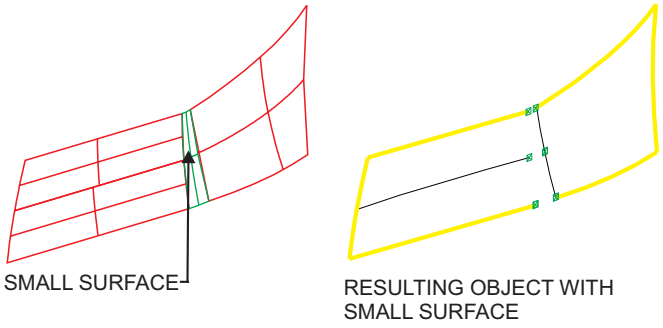

Figure 2-98: Small Surfaces

It is strongly recommended to avoid short edges in your Solid model. Short edges can cause problems in Boolean operations. Use the recommended tolerance for these edges, or rebuild the surfaces.

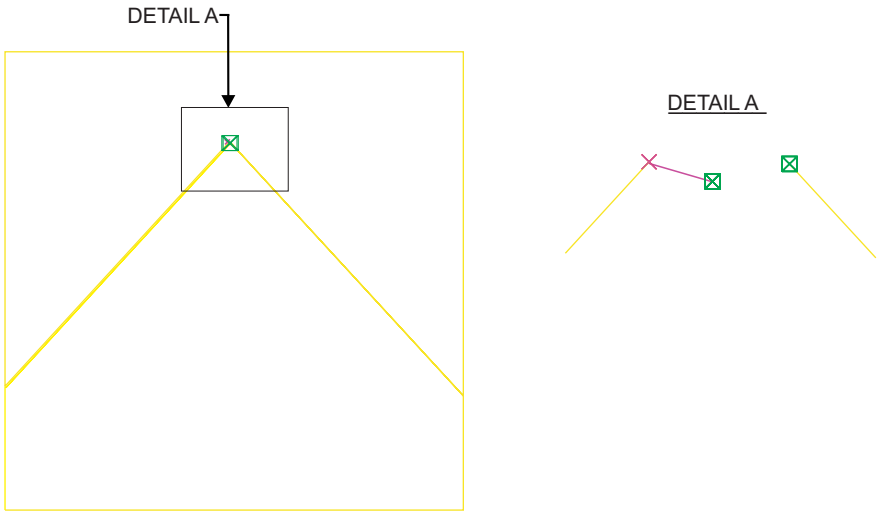

Figure 2-97: Edges that are Too Short

Remember: if you use a tighter tolerance, the number of “small” surfaces decreases, but number of open edges and open “unstitched” objects increases. You will have to close these gaps and stitch the objects, but this is preferable to leaving small surfaces in the model.

3. Analyze results and check the solid

Information about the conversion results appears on both the screen and console. Ideal results will look like the following:

CLOSED – 1, OPEN – 0,

PROBLEMATIC – 0, SMALL – 0.

If only one open object is created, you can easily close it. But if more than one open object is created, it is likely that some objects were not stitched.

In such cases, do the following:

1. Check for duplicated surfaces
2. Try to stitch the objects using DETAIL >> STITCH.
3. Use other solid repair tools to stitch the objects.

When problematic surfaces are created (shown in red):

In most cases these are incorrectly trimmed surfaces that have not been converted into solid faces. Sometimes you can ignore problematic surface (for example, surfaces that are too small). In other cases, you can close the gap because the problematic surface was not included into solid model. The SURFACE >> REGION function, for example, can be used to close gaps.

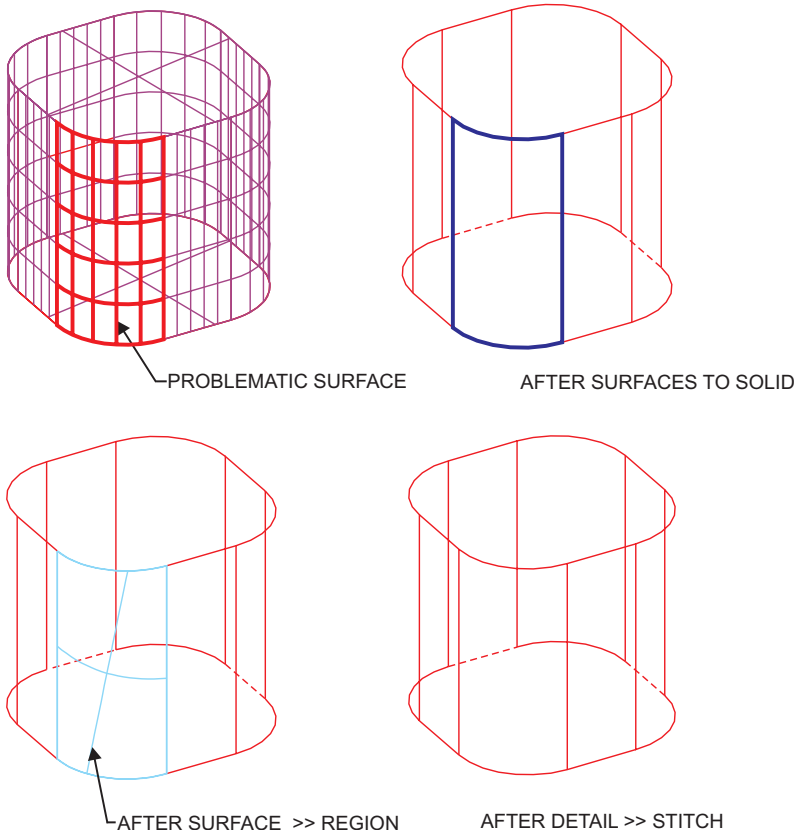

Figure 2-99: Fixing Problematic Surfaces Using SURFACE >> REGION

In most cases, problematic surfaces should be repaired and not ignored.

How to repair problematic surfaces:

1. Try SURFACES TO SOLID >> FIX TRIMMED SURF (or TRMSRF >> FIX BOUNDARY) if you have not done so already.

2. Try MODIFY >> FAIR or USER >> GEOMETRY >> APPBEZ in the WIREFRAME environment. Problematic surfaces will be approximated by another surface. Unsmooth surfaces or surfaces with small waves will be faired, and boundaries of trimmed surfaces will be redefined.

3. Use TRMSRF >> PARAMETER to divide the problematic surface into 2 or more surfaces, in order to simplify it.

4. Use TRMSRF >> MODIFY BOUNDARY to find and remove unnecessary points on boundaries of trimmed surfaces.

5. Create boundary curves by the SRFCRV function. Restore the original surfaces by using TRMSRF >> ORIGINAL. Then trim the original surfaces by the created boundary curves. For example, use TRMSRF >> CONTOUR. In this way you can redefine the trimmed surface.

While trimming, try to avoid closely located break points on boundaries of trimmed surfaces.

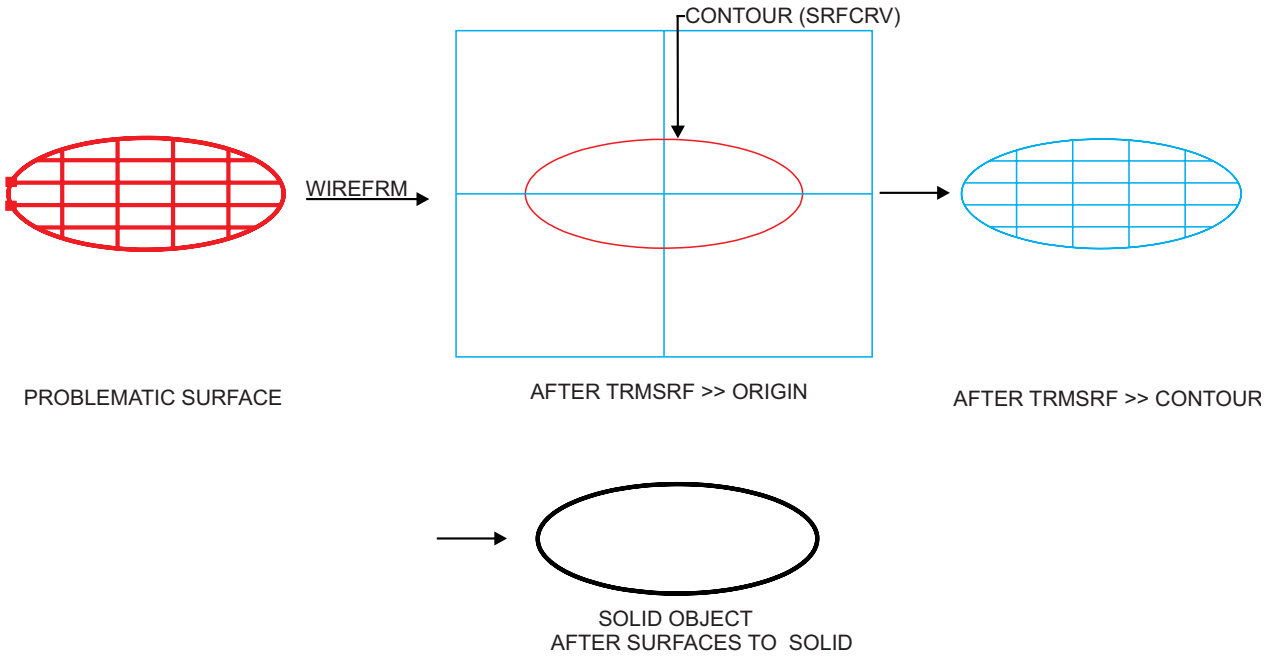

Figure 2-100: Fixing Problematic Surfaces Using TRMSRF

It is strongly recommended to check the objects created in SURFACES TO SOLID. Use UTILITY >> CHECK SOLID or VERIFY >> SOLID >> OBJECT (see page 3-9). You will see the gaps between open objects and within each open object shown in bold yellow lines. Problematic solid geometry and topology are shown in various colors to reflect their status.

Thick yellow lines: open solid edges. These can be boundaries of open objects or gaps between faces within one open object.

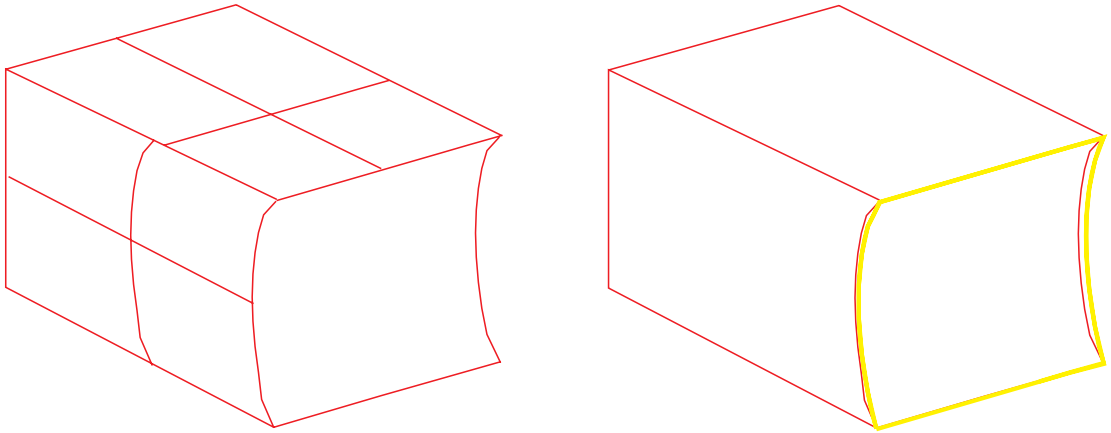

Figure 2-101: Yellow Lines Showing Open Solid Edges

Red bordered face: a face containing loop(s) with wrong directions.

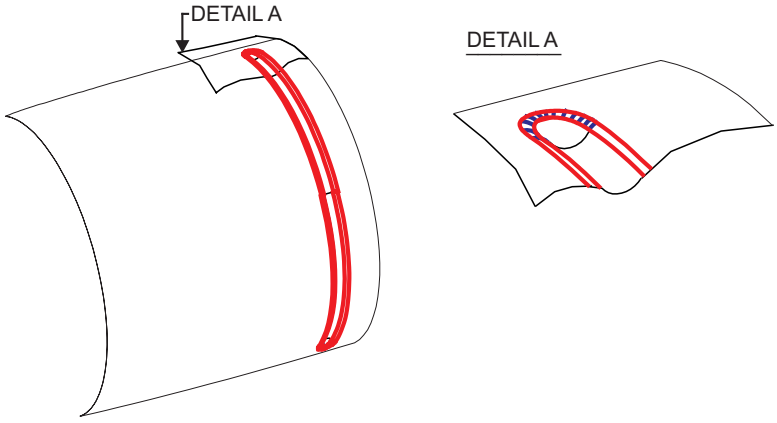

Figure 2-102: Red Bordered Face Containing Loops with Wrong Direction

Magenta lines and points: edges that are too short.

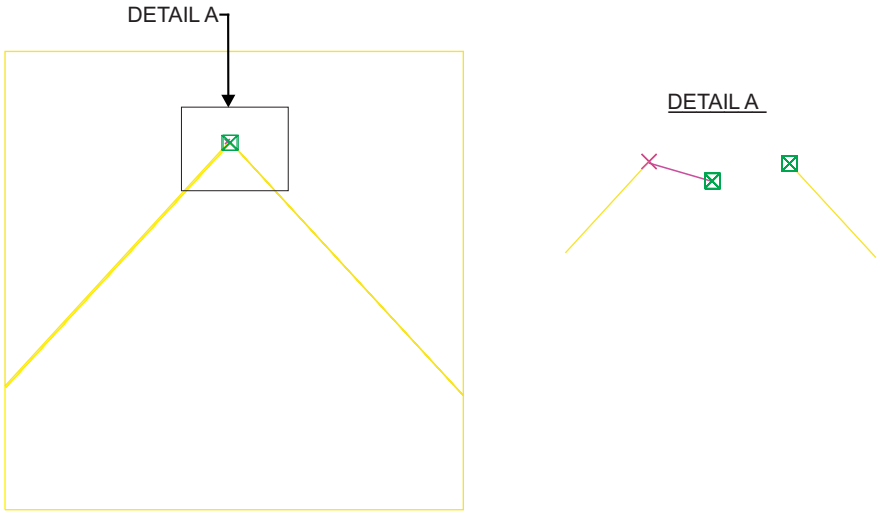

Figure 2-103: Magenta Lines and Points Showing Edges that are Too Short

Blue rectangles: end points of open loops.

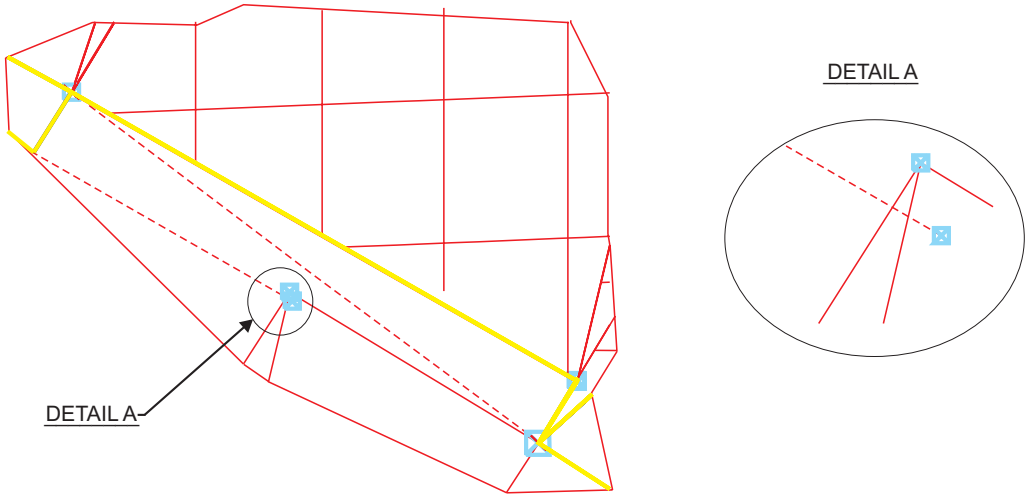

Figure 2-104: Blue Rectangles Showing End Points of Open Loops

Dark blue rectangles: self-intersecting faces.

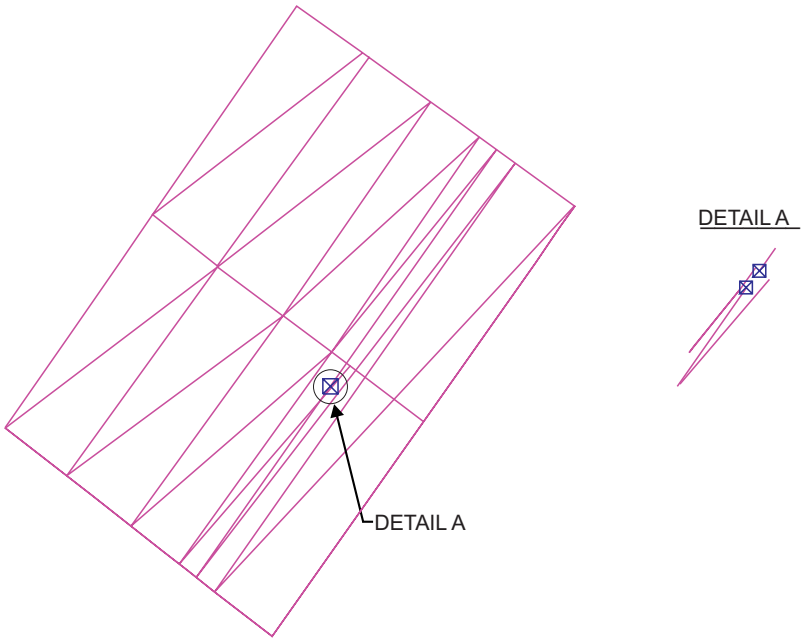

Figure 2-105: Dark Blue Rectangles Showing Self-Intersecting Faces

Green rectangles: the mismatch between end points of corresponding edges is larger than the object tolerance, but less than 10 x tolerance.

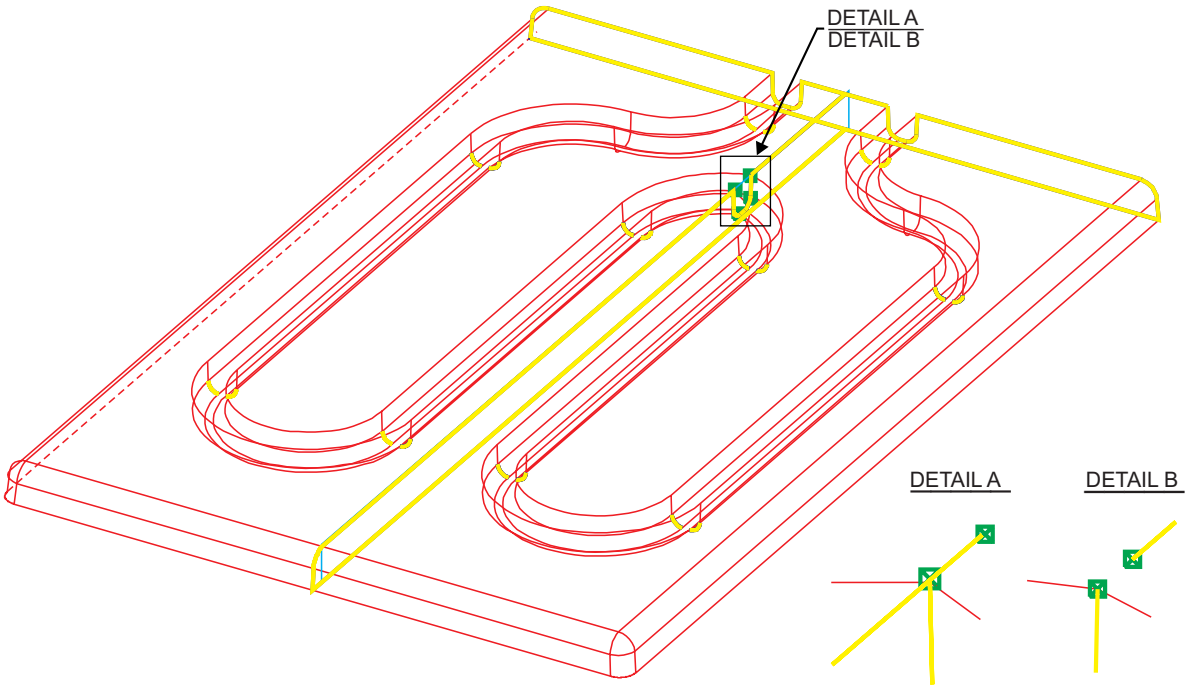

Figure 2-106: Green Rectangles Showing Mismatch

Magenta loop: degenerated loop.

If you see a red loop (loop with a wrong direction) or a magenta loop, (degenerated loop), the object is not correct. These faces must be repaired. Use DETAIL >> DELETE FACE, then rebuild the faces using SURFACES or other tools in the Solid environment.

Dark blue rectangles at the vertices indicate faces with self-intersecting loops. The object is correct, but some problems can occur during geometric operations such as Boolean operations, Round, and Shell. These faces should be rebuilt.

Green rectangles at the vertices indicate that two end points of corresponding edges are considered as one vertex. No edge was not built between these end points. The object is correct, but this inaccuracy may affect the results of Boolean operations or Rounds through and close to these edges.

The best way to fix these problems is to trim corresponding surfaces more accurately in the Wireframe environment, then replace solid faces by new trimmed surfaces.

4. Repair open solids

The healing functions listed on page 2-141 can be used for closing gaps within open solid objects and between objects. The following are suggested methods for various types of repairs.

To decrease the number of gaps:

1. Try to stitch all objects by using DETAIL >> STITCH, selecting all.

2. If you still have a large number of objects, try stitching between each pair of neighboring objects.

To close large planar areas, or nearly-planar open areas:

Use SURFACE >> REGION, with the option LOOP.

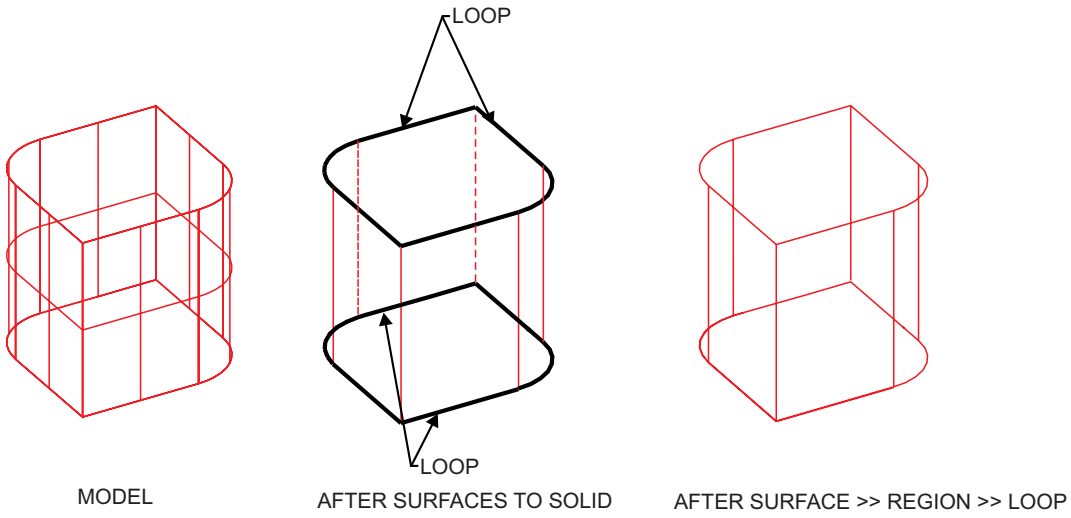

Figure 2-107: Closing Areas Using SURFACE >> REGION >> LOOPS

To close open areas with 3 or 4 boundaries:

1. Use SURFACE >> BLEND with or without boundaries, with or without slope control.

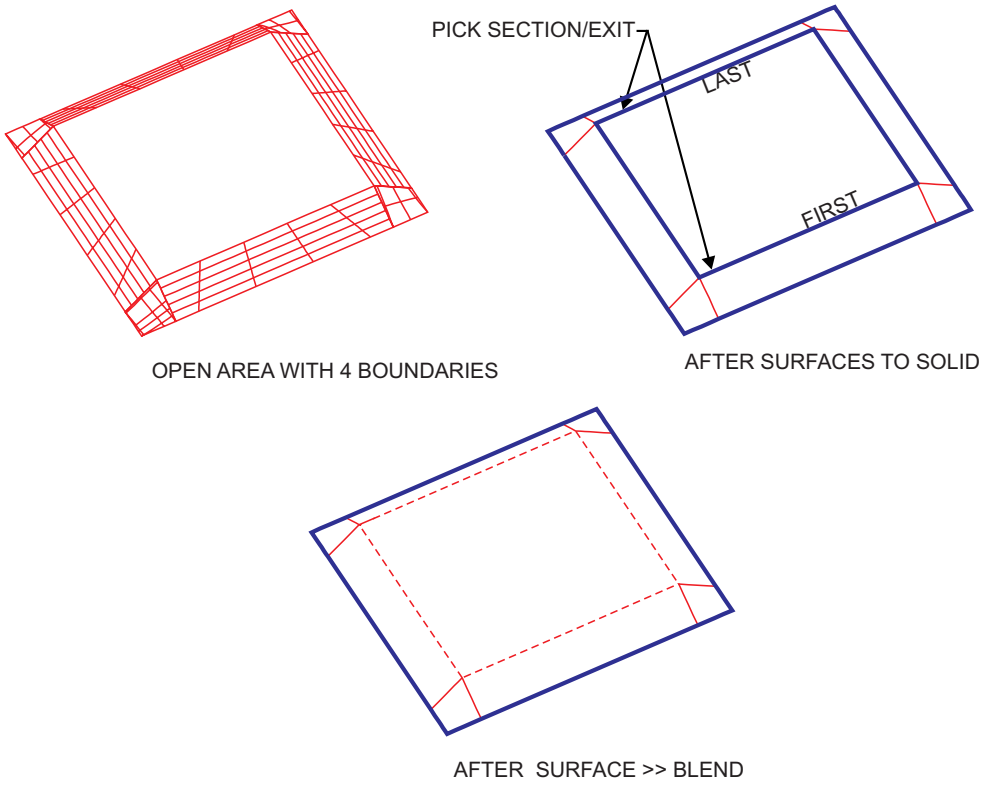

Figure 2-108: Closing Open Areas Using SURFACE >> BLEND

2. Use SURFACE >> REGION with the option LOOP. Note that a REGION surface is created without slope control to neighboring faces.

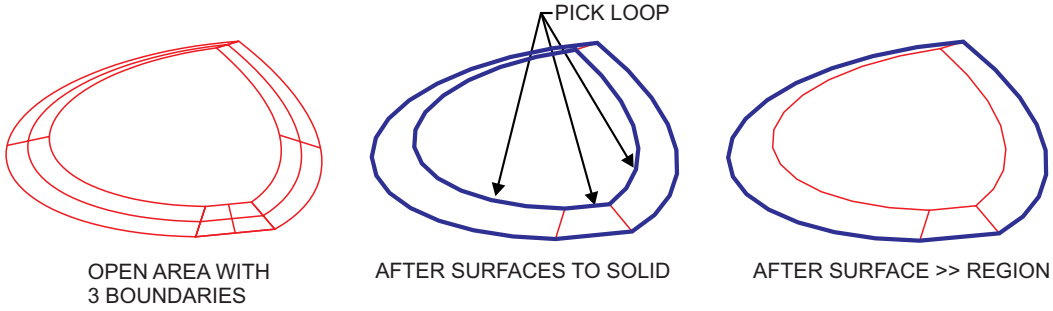

Figure 2-109: Closing Open Areas Using SURFACE >> REGION

To close open areas with 3 or 4 boundaries with break edges:

If one or more boundaries surrounding the open area consist of 2 or more edges, use MODIFY >> BREAK EDGE to divide the opposite edge into 2 or more edges.

Then use SURFACE >> BLEND or SURFACE >> DRIVE for every sequence of corresponding edges.

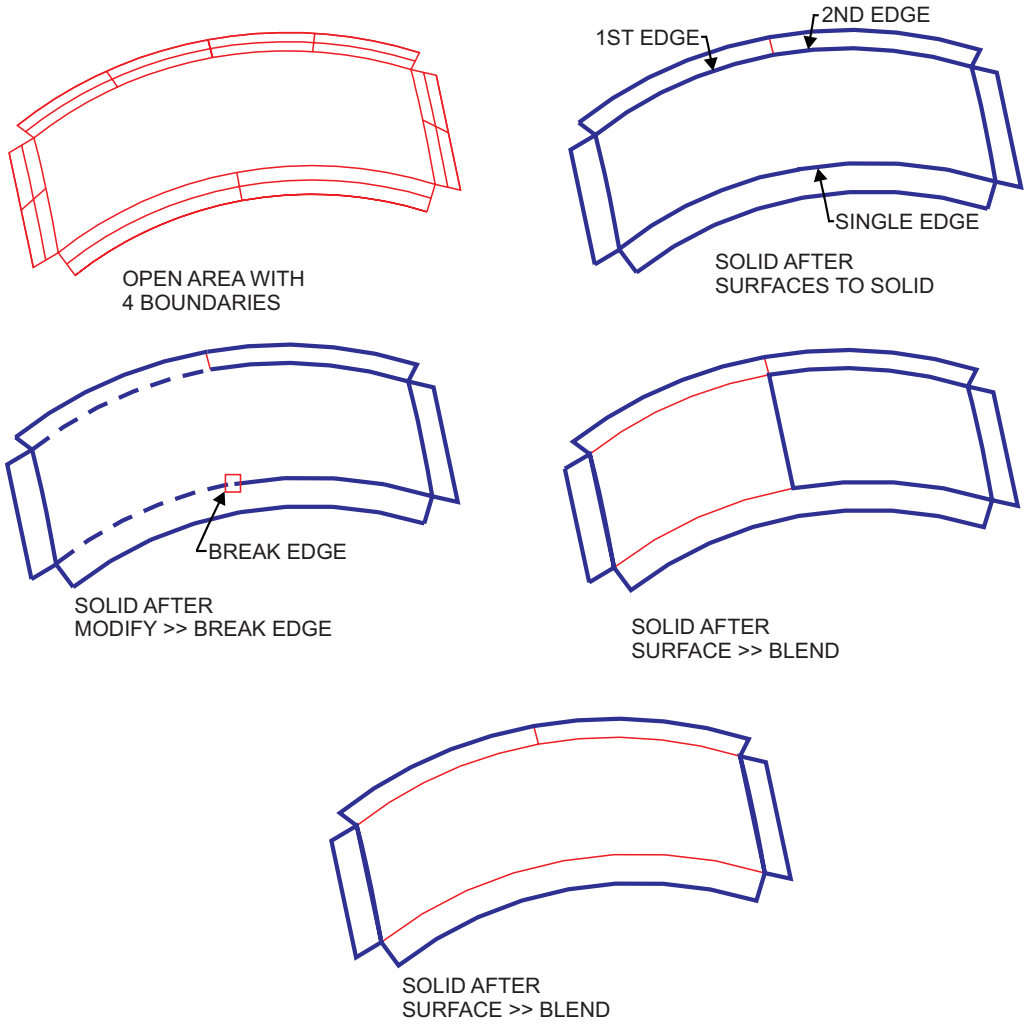

Figure 2-110: Closing Open Areas Using SURFACE >> BLEND

To close open areas bounded on 2 sides:

1. Use SURFACE >> BLEND or SURFACE >> REGION if the area is not too narrow.
2. Use MODIFY >> ADJOIN FACE for narrow gaps.

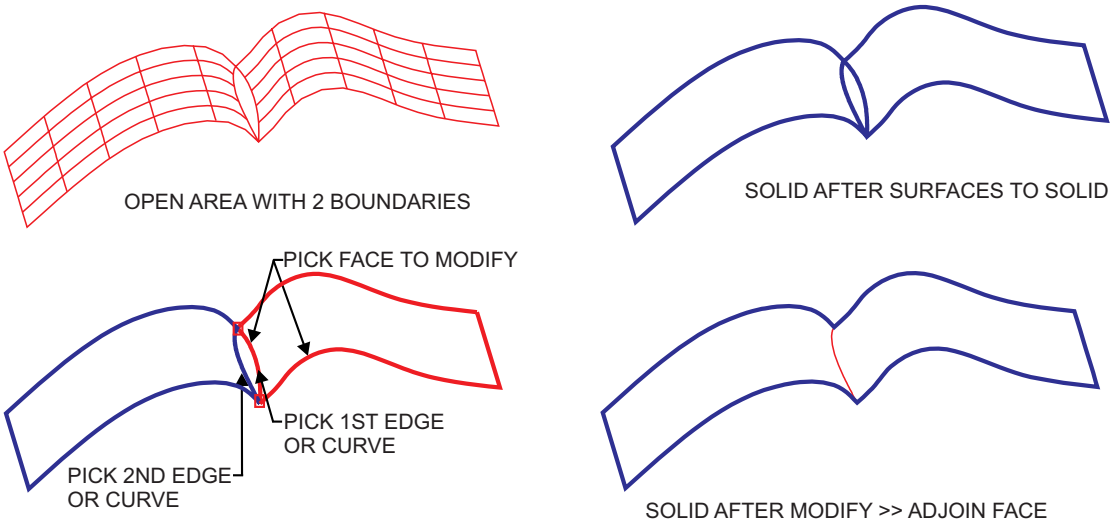

Figure 2-111: Closing Open Areas Using MODIFY >> ADJOIN FACE

Important: corresponding vertices of 2 faces must coincide.

Try the first face for modification; if that does not succeed, try the second face.

Use MODIFY >> ADJOIN FACE to fix overlaps between faces.

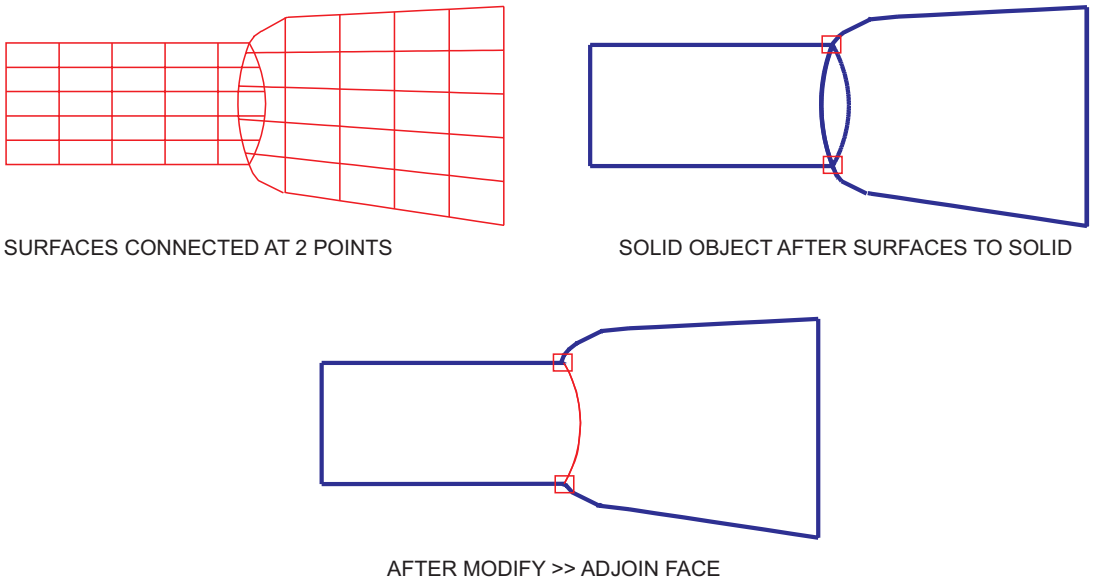

Figure 2-112: Fixing Overlaps Using MODIFY >> ADJOIN FACE

To close open areas when vertices do not coincide:

Use MODIFY >> ADJOIN >> BOUNDARIES on the original surfaces in the Wireframe environment.

Use options GLOBAL or LOCAL to define an area of modification.

Replace the face by the modified surface as follows:

Use DETAIL >> DELETE FACE in the Solid environment, TRANSL >> SURFACES TO SOLID

Then use DETAIL >> STITCH for stitching the new object to the existing object.

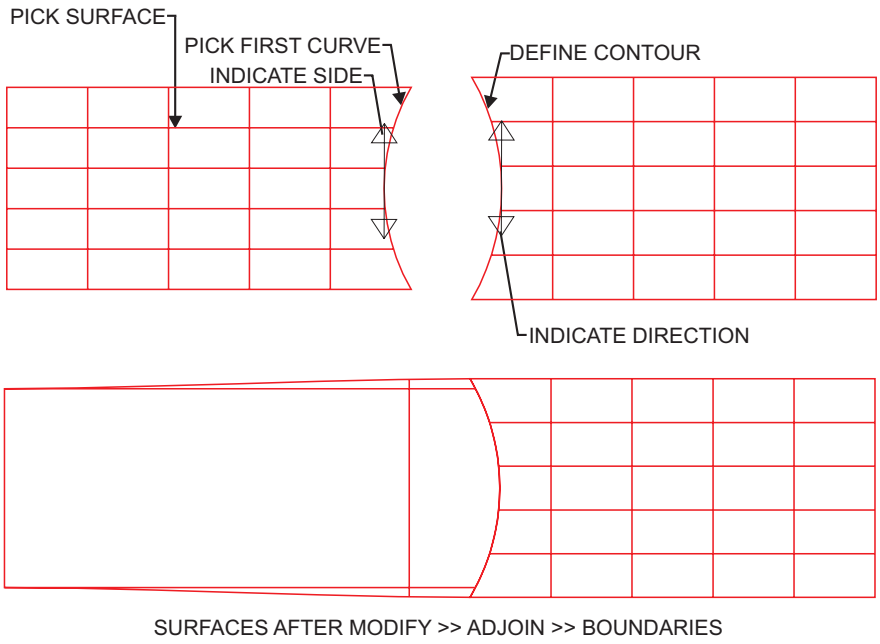

Figure 2-113: Fixing Open Areas Using MODIFY >> ADJOIN >> BOUNDARIES

To remove overlaps which cannot be fixed by ADJOIN FACE:

Use TRMSRF to trim one of the surfaces and join it to the neighboring surface. Then repeat the steps above for replacing a face by a modified surface.

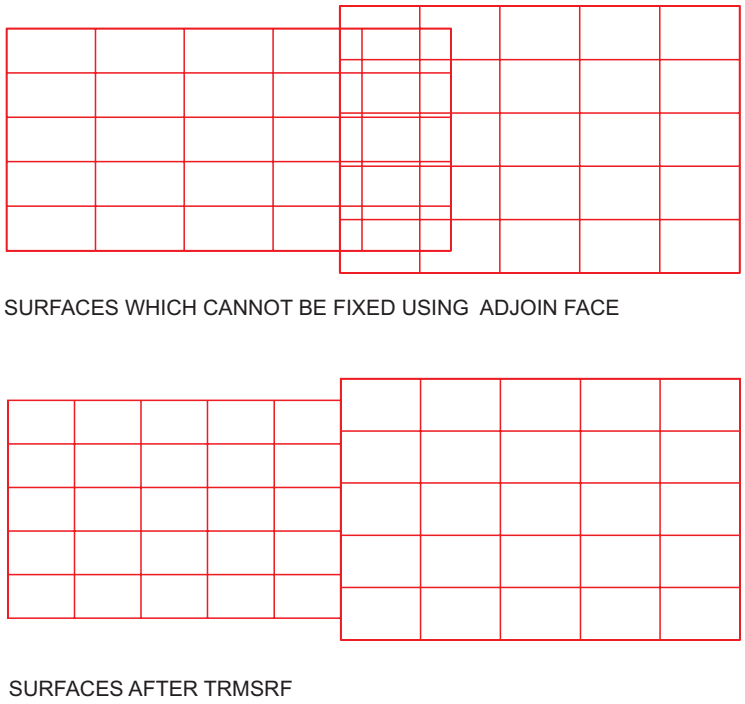

Figure 2-114: Removing Overlaps Using TRMSRF

SURFACES TO SOLID Limitations

1. When an open solid object is created by SURFACE TO SOLID, verify its external normal using MODIFY >> INVERT. Invert the object if the arrow is directed inward.
2. If at least one solid object is created, the object tolerance value cannot be changed in SURFACE TO SOLID.

   For this reason, it is not recommended to change OBJECT TOLERANCE in UTILITY >> SET TOLERANCE. All new objects will be created with the new tolerance, but the first object created by SURFACE TO SOLID will not be changed.
3. The procedure passes regeneration, but the model is not recalculated from the beginning.
4. The SURFACE TO SOLID procedure cannot be deleted if it is the first procedure in the file. Use UNDO or REPLAY and TRIM in order to delete it.

TRANSL >> SOLID -> WF

Create wireframe entities from a solid geometry.

The following options are available:

EDGES
FACES
CURVES
POINTS

SOLID -> WF >> EDGES

Create wireframe curves at solid edges.

<CR> TO CONTINUE	ALL/SINGLE

SOLID -> WF >> EDGES >> ALL

Create curves at all the edges of the solid model.

SOLID -> WF >> EDGES >> SINGLE

<PICK> an edge to be created as an entity in wireframe mode.

SOLID -> WF >> FACES

Create wireframe surfaces at solid faces.

<CR> TO CONTINUE	ALL/SINGLE

SOLID -> WF >> FACES >> ALL

Create surfaces/planar faces from all faces of the solid model.

<CR> TO CONTINUE	PLANAR SURF

PLANAR SURF All solid planar faces will be created as wireframe planar surfaces.

PLANAR FACE All solid planar faces will be created as wireframe planar faces.

SOLID -> WF >> FACES >> SINGLE

<PICK> a face to become a surface/planar face in wireframe mode. If the picked face is a planar face, the following options are displayed:

SELECT	PLANAR SURF	PLANAR FACE

PLANAR SURF The face will be created as a wireframe planar surface.

PLANAR FACE The face will be created as wireframe planar face.

SOLID -> WF >> CURVES

Create wireframe curves at solid edges.

<CR> TO CONTINUE	ALL/SINGLE

SOLID -> WF >> CURVES >> ALL

Create curves from all the reference curves of the solid model.

SOLID -> WF >> CURVES >> SINGLE

<PICK> a reference curve to become an entity in wireframe mode.

SOLID -> WF >> POINTS

Create wireframe solid model at points.

<CR> TO CONTINUE	ALL/SINGLE

SOLID -> WF >> POINTS >> ALL

Create points from all the reference points of the solid model.

SOLID -> WF >> POINTS >> SINGLE

Pick a reference point to be created as an entity in wireframe mode.

TRANSL: Modal Parameter Definitions

- AUTO STITCH — Perform automatic stitching for all solid objects created by SURFACES TO SOLID.
- CANCEL — Cancel the conversion process.
- CONTINUE — Continue the conversion of surfaces to solid objects.
- DELETE SURFACES — Delete original surfaces.
- OBJECT TOL=0.001 — Solid tolerance.
- KEEP SURFACES — Retain original surfaces.
- PARTIAL RESULTS — Display the results so far of the conversion of surfaces to solid objects.
- WITH RECOMMEND. TOL — Recommend a minimum tolerance. Based on the surfaces of the model and the user-defined OBJECT TOLERANCE, the system analyses the gaps and the small edges and displays a message regarding the recommended tolerance.

  Using this parameter, Cimatron checks within a range between the tolerance entered and a tolerance ten times its value (this prevents the system from recommending changes in tolerances that are too large, such as between two faces that are far apart).
- WITHOUT RECOMMEND. TOL — Do not recommend a minimum tolerance.

■ WITHOUT STITCH — Perform the conversion to solid objects by using SURFACES TO SOLID, but do not stitch the objects after conversion.

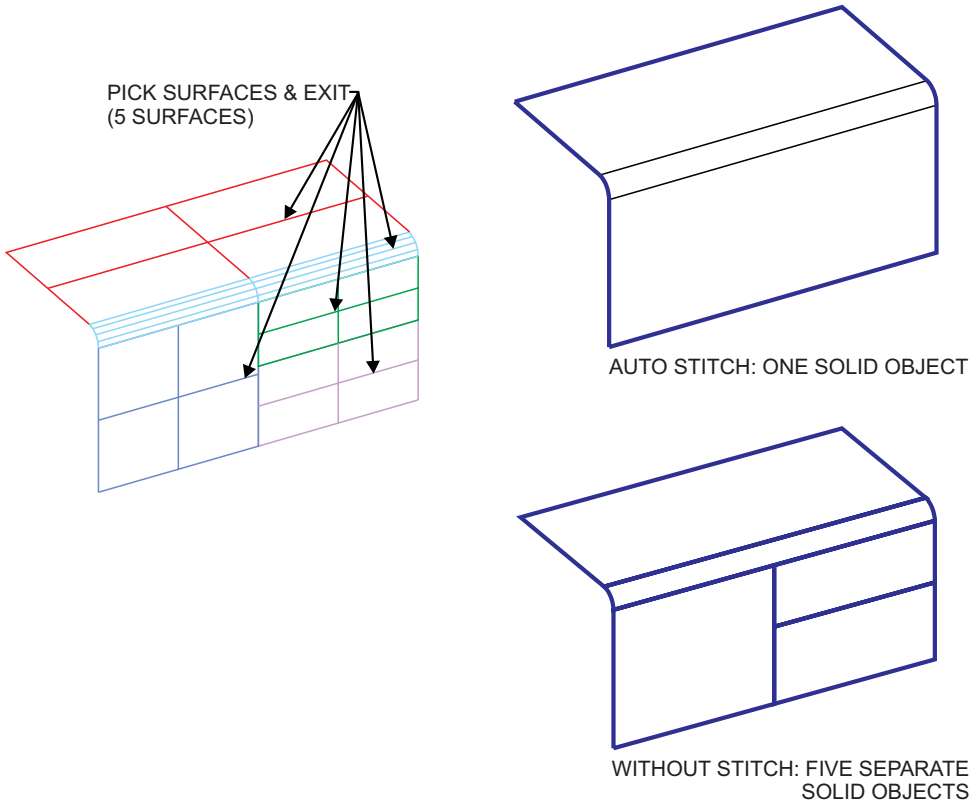

Figure 2-115: AUTO STITCH, WITHOUT STITCH

TRANSL: Usage Envelope

1. Try to remember the tolerance you used while creating the geometric model. The same applies for the models received from IGES or VDA. It will be useful when you have to decide what tolerance to use in the SURF >> SOLID option of the function TRANSL.
2. If possible, use a smaller tolerance value in the original system where the geometric model has been created. We recommend using planar faces (instead of trimmed planar surfaces) as it is helpful for the function stability and precision.
3. Problematic surfaces marked in red should be redefined. In most cases these are incorrectly trimmed surfaces that have not been converted to solid faces.
4. Small surfaces are displayed only for informative purposes. Those areas where small surfaces are found are closed with neighboring faces.
5. Closed surfaces are divided automatically by parameter as in the function USER >> GEOMETRY >> DIVTRS.
6. If the SURF >> SOLID option of the function TRANSL is used and at least one solid object is created, the object tolerance value cannot be changed. Only use the function UTILITY >> SET TOLERANCE and change OBJECT TOLERANCE.
7. It is strongly recommended to check the objects created in the SURF >> SOLID option of the function TRANSL using UTILITY >> CHECK SOLID.
8. Use the function MODIFY >> INVERT to invert an open solid created on the opposite side.
9. In cases where incorrect trimmed surfaces were found, try to use the function MODIFY >> FAIR with a small tolerance before using the SURF >> SOLID option of the function TRANSL.
10. If you get gaps between solid faces, we recommend the following solutions:

    SURFACE >> REGION to close gaps.

    DELETE FACE + SURFACE >> BLEND or SURFACE >> MESH to create another face instead.

    DETAIL >> STITCH for manual merging of objects. ❑

UTILITY

A collection of SOLID utilities.

Main Options:

RESIZE
SET TOLERANCE
CHECK SOLID
REGENERATE

RESIZE Resize a datum plane or axis.

SET TOLERANCE Set the tolerance of the solid object.

CHECK SOLID Check the validity of solid objects and detect incorrect geometry.

REGENERATE Replay each step in the solid generation.

UTILITY >> RESIZE

Resize a datum plane or axis.

PICK PLANE/AXIS <Pick> the plane or axis to be resized.

RESIZE AXIS or RESIZE PLANE Resize the axis or plane by clicking and dragging an end or a corner. <EXIT> when finished.

UTILITY >> SET TOLERANCE

Set the tolerance of the solid object.

<CR> TO CONTINUE	OBJECT TOL=0.01

Set the tolerances and press <CR>.

OK TO REGEN? Y / N

YES Regenerate the part. The following message is displayed: THE PART MUST BE REGENERATED FROM THE BEGINNING.

NO Exit the function.

UTILITY >> CHECK SOLID

Check the topology and geometry of the solid object.

SELECT	ALL OBJECTS	SINGLE OBJECTS

Select whether all objects or single objects are to be checked.

If the topology and geometry checks are passed successfully, an appropriate message is displayed.

If problems are found, the results of the analysis are displayed as follows:

— Green vertexes: gaps that are bigger than the specified object tolerance.

— Blue loops: gaps that are bigger than ten times the specified object tolerance.

— Red loops: loops with the wrong direction.

— Yellow edges: edges of open faces.

UTILITY >> REGENERATE

Regenerate each step in the solid generation process.

<Step name>	|<——	<——	——>>	——>>|	JUMP

Note: • For old files, step-by-step regeneration is not available.

UTILITY: Modal Parameter Definitions

- \|<— Go to the beginning of the solid generation.
- <—— Go back one step.
- ——> Go forward one step.
- ——>| Go to the last step.
- ALL OBJECTS Check the topology and geometry of all objects.
- DISPLAY TOL=0.250 Solid display tolerance.
- JUMP <PICK> an entity. All steps up to and including the creation of the picked entity will be displayed.
- OBJECT TOL=0.01 The maximum distance between two adjacent faces that are considered to be intersecting.
- SET DEFAULTS Reset the tolerances to the system default values. The system defaults will appear in the modal fields.
- SINGLE OBJECTS Check the topology and geometry of single objects. ❑

Chapter 3
General Functions in the PART Environment

Introduction

The general function menu, as shown below, can be used with **Solid** Modeling as with other **Cimatron** functions.

Overlay I
UNDO
DELETE
BLANK
LEVELS
LINATT
UCS
FILE
WINDOW
VERIFY
EXIT

Overlay II
USER
DISPLAY
DIGITIZR
PLOT
IMAGE
ATRBUTES
SHADE
ANALYZE
REGISTER
EXIT

The option ANALYZE >> SOLID VOLUME is specific to the SOLID Modeling Part Environment. This function calculates the volume, surface area and center of gravity of the solid object.

The color of solid objects and datums can be changed using the LINATT function.

The option FILE >> TYPE is specific to the SOLID Modeling Part Environment. This option saves a file as a library file.

The option VERIFY >> SOLID is specific to the SOLID Modeling Part Environment.

The option SHADE >> DEFINE SECTION is also specific to SOLID Modeling Part Environment. This function allows you to set the depth of a clipping plane to display sections of a solid object.

The BLANK function is applicable to solid objects. For more details, see the General Service Functions chapter in the **Fundamentals and General Functions Manual**.

The LEVELS >> MOVE and LEVELS >> ACTIVE options are applicable to solid objects (solids and surfaces).

ANALYZE

Calculate area, surface area, center of gravity, moment of inertia and volume.

Main Options

SELECT:	AREA
	SWEPT VOLUME
	SURFACE AREA
	SURFACE VOLUME
	SOLID VOLUME

AREA Calculate the area, center of gravity, and moment of inertia for 2D closed contours and allow islands.

SWEPT VOLUME Calculate volume and center of gravity for extruded bodies created by a linear sweep, and allow for holes.

SURFACE AREA Calculate the area of a surface.

SURFACE VOLUME Calculate volume of 3D spaces enclosed by surfaces or one part of the space which has been cut by a plane.

SOLID VOLUME Calculate volume, surface area, moments of inertia and center of gravity for a solid object.

A description of the option SOLID VOLUME follows. For full descriptions of the remaining options, refer to the function ANALYZE in the **Fundamentals and General Functions Manual**.

Notes:

- Results may be converted to smaller or larger units of measure in the unit type of the part only (in metric or in feet and inches).
- Modals preceded by * are system-generated and may not be modified.

ANALYZE >> SOLID VOLUME

Calculate volume, surface area, moments of inertia and center of gravity for a solid object.

<CR> TO CONTINUE	TOL = 0.100	DENSITY = 1.000	DISPLAY C.G.

<CR> TO CONTINUE <CR> to activate the calculation.

EXECUTING Volume, surface area and center of gravity are calculated.

<CR> TO CONTINUE	MM	*VOLUME = 0.000	*SRF AREA = 0.000
	*XCG = 0.00	*YCG = 0.000	*ZCG = 0.000
	IXX/CG = 0.00	IYY/CG = 0.00	IZZ/CG = 0.00
	IXX/UCS = 0.00	IYY/UCS = 0.00	IZZ/UCS = 0.00
	IXY/CG = 0.00	IYZ/CG = 0.00	IZX/CG = 0.00
	IXY/CG = 0.00	IYZ/UCS = 0.00	IZX/UCS = 0.00

<CR> TO CONTINUE <CR> when finished viewing results.

ANALYZE: Modal Parameter Definitions

- DENSITY — Specify the density of the solid object.
- DISPLAY C.G.
  DO NOT DISPLAY C.G. — Set modal for the display or non-display of the center of gravity symbol, a circle with an X in its center.

  The symbol appears at the center of gravity, on a plane parallel to the active XY plane. It may be deleted with the DELETE function.
- INCH
  FOOT — If the active unit of measure is feet or inches, these modals will appear.
- IXX/CG, IYY/CG, IZZ/CG
  IXY/CG, IYZ,/CG, IZX/CG — Values of moments of inertia relative to center of gravity.
- IXX/UCS, IYY/UCS, IZZ/UCS
  XY/UCS, IYZ,/UCS, IZX/UCS

  Values of moments of inertia relative to the UCS.
- MM
  CM
  METER — If the active unit of measure is metric, these modals will appear.
- * SRF AREA = 0.000 — Total surface area in the active unit of measure, of the solid object.
- TOL = 0.100 — The maximum distance that will be tolerated between the contour and the line segments used to approximate the contour.
- * VOLUME = 0.000 — Total volume in the active unit of measurement, of the solid object.
- * XCG, * YCG, * ZCG = 0.000

  X, Y, Z values of the center of gravity in the UCS respectively.

❑

FILE

Save the part with all changes made during the current session and continue working on the same part.

SELECT	SAVE PART
	COMP FILE
	COMP LIST
	PARAMETERS
	FILE SIZE
	COMPRESS
	TYPE
	SETUP

SAVE PART Save the part file with all changes made in the current session.

COMP FILE Create a file which shows the composition of the current part file or ASSEMBLY file.

COMP LIST Create a list (in a file or on the screen) showing the composition of specific entities. For a detailed description of this function, see Chapter 6 of the **Fundamentals and General Functions Manual**.

PARAMETERS Recreate the **xxparam** file based on the settings for the current part.

FILE SIZE Inquire from the system how much disk space is required to save the current part. For a detailed description of this function, see Chapter 6 of the **Fundamentals and General Functions Manual**.

COMPRESS Compress **Cimatron** files to make disk space available. For a detailed description of this function, see Chapter 6 of the **Fundamentals and General Functions Manual**.

TYPE Save the file as a STANDARD file.

SETUP Change the **Cimatron** setup parameters.

For full descriptions of all the options in a regular PART environment, refer to the **Fundamentals and General Functions Manual**.

FILE >> TYPE

A file type may be standard or non-standard (default). A standard file may appear in several assemblies. A non-standard file may be attached to only one assembly.

Note:

- A non-standard file attached to an assembly must be copied in order to use it in another assembly.

<CR> TO CONTINUE	NON-STANDARD	<attachment>
	STANDARD	

The <attachment> can be one of the following:

UNATTACHED
ATTACHED TO ASSEMBLY <assembly name>
ATTACHED, WITH DEPENDENCIES, TO ASSEMBLY <assembly name>
ATTACHED WITHOUT DEPENDENCY

In the case of parts that are UNATTACHED or ATTACHED WITHOUT DEPENDENCY, the option to switch to Standard is presented in the following manner:

MODIFY TO STANDARD ? YES / NO

Note:

- A STANDARD part cannot be restored to a NON-STANDARD part.

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SHADE

Shade solid objects to display a model realistically, display sections.

Each solid is automatically divided into quadrilateral planar facet. Each facet is painted a different color, or filled in with a different dot pattern according to its orientation in space relative to the viewer.

This function operates differently on color monitors and monochrome or gray scale monitors.

Main Options

SELECT:	PREPARE SURFACES
	COLORS
	MATERIALS
	MODIFY LIGHTS
	LIGHT SWITCH
	DEFINE SECTION

A description of the option DEFINE SECTION follows. For full descriptions of the remaining options, refer to the function SHADE in the **Fundamentals and General Functions Manual**.

SHADE >> DEFINE SECTION

Set the depth of a clipping plane to display sections of a solid object in the specified color. The clipping plane may be moved in or out, but always remains parallel to the screen.

SELECT	IN	OUT	STEP = 10.000	DEPTH = 0.000	<color>

SHADE: Modal Parameter Definitions

- <color> — Color of the section.
- DEPTH = 1.000 — Depth of the clipping plane (the screen is located at a depth of 0.000).
- IN — Move the clipping plane inwards (from the screen towards the part) by the specified step increment.
- OUT — Move the clipping plane outwards (from the part to the screen) by the specified step increment.
- STEP = 10.000 — Increment by which the clipping plane is moved per step.

❑

VERIFY

Calculate, examine or modify data which relates to points, curves and surfaces.

Main Options:

SELECT:	**GENERAL**
	SURFACES
	SOLID

GENERAL Calculate, examine or modify the data which relates to point and curve entities.

SURFACES Display or modify the data defining existing surfaces.

SOLID Display data defining existing solid features, and check for geometric problems.

A description of the option SOLID follows. For full descriptions of the remaining options, refer to the function VERIFY in Chapter 7 of the **Fundamentals and General Functions Manual**.

VERIFY >> SOLID

Display data defining existing solid features, and check for geometric problems.

SELECT OPTION	OBJECT
	FEATURE
	FACE/SURFACE
	DATUM
	UNDERCUT

OBJECT Receive general information regarding a picked object and check its topology/geometry.

FEATURE For a selected feature, receive the feature name, the options used to create it and all feature references.

FACE/SURFACE Display a surface type and offset.

DATUM Receive general information regarding the depth and normal vector of a datum plane. Display start point coordinates and the direction of an axis.

UNDERCUT Check the faces of models, cores, cavities and various kinds of inserts to detect undercuts on them.

SOLID >> OBJECT

Receive general information regarding a picked object and check its topology/ geometry.

PICK OBJECT <PICK> anywhere on the object.

<CR> TO CONTINUE	CLOSED OBJECT	LEVEL=<level_name>	COLOR=1

<CR> TO CONTINUE Press <CR> to continue.

SELECT OPTION	CHECK OBJECT
	DRAW OPEN EDGES

CHECK OBJECT Check the geometry and topology of the object.

DRAW OPEN EDGES Indicate places where open edges occur.

SOLID >> OBJECT >> CHECK OBJECT

The geometry and topology of the object are checked.

If no errors are found, the following message is displayed:

TOPOLOGY AND GEOMETRY CHECKING PASSED SUCCESSFULLY

If a problem occurs during checking, the following message appears:

PROBLEMATIC TOPOLOGY / GEOMETRY FOUND

All problematic areas are highlighted and information about each number and color-coded problem is displayed. Below is a sample menu of a problematic object:

<CR> TO CONTINUE	Attention! 46 loop(s) with gaps! (green points)
	Attention! 2 open loop(s)! (blue points)
	Attention! 2 self intersected face(s) dark blue points

SOLID >> FEATURE

For a selected feature, receive the feature name, the options used to create it and all feature references.

SELECT FEATURE/EXIT <PICK> the feature on the object or right-click to display the feature list and then select the appropriate feature from the list.

Depending on the type of feature picked, a number of different parameters are displayed:

NEW/ADD/ REMOVE/DATUM/DIVID Feature creation method.

SKETCH. REF Indicate the reference plane and display its feature description.

DELTA Display the delta dimension.

REFERENCE PLANE Highlight the reference plane and display its feature description.

OTHER REFERENCES Highlight other references (faces, curves, etc.) and display their feature descriptions.

FROM Highlight the source plane.

TO Highlight the target plane.

REFERENCE Display the reference entities used to create the plane.

Right-clicking displays a table of features each of which can be picked to view their specific parameters. Below is an example of such a table.

	EXTRUDE9		EXTRUDE10		DRAFT11		DRAFT13
	ROUND14		ROUND15		ROUND16		ROUND17
	MOVE18		MOVE19		MOVE20		MOVE21
	MOVE22		MOVE24		MOVE26		MOVE27
	MOVE28		EXTRUDE35		AXIS36		PLANE37

If you pick a specific feature, Cimatron highlights this feature on the object and displays its parameters.

SOLID >> FACE/SURFACE

Display a surface type and offset.

PICK FACE/SURFACE <PICK> required face or surface.

If a planar face is picked:

PLANAR FACE	Xn = 0.000	Yn = 0.000	Zn = -1.000
	DEPTH = -148.402		

PLANAR FACE The X, Y, Z dimensions and depth are displayed.

If a surface is picked, the surface type and offset are displayed.

SOLID >> DATUM

Receive general information regarding the depth and normal vector of a datum plane. Display start point coordinates and the direction of an axis.

PICK PLANE OR AXIS <PICK> plane or axis of object.

PLANE Display the X, Y, Z dimensions, direction and depth.

AXIS Display start point coordinates and the direction of an axis.

SOLID >> UNDERCUT

Check the faces of models, cores, cavities and various kinds of inserts to detect undercuts on them. This check will avoid problems during the opening of the mold and exiting parts from it.

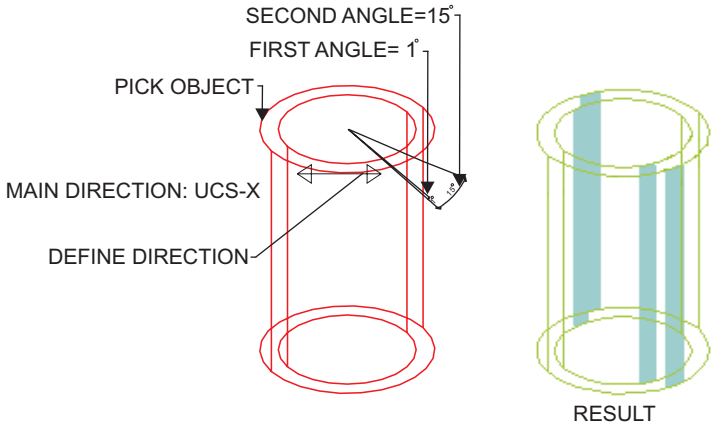

Figure 3-1: SOLID >> UNDERCUT

How To:

1. Enter the range of angles which must be mapped on the faces.
2. Choose a color for the result.
3. Pick the object to be tested.
4. Select the direction of the current opening using <SUBMENU> options.

Once you have entered the table parameters and selected the object and direction, the result will then be calculated and all regions within the angle range will be shaded on the body.

Angles for undercut checking are calculated by the following rule: an angle of 0° corresponds to any vector that is perpendicular to the vector of the current opening. The vector corresponding to the direction of the current opening has a value of +90°. Enter angles within the range of -90° to +90°.

After receiving the initial results, you can change the range of the angles and receive new results, without losing the original results. Therefore, after calculating several results, you can create a map of the different slopes of faces.

If a face that is in contact with plastic has a result for a positive range of the angles, this indicates that an undercut occurs in that specific place. This problem should be prevented.

If you only entered one angle value in each of the two fields, faces which have the same slope, according to direction, will be shaded. This option is useful for checking faces of the model or core and cavity, to detect faces which already have a draft

angle. For example, if you enter +0.5° for the first angle and -0.5° for the second, the result will be faces which have this draft angle.

Interaction:

PICK OBJECT	FIRST ANGLE = 0.000	SECOND ANGLE = 0.000
	REFINE FACTOR	<Color bar>

PICK OBJECT Pick the object to be tested.

DIRECTION: PICK CURVE Pick the curve to define the direction.

INDICATE DIRECTION Indicate the direction.

VERIFY: SOLID Modal Parameter Definitions

- COLOR = . . . — Color number of the entity.
- DELTA = . . . — Delta parameter/Delta angle formed by the arc/circle/conic section in degrees.
- DEPTH — Distance of a planar face / plane from the UCS.
- FIRST ANGLE = — The first of two angles defining a range of angles for UNDERCUT.
- FROM — Source extrusion plane.
- LEVEL = . . . — The level number to which the entity belongs.
- OFFSET — Offset distance.
- REFERENCE PLANE — Reference plane.
- REFINE FACTOR = — Refine the approximation. The higher the value, the greater the resolution.
- SECOND ANGLE = — The second of two angles defining a range of angles for UNDERCUT.
- SKETCH REF. — Indicate reference plane.
- TO — Target extrusion plane.
- Xdir, Ydir, Zdir = ... — Lengths of the x, y, and z components of the unit vector that defines the direction of an axis.
- Xn, Yn, Zn = ... — Lengths of the x, y, and z components of the unit vector that defines the direction of the normal to a planar face / plane.
- Xor, Yor, Zor = ... — Coordinates of the axis origin point.

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Chapter 4
Assembly

Introduction

In the assembly environment, the user (or group of users) can combine parts. After assembling the parts, they can be edited, and features/objects can be added/deleted. The user can manipulate parts in the assembly, fit them together, and use reference geometry from the assembly to create/edit a different part. The assembly environment allows you to use both bottom-up and top-down design.

The user can also use the assembly verify options to measure distances, check interference between parts, etc.

Assembly operations can be performed on solid parts and solid assemblies within the assembly environment. This environment is accessed by adding a flag (**-apenv**) when invoking **Cimatron**.

Assembly File Definition

The assembly environment produces an assembly file <filename>.pfm. Changes made to a solid part will also appear in all subassemblies that contain this part, while maintaining all placement relations defined in the assembly.

Note:

- Erasing or renaming a part file <filename>.pfm, or moving it to another directory, results in an error message when opening an assembly file in which it is included.
- To save all the changes made to an assembly, you must save the main assembly file.

Assembly Structure

The assembly is comprised of parts and/or subassemblies that are assembled according to placement dependencies. The first component UCS in the assembly, the base component and the assembly UCS coincide.

Each component is fetched to the assembly, and it is attached to the assembly by planes, axes or points taken from the assembly or a coordinate system taken from other assembly components. Therefore, each component, except the base component, are **placement dependent** on other components.

Notes:

- If part A is place dependent on part B, then it is also place dependent on all the parts on which part B is dependent.
- Placement loops are forbidden. If part A is place dependent on part B, then part B may not be place dependent on part A.

The assembly environment also supports two other types of feature dependency. A feature in the assembly part may have **Geometric** or **Parametric** dependencies, subject to the following limitations:

Geometric

When working in Assembly-Mode (PART >> NEW or PART >> OPEN), features created or modified can use reference geometry from other assembly parts. Such a feature can be geometrically dependent upon that part from which the geometry came. A part with geometrical dependent features is geometrically dependent.

Parametric

The parameter(s) of the feature can be related to parameter(s) of other parts. Parametric relations take the form of an equation, defined using EDIT-RELATIONS.

If a part contains a feature which is parametrically dependent, the part is parametrically dependent.

Parametric relations are uni-directional (in contrast to the relations in a free part). This means that only the parameter which lies on the left side of the equation is dependent.

Dependency Notes:

- If part A is parametrically dependent on part B, then it is also parametrically dependent on all the parts on which part B is parametrically dependent.
- Parametric loops are forbidden. If part A is parametrically dependent on part B, then part B may not be parametrically dependent on part A.
- There is no connection between parametric dependencies and placement/geometric dependencies.
- The child in the relation must be located in the first level of the active assembly.

Assembly Component Types

Each assembly component has its own pfm file.

When fetching a non-standard component that is not attached to any other assembly, it becomes attached to the current assembly.

Components attached to other assemblies may not be fetched into the current assembly.

If the component is a part, then a free copy of that part can be created and fetched instead.

Notes:

- Do not copy or move the pfm file of an attached part using the operating system. The result will be an attached part that cannot be found by the Assembly environment.
- To detach a component from an Assembly, use DISCARD in the Component Menu.
- Use External >> Assembly >> Copy Regular and External >> Assembly >> File Type with Detach options in order to create a free copy to the subassembly.

Standard Assembly Components and Instances

If you need to repeatedly assemble one component, the component can be made standard, or you can assemble instances of the component. Instances can only be used for the currently active assembly.

Standard assembly components have the following properties:

- The components can be used in more than one assembly
- Only attached components without parametric or geometric dependencies can be converted to standard parts.

Hints:

- A part can be converted to a non-standard part by saving it under a different name in the part environment.
- An assembly can be converted to a non-standard assembly by using the External >> Assembly >> File Type. Doing this might could cause damage to your work if the assembly is already in use in other places.

An instance is a component that was assembled by the ASSEMBLE >> PICK, ASSEMBLE >> REPORT option. A change in one instance affects all instances.

Component Alignment

A component may be aligned in any of the following ways:

By Constraints	Align a component relative to other components, by applying a sequence of geometric constraints.
By UCS	(User Coordinate System). Align the component's coordinate system with a coordinate system on the assembly.
By Reference	When a new part is created in an assembly using PART >> NEW, the part is attached to the assembly by reference. The attachment method may be changed using EDIT >> RE-ASSEMBLE.

Assembly Workflow

Work can only be done on the active component. The first active component is the main assembly. you can make another sub assembly active by using the SUB_ASSM function, and a part can be made active by using the PART function.

Commands within the EDIT function can be performed only on the active component.

When making a part active, the user must define whether to display the entire assembly (ASSEMBLY mode) or just the part (FREE PART). If ASSEMBLY mode is chosen as the display, the entire assembly will be visible but you can only work on the active part.

If a subassembly is active, you cannot directly activate another subassembly. First, return to the main assembly (MAIN button), and then activate the next subassembly.

Assembly Regeneration

The MODE function controls and switches between the regular assembly environment (automatic updating of the assembly after every editing operation) and the manual update environment.

The UPDATE option toggles the assembly automatic update flag. The default state of the assembly auto.update flag is ON. This means that when assembly components are changed, the assembly is automatically recalculated and the relations between the assembly components are updated. When the flag is OFF, this recalculation of the main assembly top level is postponed.

When the auto.update flag is OFF, you can manually update the assembly by pressing the MODE button.

To switch between auto.update ON and auto.update OFF, select MODE and choose YES.

Working with the auto.update flag OFF has the following advantages:

- Large and complicated assemblies can be loaded quickly.
- Several designers may work on the same assembly, each working on different components.

  If several designers are working on the same assembly, the combination of UPDATE OFF and the search path rule is recommended.
- Increased speed when work on parts and subassemblies. All part editing tools are available, including interference-check and creating views and sections for checking purposes. The Views and sections created will not be saved unless the auto.update flag is switched ON.

Note:

- When the auto.update flag is OFF, editing the main assembly is not possible, and the main assembly cannot be saved. To save the assembly, the auto update flag must be switched ON.

Search Path Method

Search path rules enable the creation of assembly versions or allow a group of users to work on the same assembly concurrently.

How does it work?

While **Cimatron** is loading, it looks for the **priv_dir.dat** file. If the file exists, the search path rule is turned on.

The search path rule is activated whenever read/write operations are carried out.

While the search path rule is on, **Cimatron** will first search for files in the single user directory. If the files are not found, the group directory is searched. Files are saved **only** to the single user directory.

To update the group directory, **move** the files from the single user directory to the group directory using the operating system (e.g., Explorer). Files are moved to the public directory so that other users may observe your changes. Do not **copy** your file into the group directory because that may cause you to work on a different component than that the main assembly.

The single user directory will only contain files that undergo changes from the assembly loading (see SAVE LIST on page 4-32).

It is recommended that the group files will be in read-only permission until you need to move the new files.

The search path rule also works on any other commands that create files like SESSION, FREE. COMPOSITION FILE, and SAVE LIST.

If you want to return to the original main assembly, delete the single user files by using the operating system.

Implementation

1. Create assembly versions

Change the file **priv_dir.dat** whenever you wish to create a new version of the assembly.

To open a version, specify the relevant priv_dir.dat file.

2. Working in a Group

Each user has a private directory where all changes are saved, but the user can not tell from where the file are read

To enable the rest of the group to see the changes, move your files to the group directory using the operating system (e.g., Explorer).

It is recommended to put the group path in a network, so that all users will call the assembly in the same way.

It is recommended that one user be an assembly administrator and only that user will have write permission in the main assembly. The assembly administrator should not work with the search path rule. It is the responsibility of the assembly administrator to move the files and to save the main assembly in the group directory.

After moving all the files to the main assembly, it is recommended to reload the main assembly (delete non-relevant components from your private directory). To obtain better results, it is recommended that the administrator updates and saves the main assembly. Then, if you add/discard components, the user need only move the parts that he made changes to, or the parts with their assembly.

Note:

- Two users cannot move the same files to the assembly, as the changes of the first user will be deleted by the changes of the second.

3. Create several solutions of the same assembly.

Setup

Create the file **priv_dir.dat**.

The file should be located in the directory **<root>/var/profile/<user_name>** for a single user. For a group of users in the directory **<root>/var/profile/<cimatron/ user define>**, **Cimatron** must be run with the flag -wg.

The priv_dir.dat is text file containing a pairs of path names with a line between them.

In each pair, the first path is the current path of the assembly in the public directory and the second path is a new path where the changed files will be stored in the private directory.

Example: **c:\sub_assembly**

c:\week1_sub_assembly

In this case, files with the path name of c:\sub_assembly_small will be saved as **c:\week1_sub_assembly_small**.

The file can be created by any text editor.

Caution: The search path is a method of replacing strings of text. Be careful that a previous pair of path names will not change the path of the pair you are currently using.

Example: **c:\sub_assembly**

c:\week1_sub_assembly

c:\sub_assembly_1

c:\week1_sub_assembly_9

in this case the file **c:\sub_assembly_1\main.pfm** will be searched for in **c:\week1_sub_assembly_1\main.pfm**.

The single user directory does not need to be constructed the same way as the group directory. However, it is more convenient to work when the private directory tree has the same construction as the public directory.

Since the search path rule does not create a new directory, the user has to build the path before starting work.

Accessing the Assembly Environment

To access the Assembly environment, use the flag **-apenv** when invoking **Cimatron** (i.e. **cim90 -apenv**).

Loading an Existing Assembly File

There are two options relevant to loading existing assembly files available in the FILE >> SETUP options.

ASSEMBLY: UPDATE ON/OFF

ON/OFF The default is ON, unless **Cimatron** was loaded with the switch **-nasmupd (-nasu)**.

ASSEMBLY: REG. DISPLAY/SUPP. ALL

REG. DISPLAY
SUPP. ALL The default is REG.DISPLAY (regular display), unless **Cimatron** was loaded with the flag **-nasmdsp (-nasd)**. In this case all the assembly components are suppressed. When working with a saved session, it is recommended to use the SUPP. ALL (suppress all) switch.

Creating a New Assembly

Type the name of the new assembly file and press <CR>.

UNITS OF MEASUREMENT	MM	CM	M	INCH	FEET

After entering the appropriate units, the following prompt appears:

SELECT ENVIRONMENT	PART	ASSEMBLY	MOLD

Select ASSEMBLY.

Note: • When an assembly file is open, no more part files may be open.

Assembly Interactions

This section illustrates three common interactions in the ASSEMBLE module.

<PICK> from active

The following interaction is valid whenever picking features, planes axes or points from the Assembly components:

SELECT	REGULAR PICK
	PICK FROM ACTIVE

REGULAR PICK <PICK> the desired feature, plane, axis or point from the display using the mouse.

PICK FROM ACTIVE Upon selecting this option, you are prompted to <PICK> an active part, and then confirm the selection.

The next feature can only be picked from this active part. After selecting, the option automatically reverts to REGULAR PICK.

<PICK> a single assembly component

The following interaction is valid when picking an assembly component:

PICK COMPONENT

Whenever component selection is required, the user can pick a component from the screen or press <SUBMENU> to pick components from a list. Up to 30 components can appear simultaneously on screen. Pressing <SUBMENU> again will scroll forwards in the components list. Suppressed components will be marked with the letter "S".

PICK COMPONENT

<—	**ENTER NAME =**	- M	SMALL_TOP\		S0(0) U1(100)
- 1	COVER	2	BELT1	2	SCREW2
2s	FASTNER1	s 2	BELT2	s 1	BOLT1
	FASTNER2	s 1	BELT3	1	DRL32

■ ENTER NAME Pick a part by typing in its name.

Picking the arrow in the upper left corner will switch the direction of flipping through the list of parts.

To select the chosen component, <PICK> and press <EXIT>.

Even when the list appears, it is still possible to choose from the model.

To flip the scrolling direction, select —>.

<PICK> assembly components

The following interaction is valid when picking assembly components

Whenever component selection is required, the user can pick a component from the screen or from a list. Up to 30 components can appear simultaneously on screen.

PICK ENTITIES & EXIT

<— —>	**ENTER NAME =**	+	SMALL_TOP		
	COVER		BELT1		SCREW2
	FASTNER1	s	BELT2	s	BOLT1
	FASTNER2	s	BELT3		DRL32

■ ENTER NAME Pick a part by typing in its name.

Notes:

- Suppressed components are marked with the letter "S".
- <PICK> the arrows (<—>) in the upper left corner to scroll forward/backward in the components list.
- To select the chosen component, <PICK> and press <EXIT>.

  Even when the list appears, you can still choose a component directly by picking from the model..
- <PICK> the +/- modal to expand/collapse the component list.
- The letter M represents the Main Assembly and the letter A represents the Active Assembly.

- The component level in the assembly is displayed to the left a component name (e.g., +2 for the component FASTNER1 in the table above).
- In the upper right modal, the number of components that have been picked from the list is displayed, in the following format:
  Suppressed - S: n(m)
  Unsuppressed: U: x(y)
  where n and x are the number of already suppressed/unsuppressed components respectively and x and y is the current number of suppressed/unsuppressed components.
- Press <SUBMENU> to select multiple components. See PICK Multiple Entities in Chapter 3, Common Operations in the Fundamentals & General Functions Manual.

Main Options:

The assembly functions are contained in one overlay.

EDIT
ASSEMBLE
UTILITY
SUPPRESS
MODE
SUB_ASSM
PART

The EDIT menu

PARAMETERS Change the value of the part's parameters and update the parts.

RELATION Define relationships between parameters of features in different parts.

RE-ASSEMBLE Edit the **alignment** of a component to the assembly.

CONSTRAINTS Edit the **alignment parameters** (offset and theta) of the components in the assembly. This modal is similar to the RE-ASSEMBLE modal above except for some changes made for the user's convenience.

DISCARD Discard components of an assembly.

The ASSEMBLE menu

BY NAME	Pick component by entering it's name.
BY PICK	Pick the component by indicating it on screen.
REPEAT	Quickly assemble a component, several times on the same plane.

The UTILITY menu

INTERFERENCE	Check interference between parts in the assembly.
EXPLODE	View the default explosion. This option allows editing.
COLORS	Change the color of an assembly component.
RENAME	Change the name/path of an assembly and/or its components.
SAVE LIST	Gives information on files that will be saved in the next saving operation.
BLANK	Blank and unblank assembly components/datum.
DETACH	Change an instance to a free part.
TREE	Create an assembly tree structure file.

The SUPPRESS menu

Suppress and unsuppress assembly components.

The MODE menu

Switches the ASSEMBLY environment between automatic and manual ASSEMBLY updating. Manual updating assists editing parts in large assemblies.

The SUB_ASSM menu

Use the subassembly environment application to make modifications within the sub-assembly.

The PART menu

OPEN	Use the part environment application to modify within the assembly.
NEW	Access the part environment to create a new part.

ASSEMBLE

Build an assembly.

When creating a new assembly, the first component is the base component, and its UCS origin coincides with the Assembly origin. Each additional component is aligned to the existing components in the assembly.

The picking options are:

BY NAME
BY PICK
REPEAT

Select the component to be assembled:

BY NAME Fetch a component from a file.

BY PICK Pick a component from the assembly. An additional component (instance) will be fetched.

REPEAT Assemble an existing component (instance) several times on the same plane.

If BY NAME or BY PICK is chosen:

IND. LOCATION POINT Set a temporary location for the component.

SELECT				
	ALIGN PLANE	ALIGN AXIS		ALIGN UCS
SET COLOR				

Select the alignment method.

ALIGN BY PLANE / AXIS (by constraints)

SELECT				
NEXT —->	ALIGN PLANE	ALIGN AXIS	ALIGN POINTS	FIX
<— PREV	FLIP		OFFSET=0	THETA=0
SET COLOR	REF OFFSET = 0	SHIFT	ROTATE	ACTIVE PICK

Align a component relative to other components in the assembly by constraining features in the component relative to features in the assembly.

During alignment, constraints are applied to the 6 degrees of freedom of the component until it is fully constrained.

If required, use the SHIFT and ROTATE modals to move the component to a more convenient location with respect to the assembly. The current constraints are not affected. The component location is affected by all previous constraints.

Select an available constraint option: PLANE, AXIS or POINT.

<PICK> a reference entity on the component, then <PICK> the corresponding reference entity on the assembly. A sub-menu is available when picking.

Note:

- If you have problems to pick the right feature use the ACTIVE PICK modal. Select an active component, and now only picking from the active part will be available.

Set relevant parameters: OFFSET, ANGLE. The offset can be defined directly or can be calculated by reference.

The REF OFFSET modal allows you to define an offset between components as a result of an offset between two other components.

How To:

To define the REF OFFSET:

1. Enter the required REF. OFFSET in the modal.
2. <PICK> a plane in the assembly part. Press <SUBMENU> to pick an axis or a point. The picked feature/datum must be parallel to the constrained feature.
3. Similarly, <PICK> the second feature from the entire assembly. An arrow appears.
4. <PICK> the offset side. The offset is calculated and displayed.

For the current constraint, entities may be repicked by pressing the buttons ALIGN AXIS, ALIGN POINT and ALIGN UCS. Each button opens three options:

Re-pick Align. Re-pick the reference entity in the component.

Re-pick Alignment. Re-pick the reference entity in the assembly.

Re-pick Both Re-pick both reference entities.

Press NEXT to apply this constraint and advance to the next constraint.

As each constraint is applied, the degrees of freedom are decreased, and only the applicable constraint options will be shown on the menu bar.

If the last degree of freedom is rotation, the FIX option may be used. This option is intended for axi-symmetric components.

To go back one step in the constraint sequence, press <—-PREV.

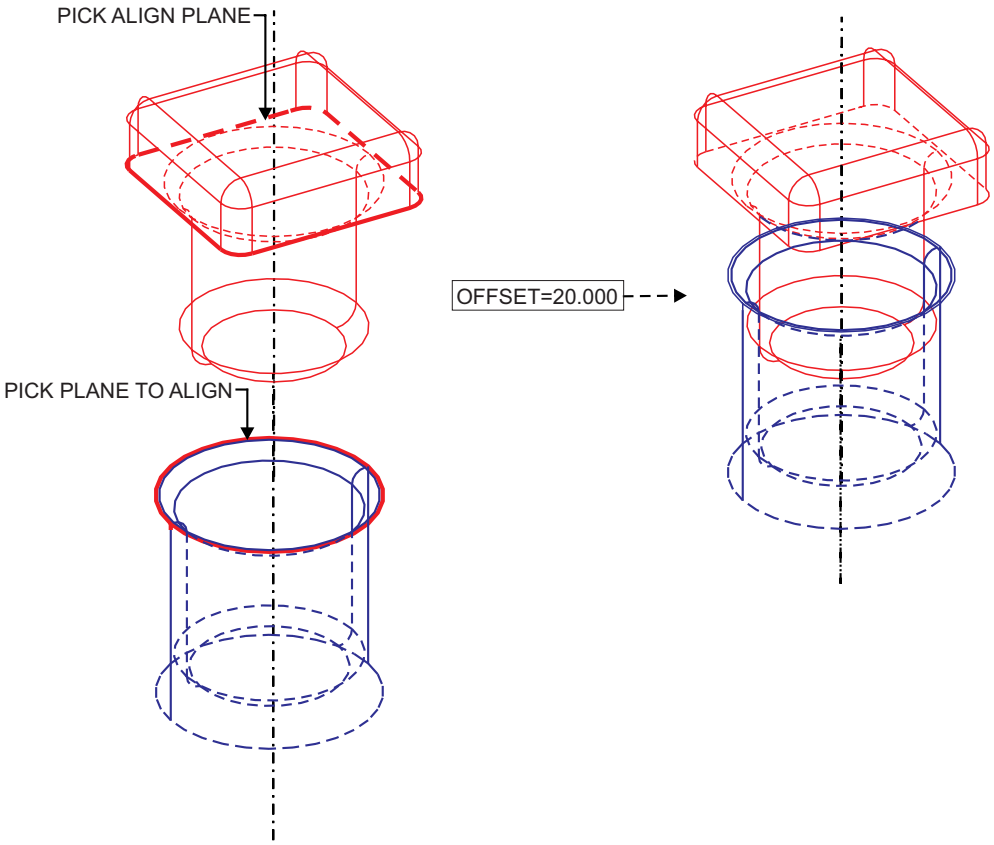

Figure 4-1: ASSEMBLE >> ALIGN BY PLANE

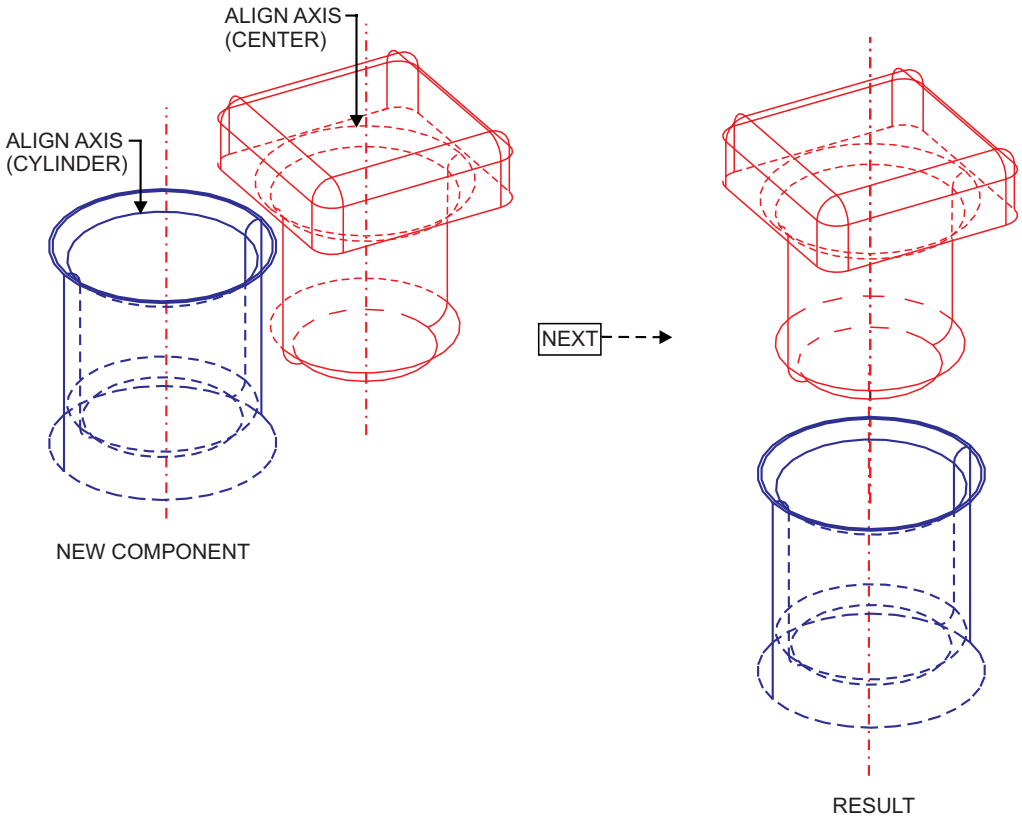

Figure 4-2: ALIGN BY AXIS

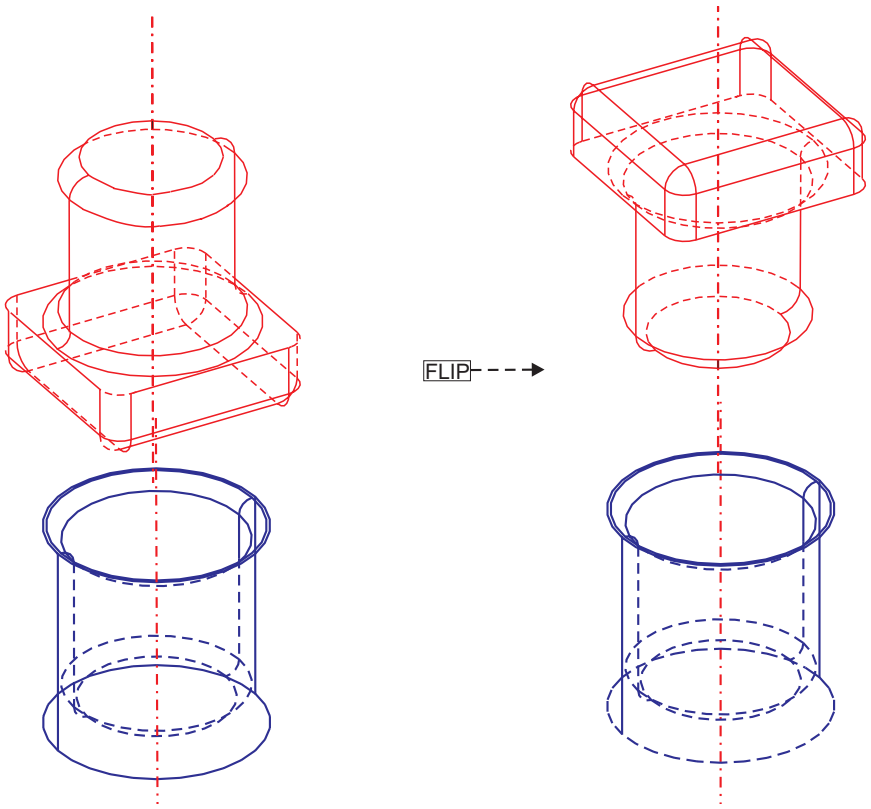

Figure 4-3: ALIGN BY AXIS and FLIP

ALIGN BY UCS

SELECT				
				ALIGN UCS
<— PREV	OFFSET - X = 0	OFFSET - Y = 0	OFFSET - Z = 0	
SET COLOR	THETA - X =0	THETA - Y =0	THETA - Z =0	ACTIVE PICK

Define coordinate systems for the component and assembly, and align both as needed.

Press the ALIGN UCS modal.

Define the component UCS by picking an origin point, and points in the positive X and Y directions.

Specify offsets and rotation angles (OFFSET & THETA) between the two coordinate systems.

Note:

- The offset is defined with respect to the assembly UCS, while the rotation angle is defined with respect to the component UCS.

Press <CR> TO CONFIRM to attach the component to the assembly and return to the ASSEMBLE menu.

When finishing the alignment process the modal table appears as follows. When reassembling the component, the modal table will begin from this state.

<CR> TO CONFIRM				
BY UCS				
<—-PREV				
SET COLOR	EXPLOSION			

- PREV — Returns to the previous modal table.
- EXPLOSION — See UTILITY >> EXPLODE, page 4-29.

If REPEAT is chosen:

PICK COMPONENT

Only components that were assembled with the following set of constraints can be manipulated with the REPEAT function:

1. PLANE AXIS FIX.
2. AXIS PLANE FIX.
3. PLANE POINT FIX.

Select a component that fulfills the above criteria.

After picking a component, the following prompt will appear:

IND. ALIGN POINT	<color of component>	SHOW ALIGNMENT	ACTIVE PICK

Pick an axis or a point, depending on the way the component was assembled. A new instance of the component will be immediately placed in the new location. The new instance of the component will be constrained in the same fashion as the original component (including offsets) EXCEPT for the new axis or point that has to be set.

- SHOW ALIGNMENT — Display the component constraints.
- ACTIVE PICK — Allow picking from the graphic display.

Note:

- There is no linkage in the constraints between the original and new instances created with this function.
- All the instances that were assembled using this option can be reassembled and discarded. ❑

EDIT

Edit the assembly by modifying feature parameters, setting relations between parts, reassembling, and discarding components.

Main Options:

PARAMETERS
RELATION
RE-ASSEMBLE
CONSTRAINTS
DISCARD

PARAMETERS Modify parameters of one feature.

RELATION Set relationships between parameters.

RE-ASSEMBLE Re-assemble a selected component from the beginning.

CONSTRAINTS Edit the alignment parameters (offset and theta) of the components in the assembly.

DISCARD Discard components from the assembly.

EDIT >> PARAMETERS

Modify parameters of a feature in the first level of the active assembly.

PICK FEATURE	REGULAR PICK
	PICK FROM ACTIVE

PICK FEATURE Select the feature to be modified.

- REGULAR PICK — See “Assembly Interactions”, page 4-7.
- PICK FROM ACTIVE — See “Assembly Interactions”, page 4-7.

IND. DIM POS / EXIT	VALUE=10.000

IND. DIM. POS Enter the updated value, and indicate the new position for the dimension.

Notes:

- Dimensions dependent on a parameter in another feature in the assembly will appear in light blue and cannot be edited.

EDIT >> RELATION

Define relationships between a parameter of a part in the first level of the active assembly, and parameters of other parts in the assembly. The result of this will be parametric dependency of this part on the related parts.

If a relational dimension exists in the assembly, the following prompt will appear:

SHOW ALL RELATIONS	YES	NO

YES All the relevant features are lit. Relations that are lit are only for the currently active assembly.

NO The features remain dimmed.

PICK FEATURES & EXIT	REGULAR PICK
	PICK FROM ACTIVE

- REGULAR PICK — See “Assembly Interactions”, page 4-7.
- PICK FROM ACTIVE — See “Assembly Interactions”, page 4-7.

PICK FEATURES & EXIT Select the features which will be related. The parameters will appear using the names assigned by the system, for example 1:R3.
The first number, followed by a colon, is the part ID. This is followed by the dimension type: L - Linear, R - radial, D - diameter, A - angular. The 2nd number is the ID within the group of parameters of this type, in the part. In the above example, 1:R3 is the third radial dimension in part 1.

Select the parameter which will be related.

IND. DIM. POS.	NAME = 0:L1	O:L1 = _____	VALUE = 24.1

Only dimensions from non-standard parts and parts that are not attached to subassemblies can be picked.

Define a mathematical relation between the selected parameter and one or more parameter from another parts in the assembly. Use standard symbols (+, - *, /, **, cos, sin) and precedence of operations. Parentheses may also be used. The new value will appear next to the name.

Enter the updated expression and press <EXIT>.

Select another dimension to create a new relation, or press <EXIT> again.

To discard a relation, enter <space> as the value.

Hint:

- If the child in the relation is to be a Round or Chamfer feature, first use the EDIT >> PARAMETERS to change the dimensions in the ONE option. After this step, you can assign a relation to the feature.

EDIT >> RE-ASSEMBLE

Re-assemble a selected component from the beginning.

This function enables the user to change the parameters (OFFSET & THETA), the constraint option (Plane, Axis, Point, Fix) or the alignment method.

PICK COMPONENT Select the component. The base component cannot be selected.

The ASSEMBLE menu will appear.

<CR> TO CONFIRM				
BY CONSTRAINT				
<—-PREV				
SET COLOR	EXPLOSION			

The component is presented in the final stage of the constraint sequence. Use the <—-PREV modal to step back through the constraint sequence.

Changing the value of the constraint parameters (OFFSET & THETA) at one stage in the sequence will not affect the constraints that follow.

If you change a constraint option (Plane, Axis, Point, Fix) or change which feature is aligned, the constraints that come after are automatically removed.

Note:

- If you have problems picking the right feature, use the ACTIVE PICK modal. Select an active component. Now only picking from the active part will be possible.

To change the alignment method, use the modal <—-PREV to go back to the beginning. Now choose to the appropriate method: ALIGN PLANE, ALIGN AXIS, ALIGN UCS. (See “ASSEMBLE”, page 4-11.)

If desired, define the explosion parameters.

Set the color. For a sub-assembly, if you do not define a color, the sub-assembly is set to its component’s color.

When the required settings have been made, press <CR> TO CONFIRM.

Note:

- Pressing <REJECT> at any step will return the component to its original placement.

EDIT >> CONSTRAINTS

Edit the alignment parameters (offset and theta) of the components in the assembly.

The CONSTRAINTS function interface is identical to the interface of the RE-ASSEMBLE function above, except for the following two changes:

- Only the offset and theta parameters can be changed.
- The influence of changes in these parameters are shown immediately after entering the new values. This ability enables the user to verify the output of the changes made without exiting the function.

EDIT >> DISCARD

Discard some or all of the components of an assembly. A component that is discarded becomes unattached after saving the main assembly.

PICK COMPONENTS	DISCARD	0 (100)

■ 0 (100) The first number is the number of the picked component. The second number is the total number of components.

Select the component(s) to be discarded from the assembly. To discard all the components in an assembly, <PICK> the main assembly. Press <SUBMENU> to display the Multi-Selection submenu and obtain more picking abilities. Components can also be picked or unpicked from the display.

Press YES to confirm.

Note:

- If you discard all subassembly components, the subassembly itself will be discarded.
- If you discard the first component in a subassembly, then the second component becomes the first component, and becomes automatically placed.

After discarding, save the Main Assembly to update the PFM with the changes.

Discarding a Component with Dependencies

If the discarded component has no more instances in the assembly, it is detached from the assembly. When detaching a component with parametric dependencies, these relations are cleared.

When detaching a component with geometric dependencies, the dependent features will be rerouted. These features are no longer fully dimensioned. The user can add the missing dimensions with the function EDIT >> SKETCH in the part environment. ❑

MODE

The MODE function controls and switches between the regular assembly environment (automatic updating of the assembly after every editing operation) and the manual update environment.

The MODE function toggles the assembly automatic update flag. The default state of the assembly auto.update flag is ON. This means that when assembly components change, the assembly automatically recalculates and updates the various relations between the assembly components. When the flag is OFF, this recalculation of the main assembly is postponed.

The advantages of working when the auto.update flag is OFF are:

- Quick loading of large and complicated assemblies.
- Several designers can work on the same assembly, concentrating on different parts.
- Faster work on part editing in the assembly. All part editing tools are available, including interference-check and creating views and sections for checking purposes. The Views and sections created will not be saved unless the auto.update flag is switched ON.

Note:

- When the auto.update flag is off, assembly editing is not possible, and the main assembly cannot be saved. In order to save, the auto update flag must be switched ON.

When the AUTO_UPDATE switch is ON, upon pressing UPDATE the following prompt appears:

AUTO UPDATE OFF?	YES	NO

YES The AUTO_UPDATE is switched OFF, and the assembly menu is changed so that the unavailable options are not displayed.

NO No change.

When the AUTO_UPDATE switch is OFF and the assembly is not updated, upon pressing UPDATE the following prompt appears:

UPDATE ASSEMBLY?	YES	NO

YES The assembly is updated.

NO No change.

When the auto.update switch is OFF and the assembly is updated, upon pressing UPDATE the following prompt appears:

AUTO UPDATE ON?	YES	NO

YES The AUTO_UPDATE is switched ON, and the assembly menu is changed so that the available options become displayed again.

Note:

- While Manual mode is ON, In the main assembly, the note “NO.UPD” will be displayed. ❑

PART

Enter the SOLID Modeling Module to edit a single part within the assembly environment.

Main Options:

OPEN
NEW

PART >> OPEN

Open an assembly part for editing.

PICK PART <PICK> the component to open. See "**Assembly Interactions**", page 4-7.

SELECT	ASSEMBLY MODE	FREE PART	Part name

Modify a part. The Part Environment menus will be displayed. To return to the assembly environment, select EXIT from the menu.

Save to confirm your work or EXIT without saving to reject.

Notes:

- It is possible to suppress parts in the assembly using the function UTILITY >> SUPPRESS PARTS.
- To save the changes, save the part, then save the main assembly.

PART >> NEW

Access the part environment to create a new part.

ENTER FILE NAME Enter the file name for the new part

SELECT	ASSEMBLY MODE	FREE PART

Select the appropriate mode.

NEW >> ASSEMBLY MODE

How To:

1. Select an active component. Now only the active part will be available for picking.
2. Edit the parameters 'OFFSET" & "THETA" if necessary, and SET COLOR.
3. Press CR TO CONFIRM when finished. The Part Environment will be opened
4. Create the part. By definition, the part will be aligned BY REFERENCE.
5. To return to the Assembly application, choose EXIT in the menu. >EXIT with a save to view your work in the assembly or EXIT without saving to reject your work in the part.

Interaction:

<CR> TO CONFIRM				
	REFERENCE UCS	REFERENCE AXIS	PLANE / FACE	
			OFFSET - Z=0	
SET COLOR				ACTIVE PICK

Note: • If you have problems picking the right feature use the ACTIVE PICK modal.

NEW >> FREE PART

Create an independent part in the Part Environment.

To return to the Assembly application, press EXIT in the menu. You will be prompted to assemble the new part. If you answer yes, the ASSEMBLE menu will be opened.

Assemble the part. If you did not assemble the part, it will be saved automatically.

Note: • To save the new part, save it as a part, and save the main assembly.

Enter Assembly Mode to use the geometry from the Assembly. In the sketch, you can pick the geometry. If the geometry is shown in cyan, this means it is connected, i.e., any change in a parent dimension will affect its children. If it is shown in red, it is not connected.

To use a main assembly geometry, enter a part directly from the main assembly.

PART: Modal Parameter Definitions

- ASSEMBLY MODE — Modify a part. The rest of the assembly will be displayed and reference geometry may be taken from it, to generate geometric dependencies. The Part Environment menus will appear.
- FREE PART — Modify a part independently of the rest of the assembly. The Part Environment will open.
- OFFSET Z=0 — Define an offset to the plane/face described above.
- PLANE/FACE — Pick a plane/face on which the sketcher of the first feature for the new part will be placed.
- REFERENCE AXIS — Define the location of the part by indicating Reference Y-Axis and X-direction, which define the new part UCS.

  In this mode no parameters (OFFSET, THETA) are available.
- REFERENCE UCS — Define the location of the part by indicating three points on the assembly which define the new part UCS. A sub-menu is available for picking points. ❑

SUB_ASSM

Use the subassembly environment application to make modifications within the assembly.

How To:

1. Pick a subassembly to activate within the assembly environment, then <EXIT>.
2. Pick the subassembly from the display, or press the right mouse button to display a list of subassemblies and select from the list.
3. The subassembly becomes the active assembly.
4. To return to the main assembly, pick the MAIN option from the overlay.

Interaction:

PICK SUB_ASSEMBLY Pick a subassembly to activate within the assembly environment, then <EXIT>.

Note:

- To save changes made in the subassembly, save the Main assembly.

❑

SUPPRESS

Suppress or unsuppress the display of components from the assembly.

Note:

- In all the list interactions, the first pick will select a part. A second pick will unselect it.
- To confirm the selection, press <EXIT>. Additional parts can be selected. To exit the function, press <REJECT>.

The first step is to define the scope of the action.

MAIN
ACTIVE

MAIN Action will apply to the entire subassembly except for the active component.

ACTIVE Action will apply only to the active component.

Note:

- In the main assembly, it is automatically active.

PICK ENTITIES & EXIT					
<—	— >	ENTER =	—	<partname>\	S : 0(3) U: 1(1)
	<component-name>		S	<component-name>	

■ <— —> Scroll through the component list.

For details of all the modals displayed in this table see <PICK> assembly components on page 4-8.

<PICK> the main assembly name to obtain the following options:

UNSUPPRESS ALL
SUPPRESS ALL
TOGGLE

Components can be selected using standard select procedures (see **Assembly Interactions**, page 4-7).

Notes:

- If you pick both a subassembly and a part, the function will act only upon the subassembly
- If the part is in Assembly mode, use the option UTILITY >> SUPPRESS PART.
- Use the SINGLE/BOX/POLYGON selection options to <PICK> several components. ❑

UTILITY

Perform general operations on assemblies and components.

Main options:

INTERFERENCE
EXPLODE
COLORS
RENAME
SAVE LIST
BLANK
DETACH
TREE

INTERFERENCE Check interference between parts in the assembly.

EXPLODE Create an exploded assembly.

COLORS Change/move the color of an assembled component.

RENAME Rename an assembly and/or its components.

SAVE LIST Show the list of components that will be saved with the assembly.

BLANK Blank and unblank assembly datums.

DETACH Change an instance to a free part.

TREE Create an assembly tree file.

UTILITY >> INTERFERENCE

Check interference between parts in the assembly.

Main options:

ALL TO ALL
ONE TO ONE
ONE TO ALL
CLEAN

ALL TO ALL Interference between all parts in the assembly.

ONE TO ONE Interference between two parts. Each part must be selected.

ONE TO ALL Interference between the selected part and the rest of the assembly.

CLEAN Clean display of interference areas.

If the option requires, select the parts whose interference will be checked.

Lines of interference will be displayed in red.

To erase the display of interference areas, press CLEAN.

UTILITY >> EXPLODE

The Assembly environment enables the user to create exploded pictures with the UTILITY - EXPLODE function. After an exploded picture is edited, the picture can also be shaded and exported as a GIF file. Exploded 2D views can be created from these pictures. See the DRAWINGS function.

After creating an exploded picture and editing it, the model must be RESET before continuing to work on the model.

<CR> TO CONTINUE	GLOBAL OFFSET = 10.000

Define the initial offset.

SELECT	GLOBAL OFFSET=10	LOCAL CHANGES	RESET

Note: Exiting the function without resetting is possible when creating a GIF file (See the PLOT function in the **Fundamentals and General Functions** manual).

EXPLODE >> LOCAL CHANGES

Controlling Part Explosion

There are three ways to control the exploded picture of a part that is directly connected to an assembly (that does not belong to a subassembly). When choosing such a part, the following menu will appear:

<CR> TO CONFIRM			
BY DEFAULT	BY PLANE	BY AXIS	EXPLODE FROM
FLIP	GLOBAL OFFSET* / LOCAL OFFSET	OFFSET = 10.00	
EXPLOSION			

How To:

1. The direction of explosion can be set with the following modals:

   FLIP, BY DEFAULT, BY PLANE, BY AXIS

2. The offset value can be set with the following modal:

   GLOBAL OFFSET/LOCAL OFFSET

3. Each part is exploded from a part that is defined as its “FATHER”. The father can be changed using the modal:

   EXPLODED FROM

Controlling Subassembly Explosion

There are five ways to control the exploded picture of a sub assembly. When choosing a subassembly, the following menu will appear:

<CR> TO CONFIRM

BY DEFAULT	BY PLANE	BY AXIS	EXPLODE FROM
FLIP	GLOBAL OFFSET* / LOCAL OFFSET	OFFSET = 10.00	EXPLODE BASE
EXPLOSION	GLOBAL INT. OFFSET / LOCAL INT. OFFSET	OFFSET = 10.00	

The first three ways to control the exploded picture of a sub-assembly are used to control the explosion characteristics of the entire subassembly as one unit:

How To:

1. The direction of explosion can be set with the following modals:

   FLIP, BY DEFAULT, BY PLANE, BY AXIS

2. The offset value can be set with the following modal:

   GLOBAL OFFSET/LOCAL OFFSET

3. Each part is exploded from a part that is defined as its “FATHER”. The father can be changed using the modal:

   EXPLODED FROM

The two additional ways to control the exploded picture of a sub-assembly are used to control the internal behavior of the explosion of the subassembly:

How To:

1. The default internal offset between the subassembly parts is the GLOBAL OFFSET parameter. The value can be changed by inserting a different value into the GLOBAL INT. OFFSET/ LOCAL INT. OFFSET modal. After typing in a value into this modal, the prompt will change to LOCAL INT. OFFSET, and the GLOBAL INT. OFFSET value will cease to govern the explosion parameters of this part.
2. **Each** subassembly contains a single part that is defined as the local base for the subassembly explosion. This part will be offset only by the general offset. The rest of the subassembly’s parts will be offset relative to the base part.

   Use the parameter EXPLODE BASE to change the BASE of the subassembly.

Notes:

- When setting the internal offset of a subassembly to a non zero value, make sure that the exploded picture of that subassembly in the subassembly file is correct.
- It is recommended to define the base part as the part that is connected to the main assembly.

UTILITY >> COLORS

PICK COMPONENT/EXIT Pick the component whose color is to be modified.

SELECT	SET COLOR / RESET COLOR	<Color list>

You may select any color from the color list.

A subassembly can receive the color of its components by choosing RESET COLOR, or can be made one color.

Note:

- You cannot change colors of an assembly component while the assembly is in only one color.
- All part objects will be the same color in the assembly.

UTILITY >> RENAME

The component name list is as follows:

PICK COMPONENT/EXIT				
——->	ENTER=_____	<ASSEMBLY>	COMPONENT 1	COMPONENT 2
COMPONENT 3	COMPONENT 4	COMPONENT 5	COMPONENT 6	COMPONENT n

Pick a component or the main assembly from the screen, or select from the component name list. After picking a component the following prompt will appear:

ENTER NEW NAME

OLD NAME = <current_path_and_file_name>

After entering the new component name, the list changes as follows:

ENTER NEW NAME

NEW NAME = <new_path_and_file_name>
OLD NAME = <current_path_and_file_name>

At this stage the new name can be altered. Pressing <ENTER> replaces the old name with the new name. <REJECT> cancels the renaming. Pick another component to rename or <EXIT>.

The symbol “N” by the name indicates that name has been changed.

After making the desired changes, the prompt changes to:

CONFIRM RENAME	YES	NO

YES The renaming takes place. The assembly will automatically be saved.

NO The renaming is canceled.

Notes:

- The assembly must be saved before entering RENAME.
- When renaming standard components, the original files are not deleted.
- The function does not work with the search path rule.

UTILITY >> SAVE LIST

Show list of file names of assembly components that will be saved at the next assembly <SAVE> operation.

LIST FILE NAMES	YES	NO	# COMPONENTS TO SAVE	# WITH PROBLEMS

UTILITY >> BLANK

Blank and unblank assembly datums.

BLANK IN COMPONENT
BLANK ALL
SHOW AS IN PART

BLANK IN COMPONENT Blank certain data within a part.

PICK PART Pick the part whose data you want to blank/unblank.

The system recognizes all categories of data associated with the component. You can blank/unblank these by category or by picking on the display.

BLANK ALL Blank all datums from the assembly.

SHOW AS IN PART Show unblanked datums in all parts. Blank datums in parts are not displayed.

UTILITY >> DETACH

Change an instance to a free part.

The interaction is the same as for UTILITY >> RENAME, page 4-32. A new pfm file is created for the part, and all external relations will be discarded.

UTILITY >> TREE

Create an assembly tree file.

Each level in the tree is displayed one tab to the right of the previous level.

UTILITY: Modal Parameter Definitions

- # COMPONENTS TO SAVE — Number of assembly components that will be saved.
- # WITH PROBLEMS — Number of such components that there problems to save, i.e. no write permission.
- BY AXIS — Change the direction of the explosion to a chosen axis
- BY DEFAULT — Change the explosion direction to the default.
- BY PLANE — Change the direction of the explosion to be normal to a chosen plane.
- EXPLODE BASE — Changes the BASE of the subassembly.
- EXPLODED FROM — Changes the “FATHER” of the part/subassembly.

- FLIP — Changes the direction of the explosion by 180 degrees.
- GLOBAL INT. OFFSET — The default internal offset between the subassembly parts is the GLOBAL OFFSET parameter. The value can be changed by inserting a different value into the GLOBAL INT. OFFSET / LOCAL INT. OFFSET modal. After typing in a value into the GLOBAL INT. OFFSET parameter, the prompt will change to LOCAL INT. OFFSET, and the GLOBAL INT. OFFSET value will cease to govern the explosion parameters of this part.
- GLOBAL OFFSET — Set the offset between all the parts in the exploded picture. Specific offsets can be defined using LOCAL CHANGES.
- GLOBAL OFFSET* — As long as the offset value was not changed, the offset will be controlled by the GLOBAL OFFSET parameter. After typing in a value into this modal, the prompt will change to LOCAL OFFSET, and the GLOBAL OFFSET value will cease to govern the explosion parameters of this part.
- LOCAL CHANGES — When specific part or sub assembly locations that result from the default values do not satisfy your requirements, edit the exploded picture's parameters.
- LOCAL INT. OFFSET — The default internal offset between the subassembly parts is the GLOBAL OFFSET parameter. The value can be changed by inserting a different value into the GLOBAL INT. OFFSET / LOCAL INT. OFFSET modal. After typing in a value into the GLOBAL INT. OFFSET parameter, the prompt will change to LOCAL INT. OFFSET, and the GLOBAL INT. OFFSET value will cease to govern the explosion parameters of this part.
- LOCAL OFFSET — As long as the offset value was not changed, the offset will be controlled by the GLOBAL OFFSET parameter. After typing in a value into the GLOBAL OFFSET parameter, the prompt will change to LOCAL OFFSET, and the GLOBAL OFFSET value will cease to govern the explosion parameters of this part.
- RESET — Reset the offsets in the model.
- YES/NO — Display the file or reject.

  If YES is selected, the file will be saved with the assembly name and a ***.sav** extension. ❑

Chapter 5
General Functions in the ASSEMBLY Environment

Introduction

The general function menu, as shown below, can be used with Solid Modeling as with other **Cimatron** functions.

Overlay I
UCS
FILE
WINDOW
VERIFY
EXIT

Overlay II
DISPLAY
PLOT
ATRBUTES
SHADE
ANALYZE
REGISTER
EXIT

The interactions of the FILE >> COMP FILE, VERIFY, and ANALYZE functions are different from the SOLID Modeling Functions.

The FILE menu

SAVE PART Save the assembly with all changes made during the current session, and continue working on the same assembly.

SESSION Save/load work session of the assembly.

The VERIFY menu

COMPONENT	Verify the status of an assembly component.
GENERAL	Calculate data which is related to points, edges and faces (i.e. distances etc.).

The ANALYZE menu

ANALYZE	Verify the volume, area, center and center of gravity (C.G.) of all the parts in an assembly or all unsuppressed parts in an assembly.

ANALYZE

Verify the volume, area, center and center of gravity (C.G.) of all the parts in an assembly or all unsuppressed parts in an assembly.

<CR> TO CONTINUE	LOCAL CHANGES	ALL PARTS / ALL DISPLAYED PARTS	DISPLAY C.G. / DO NOT DISPLAY C.G.
	TOL = 0.1	DENSITY = 1.0	NUM. UNINIT = 17

Notes:

- The tolerance and density parameters will influence only the uninitialized parts (UNINIT).
- Once the data (volume, area, center and center of gravity) has been calculated for a part, it is saved for later use and the part is defined as initialized (INIT). A part for which this data has not been calculated is uninitialized (UNINIT).

PICK COMPONENT — See "Assembly Interactions", page 4-7. After picking a part, the ANALYZE menu from the part environment will appear. The menu will enable you to recalculate the mechanical parameters of the part with a new tolerance or density.

<CR> TO CONTINUE — Execute the calculations and display the results.

ANALYZE: Modal Parameter Definitions

- ALL DISPLAYED PARTS — Select only the unsuppressed parts to be entered into the calculation.
- ALL PARTS / — Select the entire assembly to be entered into the calculation.
- DISPLAY C.G. — Display the center of gravity indicator. The center of gravity indicator can be used for verification of distances in the assembly.
- DO NOT DISPLAY C.G. — Hide the center of gravity indicator. The center of gravity indicator can be used for verification of distances in the assembly.
- LOCAL CHANGES — Define or modify the density and tolerance of a part in an assembly. The following prompt appears:

PICK COMPONENT	NEXT UNINIT.

- NEXT UNINIT. — Go to the next uninitialized part. If you pick it, the ANALYZE menu from the part environment will appear. The menu will enable you to calculate the mechanical parameters of the part.
- NUM. UNINIT = 17. — Number of uninitialized parts in the assembly. ❑

FILE

Save the displayed assembly with all changes made during the current session, and continue working on the same assembly.

SELECT	SAVE PART
	COMP FILE
	PARAMETERS
	TYPE
	SETUP
	SESSION

SAVE PART Save the assembly file with all changes made in the current session.

COMP FILE Create a file which shows the composition of the current part file or ASSEMBLY file

PARAMETERS Recreate the xx**param** file based on the settings for the current part.

TYPE Save file as a **STANDARD** file.

SETUP Change the **Cimatron** setup parameters.

SESSION Save or restore an assembly session.

For full descriptions of all the options in a regular PART environment, refer to the **Fundamentals and General Functions Manual**.

FILE >> SAVE PART

Save an assembly file.

SAVE-CONFIRM? Y / N Save the assembly file.

Notes:

- During the assembly work, if components were updated they were saved in temporary assembly file versions. These temporary files are saved by the same operation.
- The system saves all files in the save list. If one file from the list cannot be saved, none of the files will be saved.

FILE >> COMP FILE

Create a composite file which shows the composition of the current assembly file.

<CR> TO CONTINUE	DISP ON
	DISP OFF

<CR> TO CONTINUE Press the <RETURN> key to continue.

■ DISP ON / OFF Activates the display to be on or off.

A composite file is created of the name of the assembly with the extension **.cmp**. The following table(s) is / are displayed:

composition of assembly d:\sp\compl_as

Name	Type	Obj - ID	Count	Dep.*
d:\sp\parts\separ_sb	Sub-Assembly	—	1	—
d:\sp\parts\plita_as	Sub-Assembly	—	1	—
d:\sp\gidro\gpriv_as	Sub-Assembly	—	2	—

* P:parametric dependency G:geometric dependency

number of components: 234
number of parts : 179

composition of sub-assembly d:\sp\parts\separ_sb

Name	Type	Obj - ID	Count
d:\sp\parts\osnvn_sb	Sub-Assembly	—	1
d:\sp\parts\redkt_sb	Sub-Assembly	—	1
d:\sp\parts\motor_sb	Sub-Assembly	—	1
d:\sp\parts\kozjh_sb	Sub-Assembly	—	1
d:\sp\parts\rotor__p	Part	5	1

number of components: 104
number of parts : 79

composition of sub-assembly d:\sp\parts\plita_as

Name	Type	Obj - ID	Count
d:\sp\parts\plita__p	Part	39	1
d:\sp\parts\balka__p	Part	40	1
d:\sp\parts\opora__p	Part	72	2
d:\sp\gidro\sh20&b15	Std. Sub-Assembly	—	8

number of components: 28
number of parts : 20

composition of sub-assembly d:\sp\gidro\gpriv_as

Name	Type	Obj - ID	Count
d:\sp\gidro\korpus_w	Part	44	1
d:\sp\gidro\porsh__w	Part	45	1
d:\sp\gidro\kryshk_w	Part	46	1
d:\sp\gidro\vilka__w	Part	47	1
d:\sp\gidro\shkal_as	Sub-Assembly	—	1
d:\sp\gidro\strelk_w	Part	49	1
d:\sp\gidro\gaika__w	Part	50	1
d:\sp\gidro\vint_w16	Part	51	1
d:\sp\gidro\ospod_as	Sub-Assembly	—	2
d:\sp\gidro\shaiba_w	Part	53	1
d:\sp\gidro\vint_w17	Part	54	4
d:\sp\gidro\sh20&b15	Std. Sub-Assembly	—	4

number of components: 49
number of parts : 40

The first table is for the main assembly.
The remaining tables pertain to the sub-assemblies that make up the main assembly.
The following definitions apply to the tables:

Name Full path name of the component.

Type The type of the component. There are four categories for Type:

1. Part
2. Sub-Assembly
3. Std. Part - a standard part.
4. Std. Sub-Assembly - a standard sub-assembly.

Obj-ID A unique identifier.

Count The number of times the component appears in the assembly.

Dep.* An indicator that the component is dependent parametrically (p) or geometrically (g) on other parts in the assembly.

Under each assembly, the number of components and parts in the assembly are displayed.

FILE >> PARAMETERS

See the FILE function on page 3-5 in Chapter 3.

FILE >> TYPE

A file type may be standard or non-standard (default). A standard file may appear in several assemblies. A non-standard file may be attached to an assembly, but only to one assembly.

Note: • A non-standard file attached to an assembly must be copied in order to use it in another assembly.

<CR> TO CONTINUE	NON-STANDARD	<attachment>
	STANDARD	

The <attachment> can be one of the following:

UNATTACHED
ATTACHED TO ASSEMBLY <assembly name>
ATTACHED, WITH DEPENDENCIES, TO ASSEMBLY <assembly name>
ATTACHED WITHOUT DEPENDENCY

In the case of parts that are UNATTACHED or ATTACHED WITHOUT DEPENDENCY, the option to switch to Standard is presented in the following manner:

MODIFY TO STANDARD ? YES / NO

Note: • A STANDARD part cannot be restored to a NON-STANDARD part.

FILE >> SETUP

See the FILE function on page 3-5.

FILE >> SESSION

Save or restore an assembly session. This option can only be used in the main assembly.

Note:

- A session file saves the following information:
  - The orientation of the display.
  - Details of suppressed/unsuppressed components.
  - Display mode (wireframe / HDL / Shade)
- There are two types of session files:
  - The automatic session file which is saved with the file and restored the next time the file is loaded.
  - There is also the option to save the session file to a new file and restore this session, similar to the concept of a “model view”. The sessions can then be switched without going through the steps involved by restoring a previously saved session.

SESSION	SAVE
	RESTORE

SAVE Save the current session: display orientation, display mode, and unsuppressed components.

RESTORE Open a previous session. This must be a session within the same assembly. Type in a session name or use the browse facilities.

❑

VERIFY

Verify the status of an assembly component or calculate data which is related to points, edges and faces (distances etc.).

GENERAL
COMPONENT

VERIFY >> COMPONENT

Verify the status of an assembly component.

PICK COMPONENT <Pick> the component to verify. See "**<PICK> Assembly Components**," page 4-8.

<name>	<type>	OBJ_ID=3	COUNT=1
SHOW PARAMETRIC DEPEND.	SHOW GEOMETRIC DEPEND.	SHOW PLACEMENT DEPEND.	
NO PARAMETRIC DEPEND.	NO GEOMETRIC DEPEND.	NO PLACEMENT DEPEND.	

If the selected component is within a sub-assembly, the following is displayed:

<name>	<type>	OBJ_ID = 3	COUNT = 1
IN:<name>	<type2>	SHOW PLACEMENT DEPEND.	
		NO PLACEMENT DEPEND.	

VERIFY >> GENERAL

Calculate data which is related to points, edges and faces (distances etc.)

See the **General Service Functions** in the **Fundamentals & General Functions Manual.**

VERIFY: Modal Parameter Definitions

- <name> — The file name of the selected component.

  When this modal is highlighted, the component is highlighted.

- <type> — The type of component selected. There are five types and they are as follows:

  PART
  SUB-ASSEMBLY
  STD. PART (standard part)
  STD. SUB-ASSEM. (standard sub-assembly)
  ACTIVE PART (the active part in ASSEMBLY mode.)

- <type2> — The type of sub-assembly where the selected component is located.

- COUNT=1 — The number of times the component appears in the ASSEMBLY.

- IN:<name> — The name of the sub-assembly where the selected component is located. When this modal is highlighted, the entire sub-assembly is highlighted.

- NO GEOMETRIC DEPEND. — The selected component is not geometrically dependent upon other ASSEMBLY parts.

  This modal does not appear for sub-assemblies or for standard parts.

- NO PARAMETRIC DEPEND. — The selected component is not parametrically upon other ASSEMBLY parts.

  This modal does not appear for sub-assemblies or for standard parts.

- NO PLACEMENT DEPEND. — The selected component placement is not dependent upon other ASSEMBLY parts.

- OBJ_ID=3 — A unique file identifier.

- SHOW GEOMETRIC DEPEND. — Highlights the features upon which the selected component is geometrically dependent.

- SHOW PARAMETRIC DEPEND. — Highlights the features upon which the selected component is parametrically dependent.

- SHOW PLACEMENT DEPEND. — Highlights the features upon which the selected component's placement is dependent. ❑

Chapter 6
Mold Functions

Introduction

Mold design operations can be performed on Solid objects within the Part environment. To perform assembly operations within the Part environment, add the switch **-apenv** when invoking Cimatron and select the Part environment option when creating a new part file.

The Part environment produces a multi-object solid file **mold_<filename>.pfm**. The Mold workflow includes model importing and placing, different Mold operations, including MoldBase 3D creation and Mold Assembly creation if necessary. The result of this process is a chain of linked files that maintains up-down associativity between the design model, Mold model, Mold Assembly and Mold Assembly Components.

To define a part for mold operations, after selecting the units of measurement for the new part, select the Part environment option from the SELECT ENVIRONMENT menu.

SELECT ENVIRONMENT	PART	ASSEMBLY

The Mold Application Menu

The Mold Application Menu functions are contained in two overlays.

Overlay I
MOLDPREP
SEPARATE
MOLDBS 3D

Overlay II
EDIT
CREATE
DETAIL
SURFACE
MODIFY
DATUM
COPY
GROUP
TRANSL
UTILITY
EXTR2ASM

Overlay I consists of all Mold functions. Overlay II consists of Solid functions, whose explanations can be found in Chapter 2.

EDIT	Edit existing features and/or operations. Interactions for this function can be found on page 2-86.
MOLDPREP	Setup the mold environment for Mold design operations.
SEPARATE	Define parting lines, parting surfaces, and divide the Mold.
EXTR2ASM	Produce and manage an assembly in a multi-object file.
UTILITY	Set of different utilities for objects and datums.

EDIT

Change all parameters for solid/datum creation operations, including the workpiece/model parameters, shrinkage, placement, or parting surface.

Note:

- If you want to change the shrinkage or placement parameters after a parting line has been created, use the REPLAY option to move back to before the parting line creation, and change the parameter in INSERT mode. Then, recreate the parting line using the Restore —> button.

For details of this function and its options, refer to the Solid EDIT function on page 2-86.

❑

EXTR2ASM

Generate assembly file(s) from components of the multi-object file.

If an assembly file has ***not*** previously been defined for this mold, the initial interaction for defining the assembly file is identical to that of EXTR2ASM >> EXPORT.

Main Options:

EXPORT
RENAME
EXCLUDE

EXPORT Export objects from the given file into an assembly file and create a separate part file for each component.

RENAME Rename files for existing exported objects.

EXCLUDE Remove any unnecessary components of the assembly from the Part file in which this assembly was created.

EXTR2ASM >> EXPORT

Export objects from the Part file into an assembly file.

ENTER ASSEMBLY FILE Type a name for the assembly file and press <CR>.

PICK OBJECTS & EXIT Select the objects to be exported into the assembly component file and press <EXIT>.

ENTER FILE NAME Enter the name of the assembly component.

PICK OBJECT(S) & EXIT Continue picking objects and exporting into a component of the assembly. Press <EXIT> when finished.

SAVING PART FILE

EXECUTING The system generates the assembly file (if not previously defined) and the assembly components.

EXTR2ASM >> RENAME

Rename the assembly file and/or its components.

PICK OBJECT & EXIT		
<path>\<assembly-name>	<path>\<component-name>	<path>\<component-name>

To rename the assembly file:

PICK OBJECT & EXIT Select the assembly file name from the list (shown in green).

ENTER ASSEMBLY FILE Enter the new name for the assembly.

To rename a component:

PICK OBJECT & EXIT Select the component name from the list or select the component on the screen.

The selected component is highlighted in blue on the screen, and its name is presented in purple.

ENTER FILE NAME Type in the new name for the component and press <CR>.

EXTR2ASM >> EXCLUDE

Remove a component(s) from the assembly file.

PICK OBJECT & EXIT	
<path>\<component-name>	<path>\<component-name>

PICK OBJECT & EXIT Select the component name from the list or select the component on the screen.

The selected component is highlighted in blue on the screen, and its name is presented in purple.

Press <EXIT> to remove the component from the assembly file.

Note:

- Associativity exists between the assembly components, the original file and the model. If the original file is updated, or moved and the assembly file is subsequently loaded, an appropriate message is displayed.

❑

MOLDPREP

Setup the mold environment for Mold design operations.

Main Options:

MODEL
SHRINKAGE
WORKPIECE
PLACE

MODEL Import the model from an external pfm file and define the model orientation.

SHRINKAGE Calculate and rebuild the model accordingly to shrinkage compensation parameters.

WORKPIECE Define the workpiece with dimensions.

PLACE Place the model relative to the workpiece, the workpiece relative to the model, or both, relative to the datum plane.

MOLDPREP >> MODEL

Import the model from an external pfm file and define the model orientation.

How To:

1. Type or choose the pfm file that contains the model. Activate the submenu for the displayed list of files.

   There are two ways to define a slope; steps 2 and 3.

2. Use the slope of an existing edge or axis as a reference slope.

   a). Choose an axis or straight edge which will be used to define the slope. Use the AXIS/EDGE parameter.

   b). Reverse the indicated direction, if desired. Use the FLIP parameter.

3. Use the slope of a vector defined by two points: an origin and a point indicating the positive direction.

   a). Indicate an origin point. Use the ORIGIN parameter.

   b). Pick a point to indicate the positive direction of the vector. Use the + DIRECTION parameter.

   c). Apply the above slope constraints to generate a new spline. Use the APPLY parameter.

4. Define the reference base point of the model. This point must not belong to the model being imported.

5. Define the angle of rotation around the defined model direction in a counterclockwise direction.

Interaction:

ENTER FILE NAME Type or choose the pfm file that contains the model. Activate the submenu for the displayed list of files.

DEFINE & EXIT	DIRECTION	BASE POINT	ANGLE=0.000

DEFINE & EXIT The model is shown in dashed lines representing a preposition of the mold. Note that at this point, the model has not as yet been imported into the Mold file.

DEFINE DIRECTION	AXIS/EDGE	FLIP	ORIGIN	+ DIRECTION	APPLY

DEFINE DIRECTION There are two ways to define a slope (see How To, above).

IND. BASE POINT Pick the base point. Activate the submenu for picking choices.

When the definition is defined, press <EXIT>. The new part is created and the model is now displayed in solid lines.

Note:

- More than one model may be imported into the current mold.
- For best results, it is recommended to set the value of the UTILITY >> SET TOLERANCE option, OBJ. TOL modal to the value of the same variable in the Solid file.

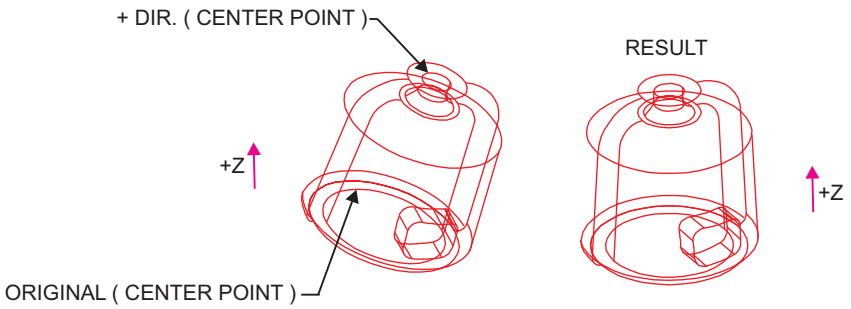

Figure 6-1: MOLDPREP >> MODEL

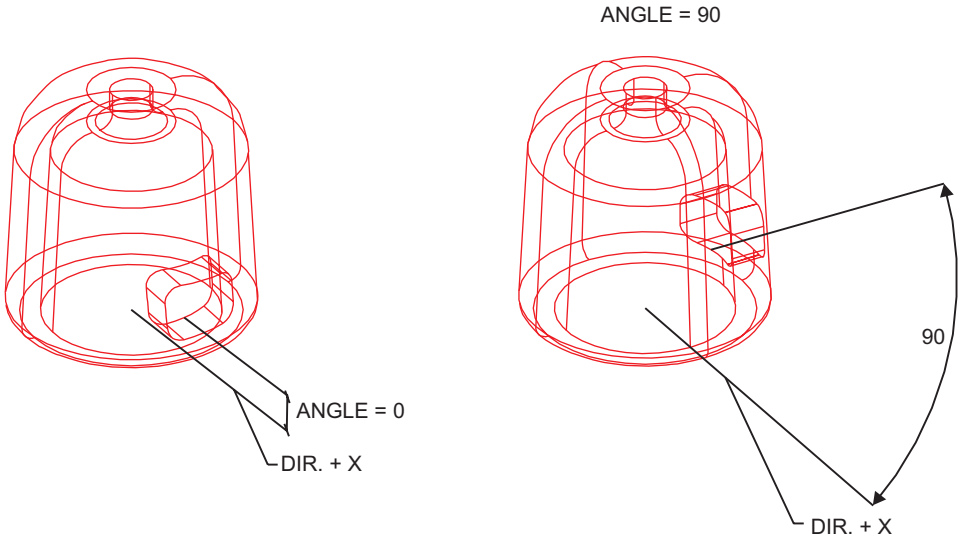

Figure 6-2: MOLDPREP >> MODEL, ANGLE Modal Parameter

MOLDPREP >> SHRINKAGE

Scale the model using uniform scaling.

IND. REF. POINT / EXIT Indicate a reference point from which all calculations will be performed.

PICK ENTITIES & EXIT	SCALE = 1.010

Pick the entities to be scaled.

Enter a scaling value.

Press <EXIT>.

MOLDPREP >> WORKPIECE

Define the workpiece.

RECTANGULAR
CYLINDRICAL
FROM FILE
BY PICK

WORKPIECE >> RECTANGULAR

Define the workpiece as a rectangular box around the model.

<CR> TO CONTINUE	LENGTH = 54.000	WIDTH = 54.000	HEIGHT = 41.000

The system uses a 1.2 factor presents default parameters that are a factor of 1.2 the maximum length of each edge of the bounding rectangle of the model, but these parameters can be changed as required.

Press <CR> to accept the values and create the workpiece.

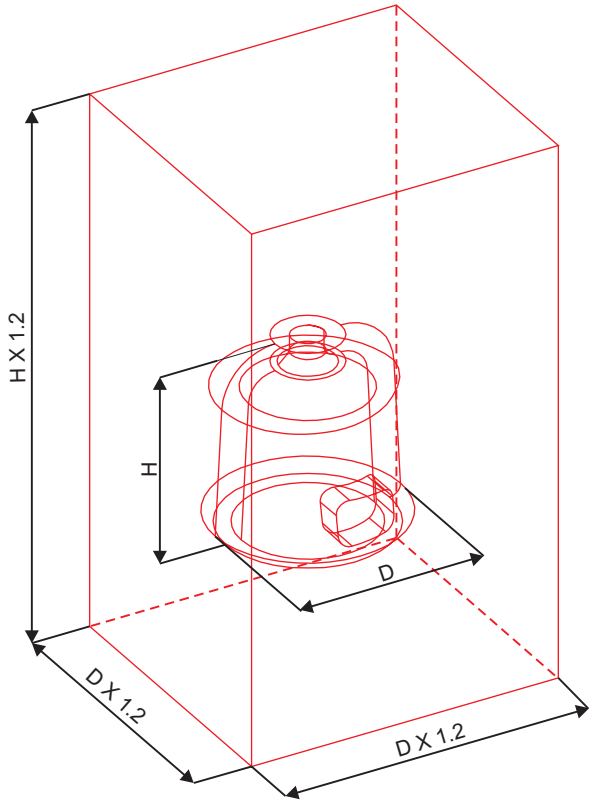

Figure 6-3: MOLDPREP >> WORKPIECE >> RECTANGULAR

WORKPIECE >> CYLINDRICAL

Define the workpiece as a cylinder around the model.

<CR> TO CONTINUE	DIAMETER = 76.00	HEIGHT = 41.000

The system presents default parameters for the diameter and height of the workpiece, but these parameters can be changed as required.

Press <CR> to accept the values and create the workpiece.

WORKPIECE >> FROM FILE

Import a workpiece from an external solid file. The following prompt is displayed:

ENTER FILE NAME Type or choose the pfm file that contains the model.

CONTINUE? YES NO Confirm the operation.

WORKPIECE >> BY PICK

In some cases you may want to define an irregular shaped workpiece. Once you have created the desired shape, using EXTRUDE for example, you need to define the shape as a workpiece. Using this option, once you have picked the shape and confirmed your selection, the shape is defined as the workpiece.

If the workpiece, created using any option, was used as the main body for a BOOLEAN/DIVIDE operation, it loses the properties of a workpiece. This means that:

- If you divided the workpiece by plane and tried to use the AUTODIVIDE operation, the AUTODIVIDE will not be successful. This is because the workpiece has already lost the properties of a workpiece.
- If a workpiece was created for some parts, you then have to use AUTODIVIDE for each part, step by step. However, after the first operation, the workpiece will lose the properties of a workpiece.
- If a workpiece was created for some parts, it may sometimes be convenient to divide this common workpiece by plane into smaller pieces and to work with them separately. However, the workpiece will lose the properties of a workpiece.

In each of the above cases, use the option WORKPIECE >> BY PICK to turn these bodies back into a workpiece again.

PICK OBJECT Pick an object as the workpiece.

OBJECT OK? YES / NO
YES Accept the picked object.
NO Pick another object as the workpiece.

EXECUTING The workpiece is created.

MOLDPREP >> PLACE

Change the placement of the model relative to the workpiece.

PICK OBJECT Pick the reference object to be moved to a different location according to X, Y and Z constraints.

PICK FACE/PLANE	CONSTR 1	CONSTR 2	CONSTR 3	- APPLY -
	OFFSET 1 = 0.0	OFFSET 2 = 0.0	OFFSET 3 = 0.0	

PICK FACE/PLANE Pick a face of the reference object.

Notes:

- If the point lies on the face of the reference object, you are prompted to choose the direction of the positive offset.
- A model/workpiece can only be placed once. If it has already been placed, use the EDIT function described on page 2-86 to relocate it.
- If it is difficult to pick edges, you can create datum planes and use them for reference during the placement of the model.

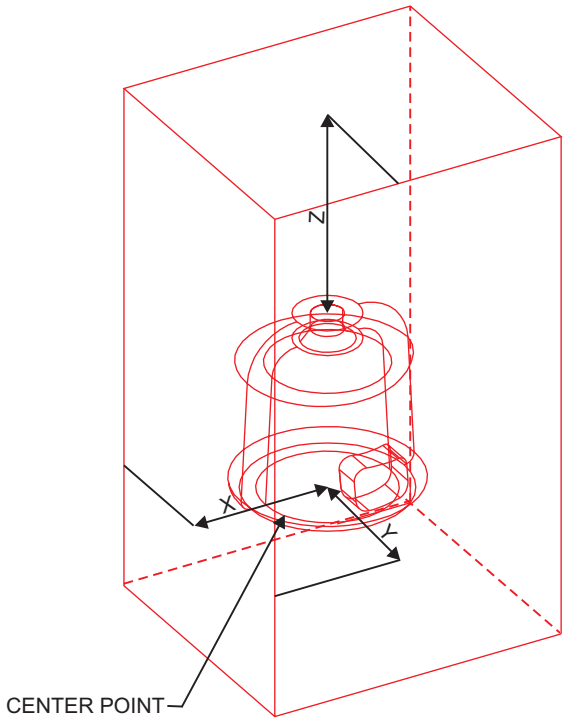

Figure 6-4: MOLDPREP >> PLACE

MOLDPREP: Modal Parameter Definitions

- + DIRECTION — Pick a point to indicate the positive direction of the vector.
- ANGLE = — Define the angle of rotation around the defined model direction in a counterclockwise direction.
- AXIS/EDGE — Choose an axis or straight edge along which will be used to define the slope.
- BASE POINT — Define the reference base point of the model. This point must not belong to the model being imported.
- CONSTR n = — Define the sides of the reference object (workpiece or datum plane).
- DIRECTION — Define the direction of the model that will coincide with +Z direction of the Mold file.
- FLIP — Reverse the indicated direction, if desired.
- OFFSET 1 = — Define the offset.
- ORIGIN — Indicate an origin point.

MOLDPREP: Usage Envelope

1. If the model in the source file is located in a particular level, and that level is not displayed in the mold file, you will not be able to see the model in Preview Mode. To avoid this problem, make sure that all necessary are displayed before the operation.
2. After creating a workpiece for a model, if you import another model it must be molded in another workpiece. A second workpiece cannot be created in this case. To avoid this problem, perform at least one dividing operation for the first workpiece, and then create the second workpiece.

❑

SEPARATE

Prepare the mold for separating by define a parting line and/or parting surface between the core and the cavity, and divide the mold.

Main Options:

PARTING LINE
PARTING SURFACE
DIVIDE

PARTING LINE Define main and internal parting lines, and merge a parting line(s) and curve(s) into one complex parting line.

When **creating** parting lines, only existing edges may be used. However, parting lines can be **merged** either with other parting lines or with datum curves.

PARTING SURFACE Calculate parting surfaces.

DIVIDE Separate the mold into core and cavity.

SEPARATE >> PARTING LINE

Define main and internal parting lines, and merge a parting line and curve into one parting line.

A parting line is based on edges. If there are not enough edges in the model, use MODIFY >> SPLIT (see page 2-98) to define additional edges and then define the parting line using SEPARATE >> PARTING LINE.

When **creating** parting lines, only existing edges may be used. However, parting lines can be **merged** either with other parting lines or with datum curves.

CREATE
MERGE
EDIT

PARTING LINE >> CREATE

Create a parting line.

PICK OBJECT — Choose the object to be parted.

If this object is open (for example after using SURFACES TO SOLID), a white arrow is displayed on the object, showing the outside direction. If this direction is incorrect, change it by selecting YES in the prompt.

MAIN DIRECTION — Choose the opening direction for the parting line. See MODIFY >> SPLIT >> BY SIL. LINE on page 2-114 for details of the direction options.

When the direction has been defined, you will be prompted to pick the type of loop.

EXTERNAL
INTERNAL ONLY

EXTERNAL — First create the external loops and then, if necessary, create the internal loops.

INTERNAL ONLY — Only one or a collection of internal loops may be created. In this case, the option AUTO ON/OFF, for the automatic detection of internal loops, will not appear in the table below.

When the kind of loop has been defined, you will be prompted to pick loops.

MAIN LOOP: PICK	CHAIN	AUTO OFF / AUTO ON	ERASE	APPLY

CHAIN
SINGLE
LOOP

Note:

- To remove a single edge from the selection, select the SINGLE option and pick the edge a second time.

MORE LOOPS ? YES/NO — If internal loop(s) are possible for the defined main direction, choose YES. You will receive the following prompt:

ADD LOOPS:PICK	CHAIN	AUTO OFF / AUTO ON	ERASE	APPLY

CHAIN
SINGLE
LOOP
ALL

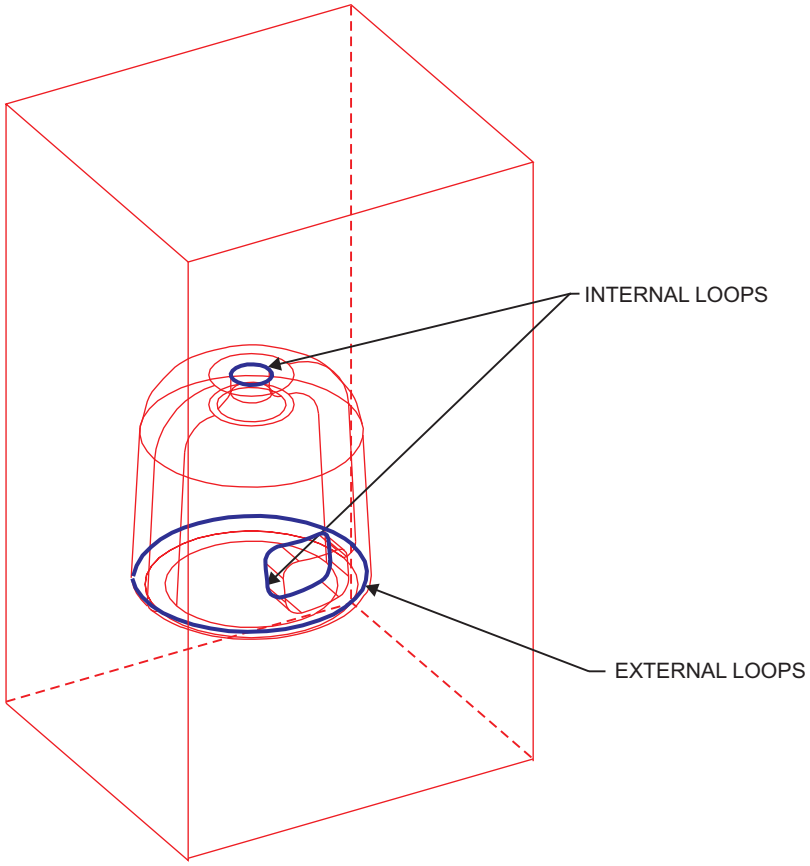

Figure 6-5: SEPARATE >> PARTING LINE

PARTING LINE >> MERGE

Merge two or more parting lines into one parting line, or combine reference curves with existing parting lines.

Note: • This option is useful when the parting line should be a combination of a predefined parting line(s) and a reference curve(s).

PICK PARTING LINE <PICK> the first parting line. This will be an external parting line after the merge operation. During the merge it is impossible to pick pre-defined internal loops before pre-defined external loops.

PICK & EXIT <PICK> additional parting lines or curves and press <EXIT>.

The parting lines/curves are merged into one parting line.

Note: • The order in which the parting lines are selected is important. The second one selected will always become the internal loop.

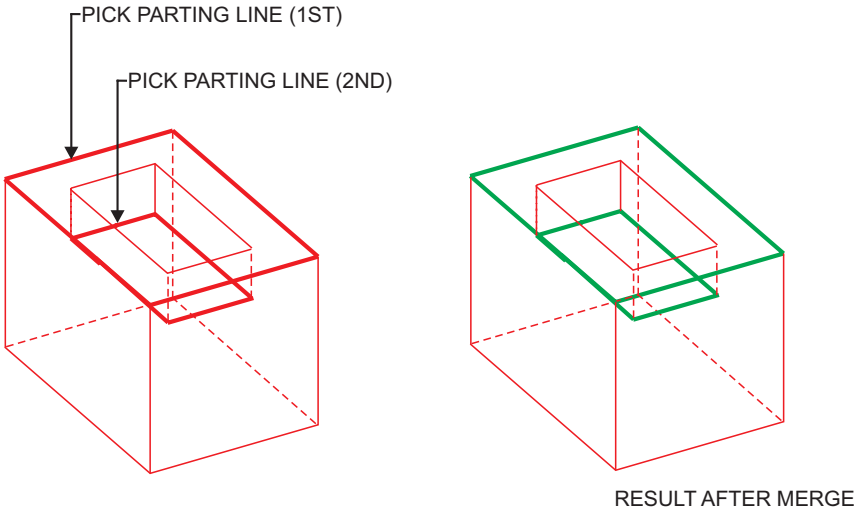

Figure 6-6: PARTING LINE >> MERGE

PARTING LINE >> EDIT

Edit a parting line. In some cases a parting line may need to be edited, for example, to change edges in loops or to delete unnecessary loops.

PICK PARTING LINE <PICK> the parting line to be edited. The parting line is displayed in cyan.

PICK PART. ITEM/EDGE	REMOVE	EXTRACT	RESET	APPLY

Pick the loop to be edited. You may need to pick additional edges to add to the loop (the new edges are displayed in cyan) or to unpick (remove) edges from the loop (unpicked edges are displayed in red).

Remember that picking an object (entity, part of the contour) while in an edit operation reverses the state of entities - existing entities are removed from the loop and new ones are added.

While a loop is still open, highlight points are displayed at the loop ends to show you the gap in the loop.

When the message LOOP IS CLOSED NOW is displayed, you can select another loop for editing.

SEPARATE >> PARTING SURFACE

Define a parting surface(s) for parting the mold, based on a parting line.

If there are several parting lines, or no parting lines exist, the following prompt is displayed:

PICK PARTING LINE Pick the parting line to be used to create the parting surface.

BY WIDTH
BY BLOCK

Note:

- If the system cannot generate a complete parting surface according to the available data, only segments of the parting surface are generated. It is the user's responsibility to complete the parting surface manually.

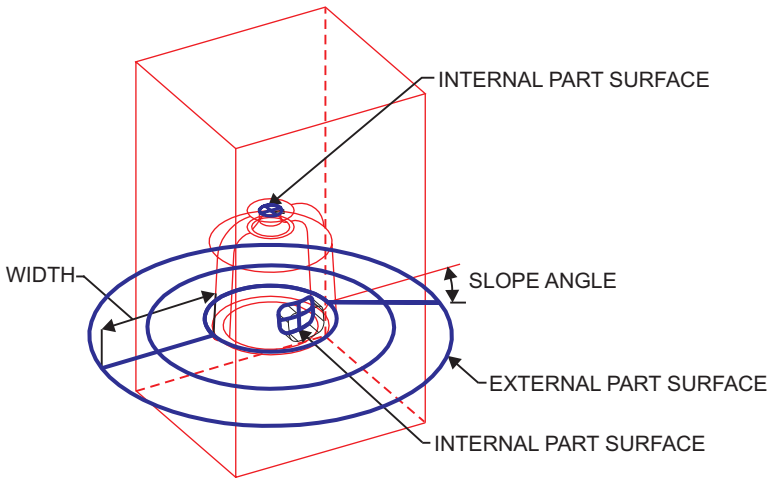

Figure 6-7: SEPARATE >> PARTING SURFACE

PARTING SURFACE >> BY WIDTH

Define the width, break fillet radius and slope angle for the parting surface.

PARTSURFACE PARAM/<CR>	PART SURFACE WIDTH=10.000	BREAK FILLET RADIUS=10.000
	SLOPE ANGLE=0.000	

Note:

- This option is useful if the user wants to create a complicated parting surface and combine it with a planar surface. After creating the parting surface with the desired width, and creating the planar surface, use CREATE >> BOOLEAN >> ADD to obtain the final parting surface.

PARTING SURFACE >> BY BLOCK

Define the break fillet radius and slope angle for the parting surface.

PARTSURFACE PARAM/<CR>	BREAK FILLET RADIUS = 10.000
	SLOPE ANGLE = 0.000

The width of the parting surfaces is equal to the diagonal of a workpiece maximum box.

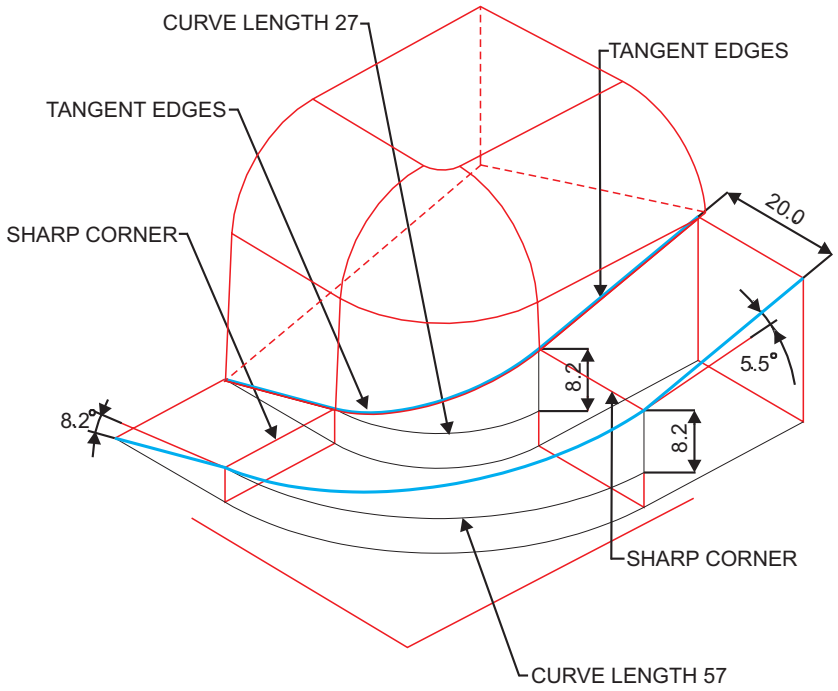

Figure 6-8: BREAK FILLET RADIUS

In most cases, parts to be molded have smooth edges and it is expected that the parting surface will also be smooth. But due to the fact that different boundary edges have different slope angles according to the XY plane, a 3D parting surface will have smooth faces near the parting line. However, at any distance from the parting line, sharp angles will automatically be created and will grow toward the periphery. After performing a divide operation by this surface, you will get a core and cavity with convex and concave edges on parting faces, and it will then be difficult to machine concave sharp angles on it.

Due to the width of the parting surface, in many places sharp corners will be created in places where two adjoining faces with different slopes meet. To avoid this problem, use the BREAK FILLET RADIUS parameter.

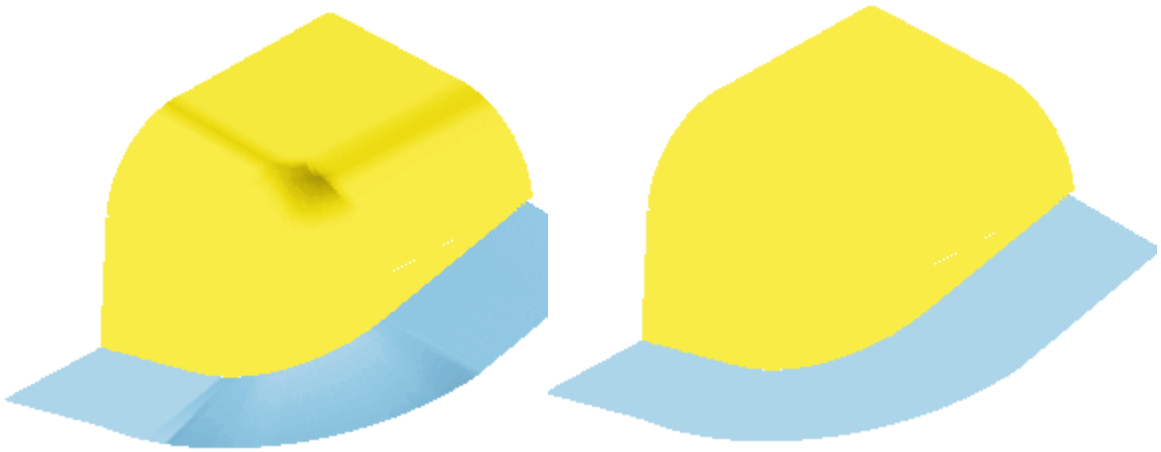

Figure 6-9: BREAK FILLET RADIUS

SEPARATE >> DIVIDE

Separate the mold (workpiece and model) into two pieces, the core and the cavity.

SPLIT ONLY
AUTO DIVIDE

DIVIDE >> SPLIT ONLY

Divide the model by parting line into two or more open solid objects (shells).

This option does not require a parting surface and may be useful in the following cases:

- If you wish to create core and cavity shells for NC operations without creating a parting surface.
- In difficult topological cases to check, before creating a parting surface, if the parting line has been created properly. If the SPLIT ONLY operation fails, this means that there is something wrong with the parting line and that it has to be edited (using SEPARATE >> PARTING LINE >> EDIT).
- In difficult topological cases, when SEPARATE >> DIVIDE >> AUTO DIVIDE fails, you can achieve good solid objects using SPLIT ONLY, BOOLEAN/DIVIDING the workpiece by parting surface and then stitching the shells of the model with the shells of the corresponding workpiece.

DIVIDE >> AUTO DIVIDE

Divide the mold by parting surface into closed solid objects.

PICK SURFACE & EXIT The system-defined surface is automatically highlighted in red. In cases where the parting surface has been edited, it will not be highlighted in red.

Pick the surface and <EXIT>.

Notes:

- In general, use this option to obtain the best results.
- If, in addition to the system-defined parting surface, there are also user-defined surfaces, <PICK> the additional surfaces, if required and <EXIT.

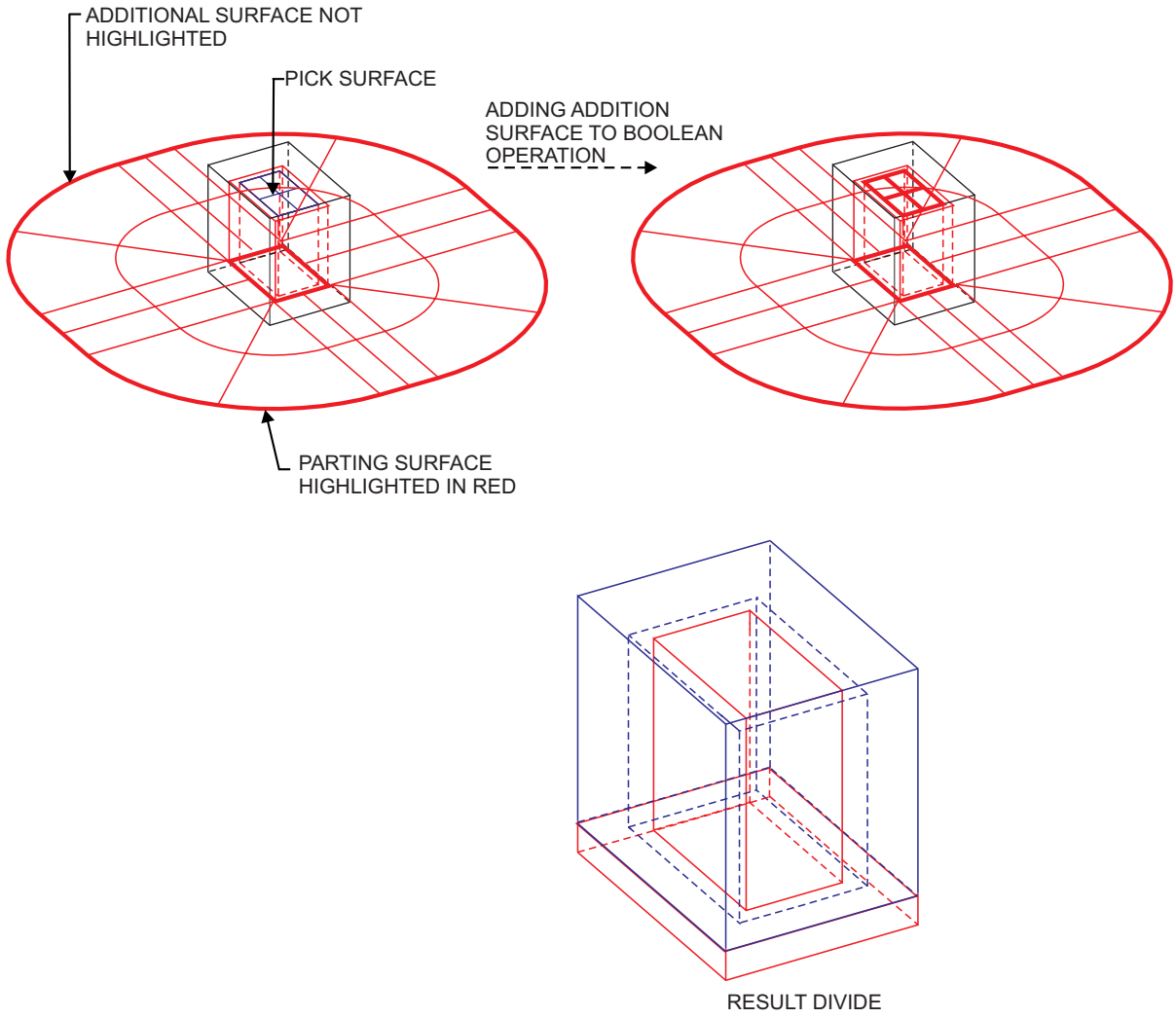

Figure 6-10: SEPARATE >> DIVIDE >> AUTO DIVIDE

SEPARATE: Modal Parameter Definitions

- APPLY — Confirm the changes and execute the operation.
- AUTO OFF / AUTO ON — Choose whether to activate automatic detection of all possible parting lines, according to the defined main direction. Candidates are displayed in blue. Pick the displayed candidates to confirm them as the main loop.
- AUTO OFF / AUTO ON — Choose whether to activate automatic detection of all internal loops for parting line. Candidates are displayed in purple.

  Pick the displayed candidates to confirm them as added internal loops.

  If the system detects two possibilities of internal loops, one is displayed in purple and the other in green. If the ALL picking method is selected, select the desired side for internal loops (purple or green). The system will use this side to define internal loops for all the holes.
- BREAK FILLET RADIUS — Enter the rounding radius for filleting sharp corners between faces of the parting surface.
- CHAIN / SINGLE / LOOP — Define the method of picking/unpicking edges.
- EXTRACT — Delete picked loops from the current parting line, but store them for the next parting line.
- PART.SURFACE WIDTH — Enter the width of the parting surface(s).
- SLOPE ANGLE — Enter the angle between the parting surface section and the normal to the main direction.
- REMOVE — Delete picked loops from the parting line.
- RESET — Restore the original parting line. The restore is effective for the current editing operation only. If you have edited a parting line and pressed APPLY and then continue editing, the restore will re-display the parting line as it was after the last APPLY operation.

SEPARATE: Usage Envelope

1. It is not possible to create a parting line in one operation using both solid edges and datum curves. To combine these into one parting line, create some segments based on the solid edges, and unite them with the datum curves by using the SEPARATE >> PARTING LINE >> MERGE.
2. Usually, after you have created a parting surface you can proceed directly to AUTO DIVIDE. However, in some cases the parting surface is not completed automatically. Completing the parting surface may be difficult because the surface may end at any point along an edge, not necessarily at a vertex. When this occurs, it is not possible to use these edges as a DRIVE or SECTION to complete the parting surface. Use MODIFY >> BREAK EDGE to update these edges. ❑

Appendix A
Working With Solid Assemblies

Full Path Name

In Version 10.0, the assembly application uses the full pathname of its part and sub-assemblies files in order to locate them in the file system (in the previous versions the relative locations of those files was used for this purpose). The benefit of the new system is an option to load an assembly from anywhere in the network without any special additional operations (Copy, Move, etc.).

Working with Assemblies on PC

Users who want the option to load and use assemblies of other group members, should make a few preparations:

a) Loading an assembly from different computers can be done only if the logical name of the files will be the same. Each user should have a network drive which is defined with that users system name. The users local drive (hard disk) should be connected to this network drive. The local disk to be shared should have 2 names such as C:\ and I:\.

   Use the options SHARE AS... and CONNECT TO in the file manager in order to share the local disk and to connect it to the network drive.

b) It is **highly recommended** to set the drives in all the computers, **in a permanent way**, to their respective owners local disks (e.g. if TOM gets network drive “T” all the users should connect their drive “T” to his local disk) .

c) After the connection, the “working directory” of Cimatron should be set to the new drive (e.g. if DAN gets network drive “I:\” he should set his “working directory” to be **I:\udd...**).

As the result of these operations, each user will be designated a unique drive in the entire network. This will make it possible to load and work on all assemblies in all the computers.

Working with Assemblies on UNIX

Due to the full path name mechanism the mount point should be fix. In the UNIX environment, there are mount systems that change the full pathname of a file from time to time or from different computers in the network. This can cause the addresses of the assembly components to become unsynchronized with the physical locations of the parts and sub-assemblies (the addresses are part of the assembly files). Such cases can cause problems when loading an assembly file in Version 10.0.

These mounting problems may be avoided in two ways:

a) WORKING WITH TCSHELL (**tcsh**) - In this shell the system does not include temporary directories (that were created by the mounting system) in the full pathname of the files. This option is recommended especially in auto-mount systems. This shell is shareware and can be download from the Internet.

b) MOUNT - There is an option to create a permanent MOUNT (hard and not soft mount) to the working directory. This operation can be done by the system administrator.

Note:

- There is an option to save 20%-30% of the loading time of the assembly by using the flag **-naslnk** (only in the Unix environment).

  When using this flag it is very important to avoid a situation where one assembly component has more than one full pathname in use (it can happen if the user creates more than one link to the same directory). Assembling one component (one file) twice in the same assembly, with different names, can destroy the assembly.

Assembly Utilities

An assembly file contains a **list** of the addresses (full pathnames) of all its parts and sub-assemblies. The assembly can be loaded only if those addresses fit the physical location of the files. Therefore, in order to copy or move the assembly to a new location in the network, transferring the files to the new location is not enough; the **list** also must be updated.

A group of external utilities (ASSEMBLY UTILITIES) helps the user perform common operations in a simple and fast way. The ASSEMBLY UTILITIES menu is number (4) in the main EXTERNAL menu (in Windows NT/95, the ASSEMBLY UTILITIES have an icon in the EXTERNAL window).

The concept of those utilities is very simple:

a) Choosing the operation.

b) Enter the assembly name (only the assembly name). In some utilities additional information must be entered, e.g. a new location in the copy utility.

1. Convert Assembly between Hardware (Consys) (or in PC: Convert Systems)

Prepare a Cimatron assembly (including all its components) that was created in one hardware environment for use in a different hardware environment.

2. Convert Assembly to New Version (or in PC: Convert Version)

Convert the assembly, including its components from Version 8 to Version 9.

Note:

- In the first loading of the assembly after the conversion, the system will still use relative pathnames.

Therefore the "WORKING DIRECTORY" must be set to the original directory (from Version 7). After the first SAVE operation, the assembly and its sub-assemblies can be loaded from anywhere in the file system.

3. Copy Assembly (or in PC: Regular copy)

Copy the assembly (including all of its components).

This option copies assemblies on the same platform (PC or UNIX).

4. Copy Assembly between hardware (or in PC: Advance Copy)

This option has two uses:

a) Enable copying of assemblies across platforms (P.C./UNIX).

b) Make the pathnames and locations identical, after moving or copying an assembly to a new location in the file system (not via the Assembly Utilities).

For more details and examples on the new Assembly utilities, see the Utilities manual.

Hardware Requirements and Configuration

Memory

Working with assemblies requires a lot of memory for some operations. To ensure good performance and continuous work, we recommend:

a)

For small and medium assemblies (up to 150-200 parts): 64[Mb] (32[Mb] is the minimum) RAM 300 [Mb] SWAPPING (VIRTUAL MEMORY IN PC.)

b)

For medium and large assemblies (more than 200 parts): 64[Mb] RAM 500 [Mb] SWAPPING (VIRTUAL MEMORY IN P.C.)

Note:

- In the UNIX platforms in order to use **all** of the defined swapping, a system parameter name in H.P. **maxdsize** (**datasize** & **stacksize** in SUN) should set the swapping size:

  maxdsize = SWAPPING-50 [Mb]
  stacksize = datasize 2000 [Mb] (default values)

  ❑

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