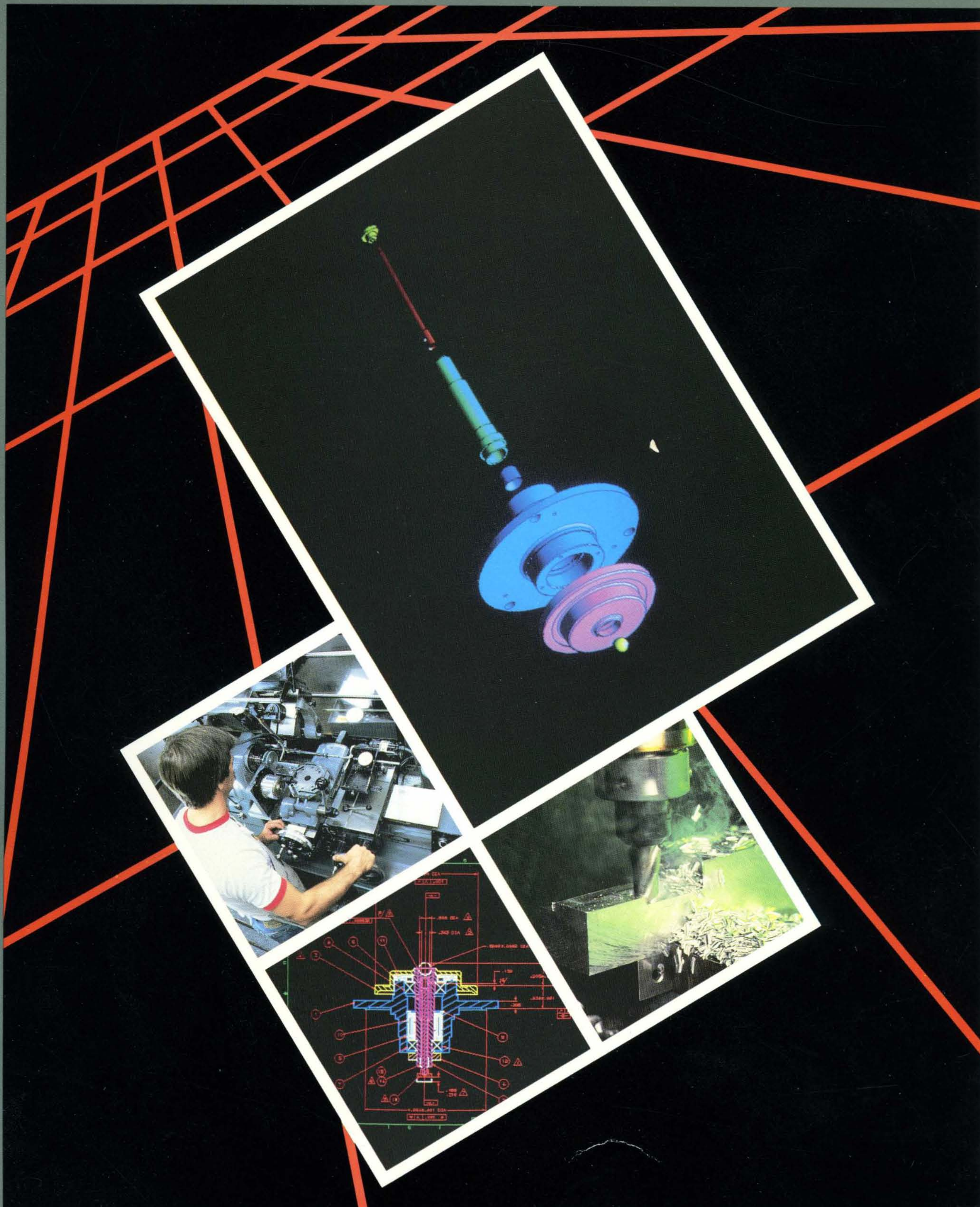
ICEM Advanced Design
for NOS

CONTROL DATA



Reference

60461430

ICEM Advanced Design
for NOS

Reference

This product is intended for use only as described in this document. Control Data cannot be responsible for the proper functioning of undescribed features and parameters.

Publication Number 60461430

Related Manuals

Background:

- User's Guide 60456940

Manual Set:

- Introduction and System Controls Reference 60457130
- Data Management Reference 60461410
- Basic Construction Reference 60461420
- Drafting Functions Reference 60461440
- Advanced Design Reference 60461430
- Numerical Control Reference 60461450
- GPL Reference 60462520

Additional:

- ICEM DDN Instant 60457140
- ICEM Tablet Overlays 60458080
- System Programmers Reference 60458560

Manual History

Revision C, printed in December 1985, documents ICEM DDN Version 1.6. The manual reflects changes in Composite Curve, Surface-Surface Intersection Curves, Surface Curves, Advanced Design Modals, and Surface Viewing, as well as various other technical and editorial changes.

Previous Revision	System Version	Date
A	1.53	November 1984
B	1.57	May 1985

Contents

About This Manual

This manual describes the CONTROL DATA® Integrated Computer aided Engineering and Manufacturing Design/Drafting/Numerical Control (ICEM DDN) software system. This volume documents the extended geometry capabilities of ICEM DDN including construction and modification of 3-D curves, surfaces, solids, cross section slices, and surface development.

Audience

This manual is a reference source for design engineers and drafting personnel who have already had initial training in the use of the ICEM DDN system. It is not intended to be a tutorial guide to ICEM DDN. New users should refer to the ICEM Design/Drafting User's Guide for a step-by-step introduction to the ICEM DDN system.

Organization

This manual is organized into seven chapters. Chapter 1 is a general menu overview, chapter 2 describes 15.1 ADVANCED DESIGN MODALS, chapter 3 describes menu 15.2 3-D CURVES, chapter 4 describes menu 15.3 SURFACES, chapter 5 describes menu 15.4 SOLIDS, chapter 6 describes menu 15.5 CROSS SECTION SLICE, and chapter 7 describes menu 15.6 SURFACE DEVELOPMENT.

This manual is part of the ICEM DDN manual set, described next.

The ICEM Design/Drafting Introduction and System Controls manual gives an overview of the major ICEM DDN concepts and describes menus 1 through 4 of the main menu: modals and fonts, blank/unblank operations, delete operations, and the file/exit sequence.

The ICEM Design/Drafting Data Management manual describes menus 5 through 8 of the main menu: special functions, data base management operations, input/output operations, and display control.

The ICEM Design/Drafting Basic Construction manual describes menus 9 through 14 of the main menu: point construction, line construction, arc construction, special curve construction, entity manipulation, and data verification.

The ICEM Design/Drafting Drafting Functions manual describes menus 16, 18, and 19: drafting functions, analysis, and SI/US resize.

The ICEM Numerical Control manual describes menu 17 NUMERICAL CONTROL, the numerical control programming part of ICEM DDN.

The ICEM GPL manual describes menu 5.13 GPL and the GPL programming language.

Conventions

In this manual, headings contain a series of numbers separated by periods. These numbers represent the selections available within the ICEM DDN menu hierarchy. The first number in the heading is the main menu choice, the second number is from the second-level menu, and so on. For example, menu choice 12.7.3 HEXAGON is from the third level of the menu hierarchy.

When the word 'system' is used, it refers to the ICEM DDN software system. When the Network Operating System is referred to, it is called either NOS or the operating system.

All text that the system displays is shown in uppercase letters and highlighted with a special typeface, as shown below:

```
PEN THICKNESS
1.ON
2.OFF
3.SET PEN THICKNESS
```

Additional Related Publications

You can find related information in the following publications:

Manual Title	Publication Number
Automatically Programmed Tooling System (APT IV)	17326900
Graphics Terminal Assist Version 1 User's Guide/Reference Manual	60476100
ICEM Design/Drafting GRAPL Programming Language	60461460
ICEM Engineering Data Library Version 1 Reference Manual	60459740
ICEM User-Defined Tablet Overlay	60457650
Network Products Interactive Facility Version 1 Reference Manual	60455250
Network Products Interactive Facility Version 1 User's Guide	60455260
NOS Version 1 Reference Manual, Volume 1	60435400
NOS 2 Reference Set, Volume 1, Introduction to Interactive Usage	60459660
NOS 2 Reference Set, Volume 2, Guide to System Usage	60459670
NOS 2 Reference Set, Volume 3, System Commands	60459680
UNIPLOT Version 3 User's Guide/Reference Manual	60454730
XEDIT Version 3 Reference Manual	60455730

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Submitting Comments

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St. Paul, Minnesota 55126-6198

Please indicate whether or not you would like a written response.

Introduction 1

Introduction 1

Using Menu 15

This menu is used to create and modify advanced geometric entities.

This menu contains the following menu choices:

```
ADVANCED DESIGN
1.ADVANCED DESIGN MODALS
2.3-D CURVES
3.SURFACES
4.SOLIDS
5.CROSS SECTION SLICE
6.SURFACE DEVELOPMENT
```

This menu has the following operations:

Menu Title	Description
15.1 ADVANCED DESIGN MODALS	Sets modal (control) parameters for created curve type, same point tolerance, and surface path display.
15.2 3-D CURVES	Creates three-dimensional curves including 3-D splines, surface curves, intersection curves, draft curves, composite curves, vectors, and Bezier curves.
15.3 SURFACES	Creates various surfaces including planes, spheres, cylinders, cones, 3-D tabulated cylinders, and projected surfaces.
15.4 SOLIDS	Creates solid figures including hexahedrons, spheroids, circular rods, toroids, and ellipsoids.
15.5 CROSS SECTION SLICE	Creates a point set curve along the intersection of a surface and a plane.
15.6 SURFACE DEVELOPMENT	Creates a flat pattern from existing surfaces and wraps or unwraps selected entities to or from a surface.

Menu 15.1: Advanced Design Modals 2

Menu 15.1: Advanced Design Modals 2

Using Menu 15.1

With this menu, you can set modal (control) parameters for these Advanced Design operations:

- Created curve type
- Same point tolerance
- Surface path display

Modals have default settings which are in effect unless you modify them.

The menu for this section is:

```
ADVANCED DESIGN MODALS
1.CREATED CURVE TYPE
2.SAME POINT TOLERANCE
3.SURFACE PATHS
```

The Advanced Design Modals menu offers these functions:

Menu Choice	**Function Description**
15.1.1 CREATED CURVE TYPE	Selects machining curve or 3-D spline as created curve type.
15.1.2 SAME POINT TOLERANCE	Sets the intersection point tolerance distance value.
15.1.3 SURFACE PATHS	Sets values for surface path lines and their points.

The following sections describe the choices in menu 15.1 ADVANCED DESIGN MODALS.

15.1 Advanced Design Modals

With this menu, you can set values and select options used in all 15. ADVANCED DESIGN menus. Current modal values are shown on the right side of the menu. The following is an example of Advanced Design Modals display:

```
ADVANCED DESIGN MODALS
1.CREATED CURVE TYPE          MACHINING
2.SAME POINT TOLERANCE        N.NNN
3.SURFACE PATHS
```

15.1.1 Created Curve Type

With this choice, you can set a modal for the curve type you wish to have created during curve creation operations such as surface-surface intersections. The default curve type is machining curve. The system displays:

```
CREATED CURVE TYPE
1.MACHINING
2.3-D SPLINE
```

Enter:

1 To create machining curves.

2 To create 3-D splines.

15.1.2 Same Point Tolerance

With this choice, you can set the modal for convergence (same point) tolerance of curves that are created during 15.2.3 SURFACE-SURFACE INTERSECTION CURVE and 15.2.2.2 OFFSET SURFACE CURVE.

In using 15.2.3 SURFACE-SURFACE INTERSECTION CURVE, the convergence or same point tolerance is the criteria used to halt the iterative refinement of intersection points. If the distance between points on each surface is less than the convergence tolerance, the iteration process is complete. When two points are within the same point tolerance, the system considers this an intersection point. Depending on the closeness of the surfaces near the intersection, this tolerance may or may not be an indication of how close the computed intersection point is from the actual intersection point. If the distance between the two surface points is less than the same point tolerance, the calculated intersection may not be close to the true intersection (refer to figure 2-1).

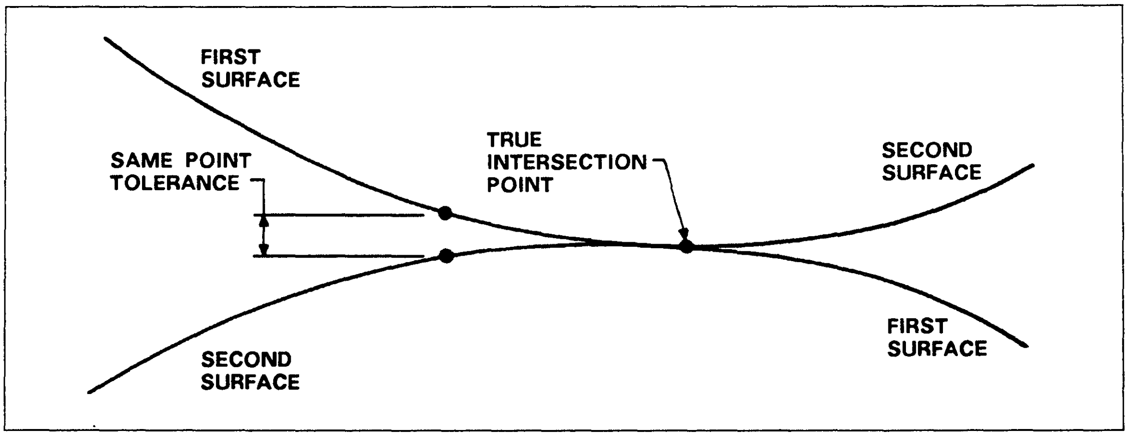


Figure 2-1. Edge View of Two Intersecting Surfaces

The same point tolerance is also used in 15.2.2.2 OFFSET SURFACE CURVE to control the accuracy of the created entities. This assures that the resulting offset curve does not deviate from the shape of the original curve by more than the same point tolerance setting.

For both Intersection Curves and Offset Surface Curves, the system displays:

`SAME POINT TOLERANCE = n.nnn`	Enter the intersection point tolerance distance. The default value is 0.13 mm (0.0010 inches).

The tolerance must be greater than zero. If you enter a value less than zero, the system returns you to the SAME POINT TOLERANCE prompt after displaying this message:

```
SAME POINT TOLERANCE MUST
BE GREATER THAN ZERO.
```

15.1.3 Surface Paths

With this choice, you can set a modal for the number of surface paths along two parameters of flow, u and v (refer to figure 2-2). The parameter pair u,v defines a unique point x,y,z on the surface of an object. U paths define horizontal paths along the curve, and v paths define vertical paths along the curve. You can describe the edges of a surface by fixing either the u or v parameter at its initial or final value and varying the other parameter from its initial or final value.

Figure 2-2 displays the same surface with various numbers of paths and points per path.

NOTE

15.3.18.2 MODIFY SURFACE PATHS can be used to change the number of paths and points per path on selected surfaces.

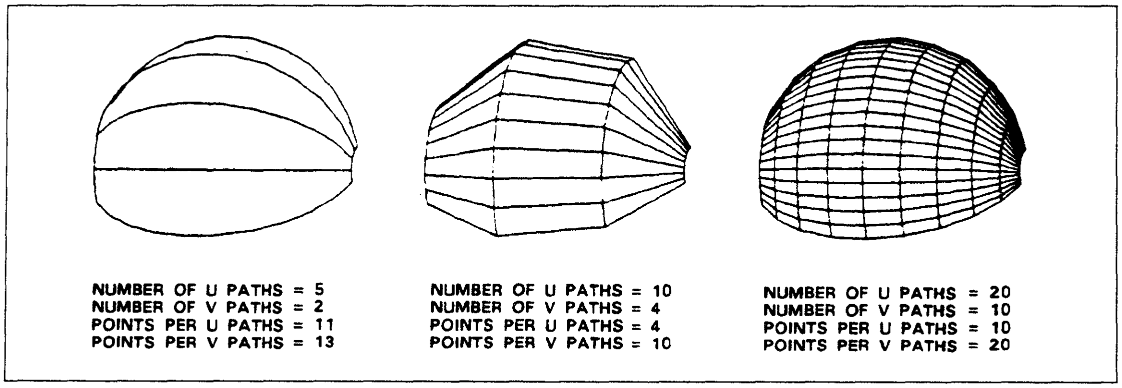


Figure 2-2. Surface Paths

You can use this modal to display the current values of surface paths lines and their points. You can also change these values for all newly created surfaces. The system displays:

`1.NUMBER OF U PATHS`	Enter the number of u paths (horizontal paths along the definition curve) to be used in defining the surface.
`2.NUMBER OF V PATHS`	Enter the number of v paths (vertical paths along the definition curve) to be used in defining the surface.
`3.POINTS PER U PATH`	Enter the number of points to be connected by lines along the u path.
`4.POINTS PER V PATH`	Enter the number of points to be connected by lines along the v path.

Since all four edges of the surface must be displayed for viewing purposes, the minimum values for the number of u and v paths is two. The minimum number of points per u and v paths is three. If you change the number of u or v paths to be less than two, the system automatically resets these values at two. If you change the number of points per u or v paths to less than three, the system automatically resets these values to three.

Some surface types have set (default) values for surface display that cannot be changed. The following surface types have default values of 3 for u paths and/or v paths: plane, tabulated cylinder, and ruled surface.

Menu 15.2: 3-D Curves 3

Menu 15.2: 3-D Curves 3

Using Menu 15.2

This menu enables you to create a variety of three-dimensional curves including 3-D splines, surface edge curves, surface-surface intersection curves, draft curves, composite curves, vectors, and Bezier curves.

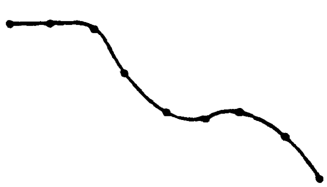
With this menu, you can create a machining curve. Surface edge curves, intersection curves, and draft curves are all considered machining curves. Because a machining curve in ICEM DDN is unique, you should be aware of some of its features.

In ICEM DDN, a machining curve is a collection of point coordinates representing an approximation of a curve related to one or more surfaces. It is displayed by connecting the points with line segments. It is evaluated, however, using an interpolating scheme to smooth out the curve between the data points. The technique used is a linear blending between two circular arcs over four consecutive data points. Use of this technique results in a curve that passes through the data points exactly, but remains smooth between these points. This smoothing is important when using iterative methods to find a normal or intersection point on a machining curve, and to assure that any surface created with a machining curve is smooth. For this reason, certain operations such as trimming may appear to yield incorrect results. Actually, these operations are evaluating the smoothed curve, which may vary from the displayed curve within its line segments. Because of the smoothing, you may find that acceptable results can be achieved by creating the machining curve with a coarser tolerance, thus using less computer time and less storage space.

The menu described in this chapter is:

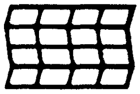
```
3-D CURVES
1.3-D SPLINE
2.SURFACE CURVES
3.SURFACE-SURFACE INTERSECTION CURVE
4.DRAFT CURVE
5.COMPOSITE CURVE
6.VECTOR
7.BEZIER CURVE
```

The 3-D Curves menu offers the following functions. An example of each curve is presented below its name and menu number.

Menu Choice	Function Description
15.2.1 3-D SPLINE 	Creates a three-dimensional spline that passes through points that you specify.

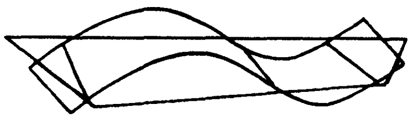
15.2.2 SURFACE EDGE CURVE

Creates a machining curve along a surface edge.



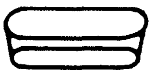
15.2.3 SURFACE-SURFACE INTERSECTION CURVES

Creates a curve or set of curves along the intersection(s) of two surfaces.



15.2.4 DRAFT CURVE

Creates a projection of a curve or set of curves in a specified direction either to a specified depth or onto a surface at a specified angle.

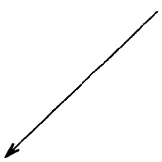


15.2.5 COMPOSITE CURVE

Creates a composite curve from a set of contiguous curves including lines, arcs, conics, two-dimensional splines, three-dimensional splines, and Bezier curves.

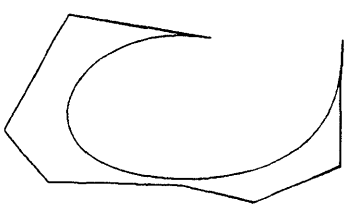
15.2.6 VECTOR

Creates or modifies vectors. Vectors are displayed as line segments that have magnitude and direction.



15.2.7 BEZIER CURVE

Creates or modifies Bezier curves. Bezier curves are parametrically-defined polynomials that are controlled by polygon points.



The remainder of this chapter describes the choices in menu 15.2.

15.2.1 3-D Spline

With this choice, you can create a three-dimensional spline that passes through a number of user-specified points. You can use any combination of four methods to define the points in the spline. You can specify any of four end conditions.

The menu described in this section is:

```
CONTROL POINT ENTRY MODE
1.SCREEN POSITION
2.KEY-IN
3.EXISTING POINT
4.CHAIN SELECT POINTS
```

Enter:

1 To use the graphics cursor for indicating positions on the screen. The D key is used to specify the desired depth.

INDICATE POSITION	Use the graphics cursor to indicate a position for a point on the spline.

2 To enter the coordinates of positions.

1.XT = n.nnnn 2.YT = n.nnnn 3.ZT = n.nnnn	Enter the transform coordinates of the position for the next point on the spline.

3 To use the graphics cursor to select an existing point.

INDICATE POINT	Use the graphics cursor to select the next point that is to be on the spline.

4 To automatically select a sequential set of existing points.

INDICATE POINT	Use the graphics cursor to select the first point of the set of points to be selected.
INDICATE POINT	Use the graphics cursor to select the final point of the set of points to be selected.

The system then sequentially selects the spline points. The point that appears closest to the previous point is repeatedly selected until the final point is reached. Reselection of the chain is then allowed by the system and the system displays:

SELECTION OK?	Enter: Y To accept the selected points as spline points. N To allow reselection of the chain of points.

The system redisplays the Control Point Entry Mode menu so you can select another method of entering points. An incorrectly selected point can be rejected by entering [.

If point selection has been terminated and fewer than three points have been specified, the system displays:

MINIMUM OF THREE POINTS REQUIRED FOR SPLINE CONTINUE ENTERING SPLINE POINTS?	Enter: Y To add to existing control points. N To discard existing points and return to control point entry mode to restart or leave three-dimensional splines.

If, in the process of defining a three-dimensional spline, you reach the limit of 55 control points, the system displays:

MAXIMUM NUMBER OF POINTS NOW ENTERED DEFINE SPLINE WITH CURRENT POINTS?	Enter: Y To go to the End Condition 1 menu to define the spline with currently selected control points. N To discard control points and return to control point entry to restart.

You automatically go to End Condition 1.

End Condition 1

After the point-entering process is complete, you choose a method to specify the curve shape at the end of the spline you select first. The system displays:

END CONDITION 1 1.TANGENT 2.RELAXED 3.CYCLIC 4.ANTICYCLIC	Enter: 1 To specify the starting direction of the spline. Continue to TANGENT. 2 To specify a second derivative of 0 at the first end. Continue to END CONDITION 2. 3 To specify the tangent and second derivative of the first end to be equivalent to the tangent and second derivative of the second end, respectively. 4 To specify the tangent and second derivative of the first end to be opposite to the tangent and second derivative of the second end, respectively.

The choices in this menu are described next.

Tangent

If you select 1.TANGENT from the End Condition 1 menu, the system displays:

```
TANGENT MODE
1.CURVE END
2.KEY-IN
3.VECTOR
```

Enter:

1 To enter the tangent from a selected curve and tangent.

INDICATE CURVE	Use the graphics cursor to select a curve near the desired spline end.

Then the system requests the direction of this tangent vector with respect to the end of the curve. The direction can be either away from the end of the selected curve or towards the middle of the selected curve.

INDICATE DIRECTION	Use the graphics cursor to indicate the direction of the tangent vector.

2 To enter the tangent for the spline.

1.DXT/DT = n.nnnn 2.DYT/DT = n.nnnn 3.DZT/DT = n.nnnn	Enter the desired tangent values.

3 To enter the tangent from a selected existing vector.

INDICATE VECTOR	Use the graphics cursor to select an existing vector to be used as the tangent.

NOTE

The tangent vector proceeds along the curve from the curve start to the end.

Continue to End Condition 2.

End Condition 2

If you select 2.RELAXED from the End Condition 1 menu, the system displays:

```
END CONDITION 2
1.TANGENT
2.RELAXED
```

Enter:

1 To specify the direction of the spline ending. Return to the TANGENT prompt to specify the tangent mode for the second end condition.

2 To specify a second derivative of 0 at the second spline end.

15.2.2 Surface Curves

With this menu, you can define a machining curve along the edge of a surface or manipulate spatial curves that lie within a given surface. You have the option of offsetting the spatial curves in the following ways:

- Along surface normals
- In a direction controlled by the curve and oriented by the surface
- Along the normal toward a second surface
- Within the surface

The offset distance may be a constant or variable value.

The system displays:

SURFACE CURVES 1.SURFACE EDGE CURVE 2.OFFSET SURFACE CURVE	Enter: 1 To create surface edge curves. 2 To offset existing curves that lie within a surface.

If you select 1.SURFACE EDGE CURVE from the Surface Curves menu, you can define a machining curve along the edge of a surface. (Refer to the description of the machining curve given earlier in Using Menu 15.2.) The system displays:

INDICATE SURFACE	Use the graphics cursor to select an existing surface.
CURVE POINTS = nn	Enter the number of points to define the surface edge curve. The default is 30.
INDICATE EDGE	Use the graphics cursor to indicate the edge of the surface on which to define the curve.

If you select 2.OFFSET SURFACE CURVE from the Surface Curves menu, you can manipulate spatial curves that lie within a given surface. You may offset these curves in any of four ways: along the surface normal, in a direction controlled by the curve and oriented by the surface, along the normal toward a second surface, or within the surface. The offset may be either a constant or a variable value. The system displays:

OFFSET DIRECTION 1.ALONG SURFACE NORMALS 2.IN VARIABLE DIRECTION 3.NORMAL TOWARD SECOND SURFACE 4.WITHIN THE SURFACE	Enter: 1 To offset along the surface normals. 2 To offset using local coordinates. 3 To offset toward a second surface along its normal vectors. 4 To offset the curve within the same surface.

If you select 1.ALONG SURFACE NORMALS from the Offset Direction menu, the system displays:

INDICATE SURFACE	Use the graphics cursor to select the surface containing the curve.
INDICATE CURVE NEAR START	Use the graphics cursor to pick the displayed surface curve to be offset. The end selected is taken to be the starting end of the offset operation.
OFFSET DISTANCE 1.START DISTANCE = n.nnn 2.END DISTANCE = n.nnn	Enter the desired starting and ending offset distances. The actual offset varies linearly between these values. The default value is zero.

Refer to figure 3-1 for an illustration of the Along Surface Normals offset operation.

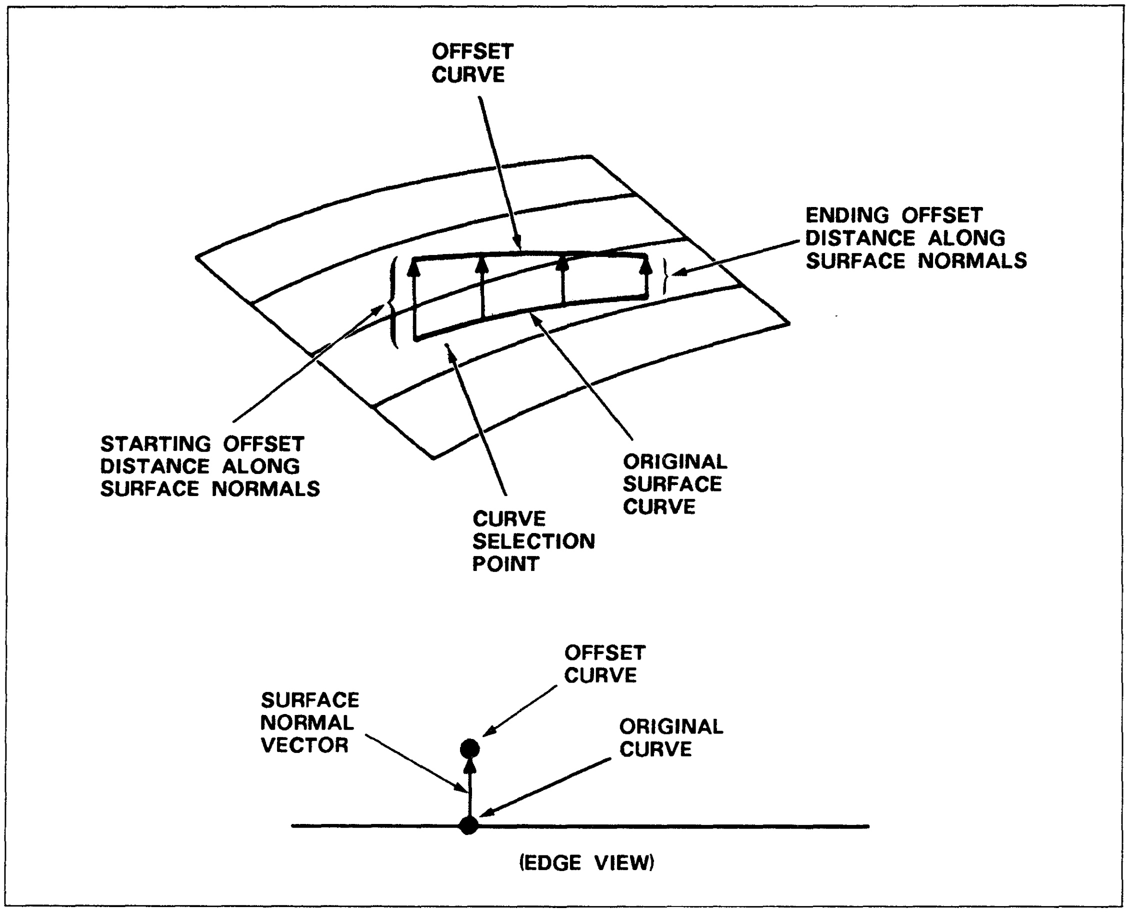


Figure 3-1. Offset Along Surface Normals

If you selected 2.IN VARIABLE DIRECTION from the Offset Direction menu, the system displays:

`INDICATE SURFACE`	Use the graphics cursor to select the surface containing the curve.
`INDICATE CURVE NEAR START`	Use the graphics cursor to pick the displayed surface curve to be offset. The end selected is taken to be the starting end of the offset operation.

`OFFSET PARAMETERS`
`1.START ANGLE    = n.nnn`
`2.END ANGLE      = n.nnn`
`3.START DISTANCE = n.nnn`
`4.END DISTANCE   = n.nnn`
	Enter the desired starting and ending offset angles and distances. The default value is zero.

This choice uses a local coordinate system to control the direction of the offset operation. In this local coordinate system, the following conditions apply:

- The x-axis is the surface normal vector.
- The y-axis is defined by the system as needed to make the coordinate system right-handed.
- The z-axis is the curve tangent vector.

The orientation of this system varies continuously along the curve being offset. The system interprets the start and end angle values you enter as the angles, in degrees, from the x-axis toward the y-axis in the x-y plane. These angles are used to define the actual offset vector. When the start and end angle values are different, the following conditions exist:

- The offset angle and offset vector direction will vary linearly between them, thus producing a smooth curve.
- The offset curve will spiral around the original curve.

The start and end values for the offset distance allows for a continuous variation of the offset distance along the offset vector. Refer to figures 3-2 and 3-3 for illustrations of this method.

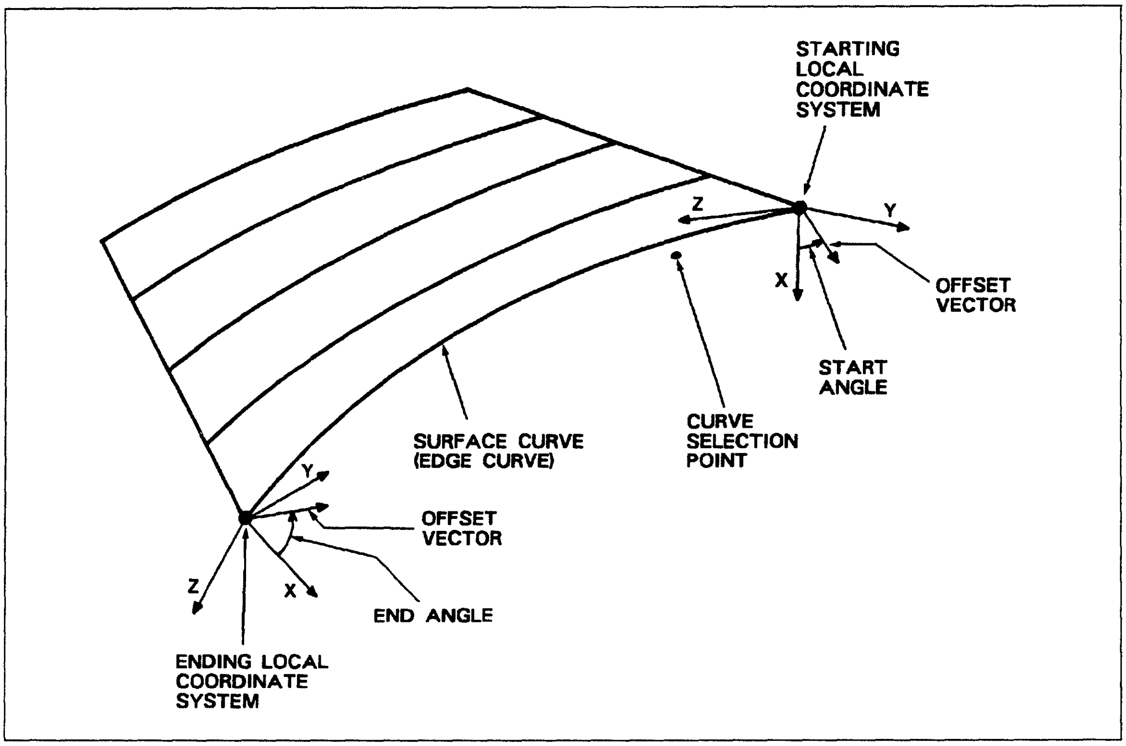


Figure 3-2. Local Coordinate Systems for Offset in Variable Direction

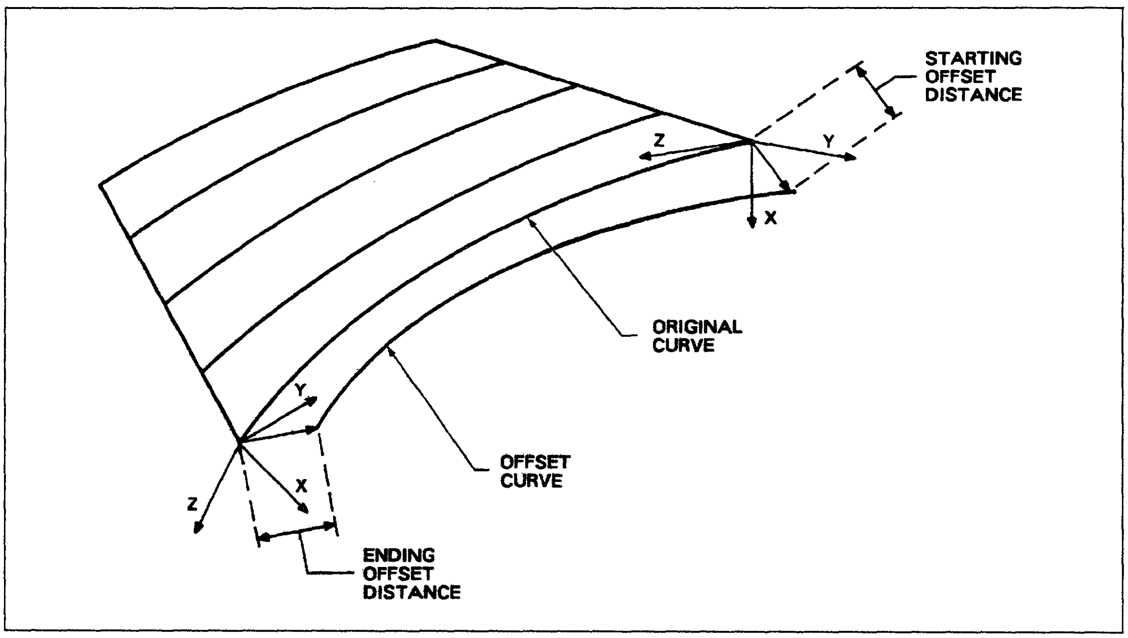


Figure 3-3. Offset in Variable Direction

If you selected 3.NORMAL TOWARD SECOND SURFACE from the Offset Direction menu, the system generates an offset curve toward a second specified surface along the second surface's normal vectors. The system does not verify whether or not curves lie on a surface. If they do not lie on the surface, unanticipated results may occur. The system displays:

`INDICATE SURFACE`	Use the graphics cursor to select the surface containing the curve.
`INDICATE CURVE NEAR START`	Use the graphics cursor to pick the displayed surface curve to be offset. The end selected is taken to be the starting end of the offset operation.
`INDICATE SECOND SURFACE`	Use the graphics cursor to select the surface controlling the offset direction.

`OFFSET DISTANCE`
`1.START DISTANCE = n.nnns.`
`2.END DISTANCE   = n.nnn`
	Enter the desired starting and ending offset distances. The actual offset will vary linearly between these values. The defaults are zero.

Figures 3-4 and 3-5 illustrate appropriate and inappropriate use of 15.2.2.2.3 NORMAL TOWARD SECOND SURFACE. In figure 3-4, the normal vector from each point on the surface curve can be dropped toward the second surface in such a way that each dropped point clearly lies on the second surface. In figure 3-5, points on the surface curve cannot be dropped normally to the second surface. The edge views of both cases illustrate the role of the second surface in defining the offset direction.

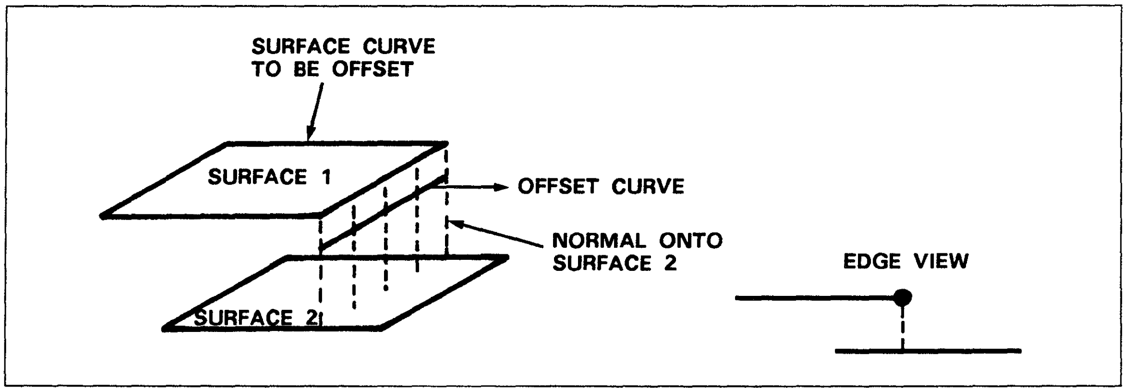


Figure 3-4. Appropriate Use of the Normal Toward Second Surface Operation

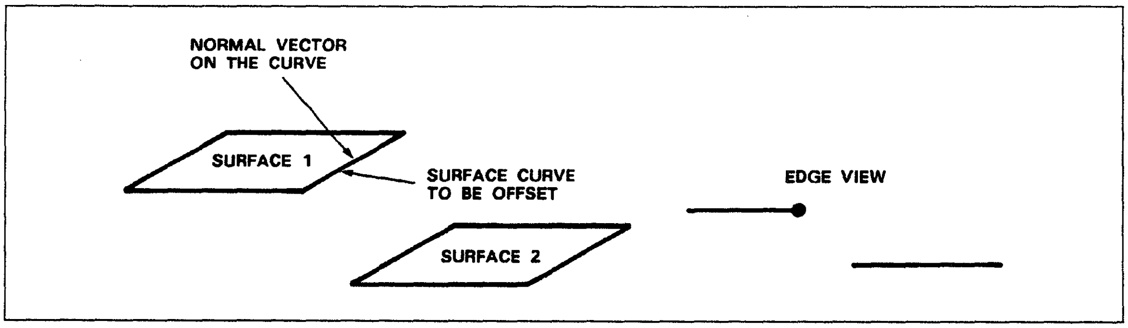


Figure 3-5. Inappropriate Use of the Normal Toward Second Surface Operation

If you selected 4.WITHIN THE SURFACE from the Offset Direction menu, the system offsets a curve within the same surface. The system displays:

`INDICATE SURFACE`	Use the graphics cursor to select the surface containing the curve.
`INDICATE CURVE NEAR START`	Use the graphics cursor to select the displayed surface curve to be offset. The end selected is taken to be the starting end of the offset operation.
`INDICATE OFFSET SIDE`	Use the graphics cursor to indicate the side of the curve on which the offset should be generated.

`OFFSET DISTANCE`
`1.START DISTANCE = n.nnn`
`2.END DISTANCE   = n.nnn`
	Enter the desired starting and ending offset distances. The actual offset varies linearly between these values. The default value is zero.

Figure 3-6 shows the Within the Surface offset operation:

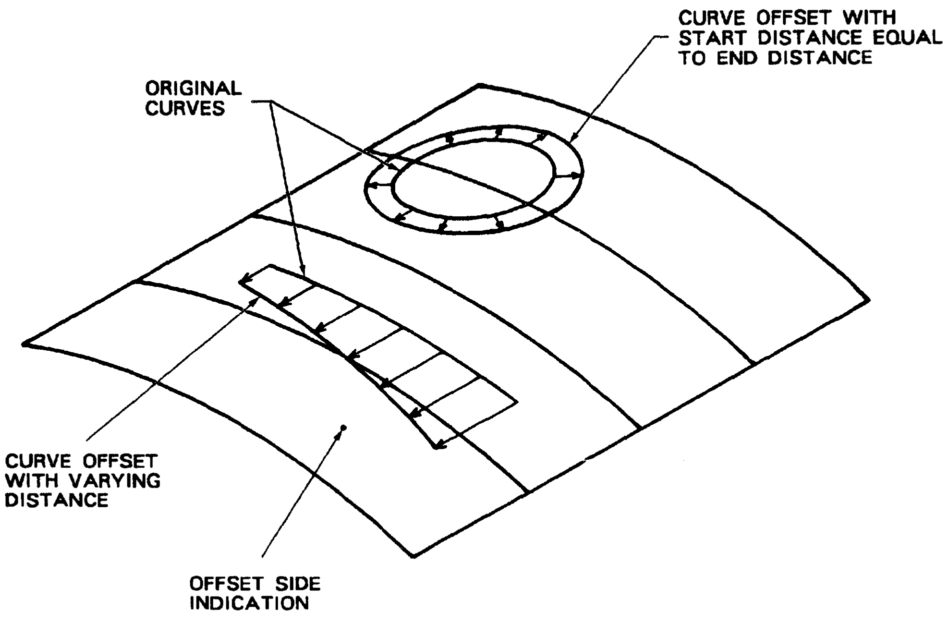


Figure 3-6. Offset Within the Surface

15.2.3 Surface-Surface Intersection Curve

With this menu, you may create surface-surface intersection curves which are formed at the intersection between two surfaces. This does not include surfaces that intersect at a single point or have a surface as an intersection.

The surface-surface intersection algorithm is adaptive. That is, the algorithm automatically adjusts itself to a given problem. The process of obtaining intersection curves can be thought of as a two-stage process. The first stage is finding the curves, the second is refining those curves.

To use surface-surface intersection, select two surfaces. The algorithm will try to produce intersection curves. Once the search for curves is complete, you are given the following operations options:

- If the curves are satisfactory, you may accept them.
- If no intersections were found or you suspect not all intersections were found, you may restart the search for curves. The system restarts using the information obtained in the first search but examines the surfaces more closely.
- If the created curves have points that are too widely spread, you may ask the program to refine the curves. The refined curves will have more points, so the curve will appear smoother.

These three options control the algorithm's actions. You may stop, search for more curves or refine the found curves.

Each time you ask the algorithm to search for more intersections or to refine the existing curves, the algorithmic parameters are more finely tuned. As this occurs, you can expect an increase in the amount of time and computer resources used. This refined searching can continue until you are sure there are no intersections, are satisfied with the intersection curve(s), or reach data storage limits.

You can control the convergence tolerance for which intersection points are found along a curve. This tolerance is set in 15.1.2 SAME POINT TOLERANCE.

The default curve type is a machining curve. You can select another type of curve with 15.1.1 CREATED CURVE TYPE.

When you use a three-dimensional spline as the curve type, it is possible that the intersection curve is a composite curve. This happens when the number of points exceeds the maximum allowed in one three-dimensional spline.

Machining curves cannot be members of composite curves. When you use the machining curve as the curve type, the system may create multiple machining curves.

Numerical control users should note that when the curve type is a machining curve, the normal vector of each intersection point will be the normal vector from the first surface selected.

The system displays:

SURFACE-SURFACE INTERSECTION CURVE INDICATE SURFACE INDICATE SURFACE	Select the two surfaces you want to intersect. The normal of machining curve is that of the first surface selected.

When you select a three-dimensional spline as the curve type, and the number of points created is greater than the limit for one three-dimensional spline, the system displays:

```
A COMPOSITE CURVE OF SPLINES WAS CREATED.
```

If more than one curve is found, the system displays:

```
MULTIPLE CURVES CREATED.
```

When you reach the data storage limits or when the surfaces obviously do not intersect, the system returns you to the INDICATE SURFACE prompt after displaying this message:

```
NO INTERSECTION CURVE FOUND.
SURFACE CANNOT BE EXAMINED ANY CLOSER.
```

If the surface data becomes lost, the system returns you to the INDICATE SURFACE prompt after displaying this message:

```
SURFACE DATA LOST.  PLEASE TRY AGAIN.
```

After you have indicated the selected surfaces, the system displays:

INTERSECTION CURVE OPTIONS Select the desired operation.

1.DONE
2.SEARCH AGAIN FOR CURVE
3.REFINE EXISTING CURVE

If you select 1.DONE from the Intersection Curve Options menu, the system returns you to the INDICATE SURFACE prompt.

If you select 2.SEARCH AGAIN FOR CURVE from the Intersection Curve Option menu, the system examines the surfaces more closely for intersection points. Use this choice if you suspect there are other curves that have not been found or if no curves were found and you believe there is an intersection.

If you select 3.REFINE EXISTING CURVE from the Intersection Curve Options menu, the system adds points along the intersection curve to create a smoother curve.

After refinement, the curves are displayed and the system returns you to the Intersection Curve Opions menu.

15.2.4 Draft Curve

With this menu choice you can project a curve or a group of curves to a depth or onto a surface with a specified draft angle. The three basic draft curve parameters are:

- Tolerance. A draft curve is calculated by breaking the curve(s) into a series of points that are then individually projected to the depth or onto the surface. The tolerance is applied to the resulting projected points to ensure the accuracy of the resulting curve(s). The tolerance must be greater than zero.

- Draft angle. The draft angle is measured as an offset angle in the projection direction. A draft curve is produced by projecting a curve or set of curves to a depth or onto a surface at this angle. The first part of the draft curve is located on the same side of the first entity selected that is specified in response to the INDICATE SIDE prompt. A zero degree angle is parallel to the projection direction. Allowable angles are zero degrees up to, but not including, 90 degrees.

  The application of this angle in the projection operation is based on a side indication entered by the user after the side(s) to be projected have been selected. The side of the projection is then fixed for all the selected curves based on the order of their selection.

- Depth. The depth is the zt value of a plane that is parallel to the current workplane. If 1.PROJECT TO DEPTH is chosen, the curve is projected to this plane at the specified draft angle. This is not a relative displacement.

Refer to menu 1.15.2 in the ICEM Design/Drafting Introduction and System Controls manual to set the display to transform or model coordinates. The draft curve feature first prompts you to generate a draft curve at a specified depth or to a specified surface.

PROJECTION TYPE 1.PROJECT TO DEPTH 2.PROJECT TO SURFACE	Enter: 1 To generate a draft curve at a specified depth. 2 To generate a draft curve on a specified surface.

15.2.4.1 Project to Depth

If you select 1.PROJECT TO DEPTH from the Projection Type menu, the system displays:

```
DRAFT PARAMETERS
1.TOLERANCE = n.nnnn
2.ANGLE     = n.nnnn
3.DEPTH     = n.nnnn
```

Enter the desired tolerance, draft angle, and depth or accept the default values with].

If the tolerance value is out of range, the system displays:

```
TOLERANCE MUST BE GREATER THAN ZERO
```

If the draft angle is out of range, the system displays:

```
DRAFT ANGLE VALUES CAN RANGE FROM ZERO
UP TO 90 DEGREES
```

If either error occurs, you are returned to the DRAFT PARAMETERS prompt.

After you have selected the draft parameters values, the system displays:

```
SCREEN SELECT MODE
1.SINGLE
2.CHAIN
  ⋮
```

Use the entity selection procedure to select as many as 240 curves for projection onto the previously selected surface or to the specified depth.

If you select composite curves, the curves are expanded into their constituent curves. This may push the total curve count over the maximum limit of 240 curves. If this occurs, the system displays:

```
TOO MANY CURVES FOR ONE PROJECTION
```

You are then returned to the SCREEN SELECT MODE to reselect the curves.

If the specified draft angle is greater than zero degrees, the system displays:

```
INDICATE SIDE
```

Use the graphics cursor to indicate the side of the first entity selected on which the draft curve is to be generated.

The system now performs the projection operation. If you selected to project on to a surface and none of the selected curves touch the surface, the system displays:

```
NO CURVES PROJECTED
```

15.2.4.2 Project to Surface

If you select 2.PROJECT TO SURFACE from the Projection Type menu, the system displays:

```
PROJECTION DIRECTION
1.ALONG ZT AXIS
2.ENTER VECTOR
3.EXISTING LINE OR VECTOR
```

Enter:

1 To cause the projection to occur along the zt-axis as it would when projecting to a depth.

2 To allow you to enter the components of the desired projection vector.

3 To allow you to project the selected curves along an existing line or vector onto the surface.

If you select 1.ALONG ZT AXIS from the Projection Direction menu, the system continues to the DRAFT PARAMETERS prompt.

If you select 2.ENTER VECTOR from the Projection Direction menu, the system prompts:

```
DEFINITION SPACE
1.TRANSFORM COORDINATES
2.MODEL COORDINATES
```

Enter:

1 To enter the vector using the transform coordinate system.

2 To enter the vector using the model coordinate system.

If you select 1.TRANSFORM COORDINATES from the Definition Space menu, the system prompts:

```
VECTOR COMPONENTS
1.IT = n.nnnn
2.JT = n.nnnn
3.KT = n.nnnn
```

Enter the desired it, jt, and kt vector values.

If you select 2.MODEL COORDINATES from the Definition Space menu, the system prompts:

```
VECTOR COMPONENTS
1.I = n.nnnn
2.J = n.nnnn
3.K = n.nnnn
```

Enter the desired i, j, and k vector values.

NOTE

The values entered for either coordinate system must define a vector of nonzero length. If a vector of zero length is defined, the system displays:

```
VECTOR CAN NOT BE ZERO LENGTH
```

You are then returned to the VECTOR COMPONENTS prompt.

If you select 3.EXISTING LINE OR VECTOR from the Projection Direction menu, the system prompts:

INDICATE ENTITY	Use the graphics cursor to select any displayed line or vector.

After you have defined the projection direction, the system displays:

DRAFT PARAMETERS 1.TOLERANCE = n.nnnn 2.ANGLE = n.nnnn 3.DEPTH = n.nnnn	Enter the desired tolerance and draft angle or accept the default values with].

If the tolerance value is out of range, the system displays:

```
TOLERANCE MUST BE GREATER THAN ZERO
```

If the draft angle is out of range, the system displays:

```
DRAFT ANGLE VALUES CAN RANGE FROM ZERO
UP TO 90 DEGREES
```

If either error occurs, you are returned to the DRAFT PARAMETERS prompt.

After you have selected the draft parameters values, the system displays:

INDICATE SURFACE	Use the graphics cursor to select the surface on which to project the selected curve.
SCREEN SELECT MODE 1.SINGLE 2.CHAIN ⋮	Use the entity selection procedure to select as many as 240 curves for projection onto the previously selected surface or to the specified depth.

If you select composite curves, the curves are expanded into their constituent curves. This may push the total curve count over the maximum limit of 240 curves. If this occurs, the system displays:

```
TOO MANY CURVES FOR ONE PROJECTION
```

You are then returned to the SCREEN SELECT MODE to reselect the curves.

If the specified draft angle is greater than zero degrees, the system displays:

INDICATE SIDE	Use the graphics cursor to indicate the side of the first entity selected on which the draft curve is to be generated.

The system now performs the projection operation. If none of the selected curves touch the surface, the system displays:

```
NO CURVES PROJECTED
```

Refer to figure 3-7 for examples of a draft angle, draft curve, and surface normal vector.

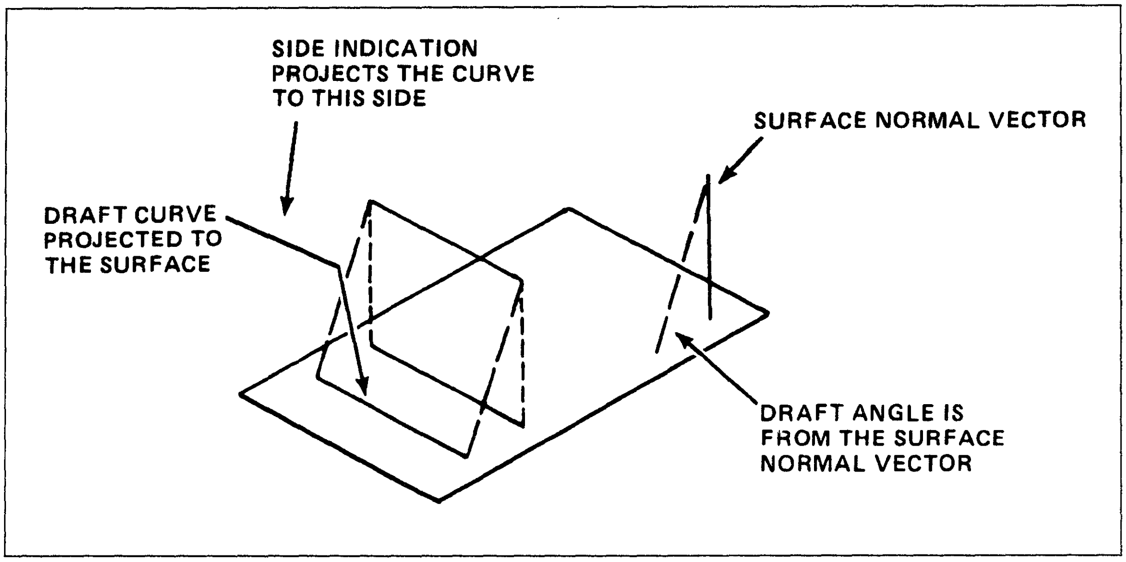


Figure 3-7. Draft Curve, Draft Angle, and Surface Normal Vector Examples

NOTE

Curves that are projected with a nonzero draft angle will remain continuous only if tangent vectors vary continuously over the curve. Refer to figure 3-8 for examples of curves with continuous and noncontinuous tangent vectors.

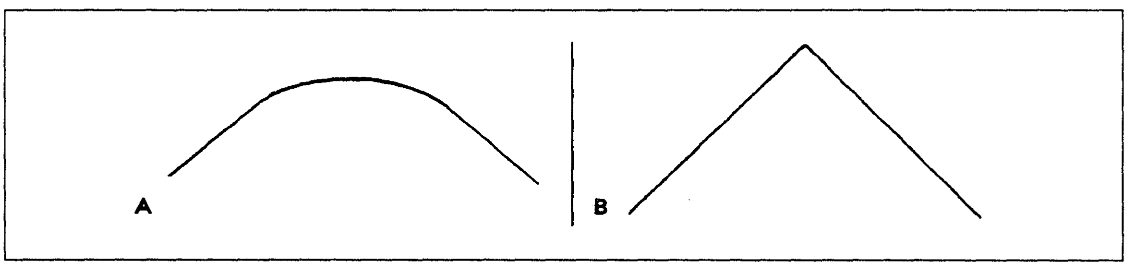


Figure 3-8. Curves Where the Tangent Vector: (a) Varies Continuously and (b) is Discontinuous at the Corner

The tolerance value is applied directly to the projected curve to ensure accurate projection. Projected curves passing across surface edges will start or end within 0.000001 units of the actual surface edge.

Projecting an entity that passes over a surface edge a number of times will result in multiple machining curves, one for each portion on the surface. A maximum of 255 machining entities can result from a single draft curve operation.

15.2.5 Composite Curve

With this choice, you can create a composite curve from a set of contiguous curves. The curves of the composite curve must be selected end to end in sequential order. You cannot refer to the subcurves of a composite curve individually, but only as the whole composite curve. The allowable entities in composite curves are lines, arcs, conics, two-dimensional splines, three-dimensional splines, and Bezier curves.

Composite curves have these characteristics:

- Composite curves are single entities.
- Composite curves are displayable entities and can be assigned display attributes, such as level.
- A composite curve cannot be a member of another composite curve.
- The number of allowable entities incorporated in a composite curve ranges from a minimum of 2 to a maximum of 100.
- Composite curves are selectable entities but the individual entities incorporated in them cannot be selected.
- Composite curves can be deleted without deleting the individual entities incorporated in them (refer to 3 DELETE in the ICEM Design/Drafting Introduction and System Controls manual). When the composite curve is deleted, the individual entities retain the display attributes of the composite curve. The original display attributes of the individual entities are lost.
- Composite curves behave as single entities when these operations are applied to them: TRANSLATE, ROTATE, MIRROR, SCALE, DUPLICATE, or any combination of these operations.

When you begin using the Composite Curve operation, the system displays:

`INDICATE ENTITY`	Use the graphics cursor to screen select the curves to be selected.

If you want to use another method of selection, enter CTRL-E to receive the following menu:

`ENTITY SELECTION`
`1.SCREEN SELECT`
`2.CHAIN`
⋮
	Use the entity selection procedure to select the contiguous curves that are to form the composite curve. Selected curves must be connected from end to end. You may select a maximum of 100 curves.

After you select the curves and enter], the system defines the composite curve. You are then returned to the Screen Select Mode menu to continue with the composite curve operation.

If you have ended the curve selection procedure and fewer than two curves have been specified, the system displays:

```
COMPOSITE CURVE NOT DEFINED.
AT LEAST TWO CURVES REQUIRED.
```

If the selected curves are not contiguous, the system displays:

```
COMPOSITE CURVE NOT DEFINED.
CURVES ARE NOT CONTIGUOUS IN THE
ORDER OF SELECTION.
```

If the selected curves are members of groups, the system displays:

```
CURVES CANNOT BE IN GROUPS
```

If you selected fewer than two curves or the selected curves were not contiguous, you are returned to the Entity Selection menu to reselect the curves.

You are not allowed to select more than 100 curves for incorporation in a composite curve. If you select more than 100 curves, the system displays:

```
MAXIMUM OF 100 ENTITIES HAS
BEEN SELECTED.
```

15.2.6 Vector

With this choice, you can create new vectors or modify existing vectors. A vector is a single line segment that has magnitude and direction. Its length represents its magnitude. The direction of a vector is from its tail to its head, which has barbs on it. Vectors are always defined in model coordinate space.

The following menu is described in this section:

```
 VECTOR
 1.SCREEN POSITION
 2.KEY-IN
 3.2 POINTS
 4.SURFACE UNIT NORMAL
 5.SCALAR TIMES VECTOR
 6.CROSS 2 VECTORS
 7.NORMALIZE VECTOR
 8.LENGTH AT GIVEN ANGLE
 9.INTERSECTION OF 2 PLANES
10.SUM OR DIFF OF 2 VECTORS
11.PT AT ANGLE TO LINE/VECTOR
12.MODIFY/REPLACE
13.REVERSE SURFACE NORMAL
```

The choices in this menu are described next.

15.2.6.1 Screen Position

With this choice, you can create a vector between two positions on the screen. The zt value for each end of the vector is at the depth in effect at the time the screen position is indicated.

INDICATE POSITION	Use the graphics cursor to indicate the location of the tail of the vector to be created.
INDICATE POSITION	Use the graphics cursor to indicate the location of the head of the vector.

15.2.6.2 Key-In

With this choice, you can create a vector whose endpoints are specified by model coordinates.

1.X1 = n.nnnn 2.Y1 = n.nnnn 3.Z1 = n.nnnn	Enter the coordinates of the tail of the vector to be created.
4.X2 = n.nnnn 5.Y2 = n.nnnn 6.Z2 = n.nnnn	Enter the coordinates of the head of the vector.

To enter the z-coordinates, you must first make or accept (by entering RETURN) all other entries, and then open the z-coordinate parameters by entering 3 or 6.

15.2.6.3 2 Points

With this choice, you can create a vector whose endpoints are two existing points.

INDICATE POINT	Use the graphics cursor to select an existing point for the tail endpoint of the vector.
INDICATE POINT	Use the graphics cursor to select an existing point for the head endpoint of the vector.

15.2.6.4 Surface Unit Normal

With this choice, you can create a vector that is a surface normal at a specified point on a surface. The vector is defined with its tail at the point on the surface and its head at one unit from the tail.

INDICATE POINT	Use the graphics cursor to select a previously defined surface normal point or surface pierce point (refer to 9.15 SURFACE POINTS in the ICEM Design/Drafting Basic Construction manual).

If the selected point is not a surface normal point or a surface pierce point, the system redisplays the INDICATE POINT prompt.

15.2.6.5 Scalar Times Vector

With this choice, you can create a vector that is a scalar multiple of another vector. The tail of the new vector coincides with the tail of the original vector. The head of the new vector is in the same direction as the head of the original vector, if the scalar is a positive number; it is 180° from the head of the original vector, if the scalar is a negative number. The original vector remains unchanged.

`INDICATE VECTOR`	Use the graphics cursor to select a vector.
`SCALAR = n.nnnn`	Enter the scalar value. Default is 50.8 mm (2.0 in).

15.2.6.6 Cross 2 Vectors

With this choice, you can create a vector that is the geometric cross product of two existing vectors (first vector cross second vector). The tail of the new vector coincides with the tail of the first vector selected.

`INDICATE VECTOR`	Use the graphics cursor to select two vectors. These vectors do not need to have a common tail.

15.2.6.7 Normalize Vector

With this choice, you can create a new vector by normalizing an existing vector. The tail of the new unit-long vector coincides with the tail of the original vector. The head of the new vector is one unit from the tail, in the same direction as the head of the original vector. The original vector remains unchanged.

`INDICATE VECTOR`	Use the graphics cursor to indicate the vector to be normalized.

15.2.6.8 Length at Given Angle

With this choice, you can create a vector with a specified length, at a specified angle from the horizontal, with its tail at a specified point.

`INDICATE POINT`	Use the graphics cursor to select an existing point to use as the tail of the vector.
`1.ANGLE = n.nnnn`	Enter (in degrees) the angle between the positive xt- axis and the new vector (a positive angle is measured in a counterclockwise direction).
`2.LENGTH = n.nnnn`	Enter the length of the new vector.

15.2.6.9 Intersection of 2 Planes

With this choice, you can create a vector along the intersection of two nonparallel planes. The order in which the planes are selected determines the direction of the vector.

INDICATE SURFACE	Use the graphics cursor to select first one plane and then another plane. Selecting the planes in the opposite order reverses the direction of the vector.

15.2.6.10 Sum or Diff of 2 Vectors

With this choice, you can create a vector by taking the geometric sum or difference of two existing vectors. The tail of the new vector coincides with the tail of the first vector selected.

SELECT OPERATION 1.SUMMATION 2.DIFFERENCE	Enter: 1 To add two vectors geometrically. 2 To subtract two vectors geometrically. The system defines the new vector as the first selected vector minus the second selected vector.
INDICATE VECTOR	Use the graphics cursor to select two existing vectors. These vectors do not need to have a common tail.

15.2.6.11 Pt at Angle to Line/Vector

With this choice, you can create a vector with a specified length, at a specified angle from an existing line or vector, with its tail at a specified point.

INDICATE POINT	Use the graphics cursor to select an existing point to use as the tail of the vector.
INDICATE 2-D CURVE	Use the graphics cursor to select an existing line or vector to use as the base for the angular measurement.
1.ANGLE = n.nnnn	Enter (in degrees) the angle between the indicated curve and the new vector (a positive angle is measured in a counterclockwise direction).
2.LENGTH = n.nnnn	Enter the length of the new vector.

15.2.6.12 Modify/Replace

With this choice, you can change the parameters defining an existing vector. Either the angle or the length or both can be changed.

INDICATE VECTOR	Use the graphics cursor to select the vector to be modified.

If you use 15.2.6.1 SCREEN POSITION or 15.2.6.2 KEY-IN to define the vector, then you can modify the vector by using the cursor to indicate the new location of the vector or by entering the new coordinates of the two endpoints.

POINT ENTRY MODE 1.SCREEN POSITION 2.KEY-IN	Enter: 1 To screen position tail and head position of new vector (refer to menu 15.2.6.1). 2 To key in new tail and head positions of new vector (refer to menu 15.2.6.2).

If you use 15.2.6.5 SCALAR TIMES VECTOR to define the vector, then you modify the vector by changing the scalar value.

1.SCALAR = n.nnnn	Enter the scalar value. Defaults to 50.8 mm (2.0 in).

If you use 15.2.6.8 LENGTH AT GIVEN ANGLE to define the vector, then you can modify the vector by changing the angle or the length or both.

1.ANGLE = n.nnnn 2.LENGTH = n.nnnn	Enter new values for the angle and length of the vector.

If you enter [, the system will prompt you to indicate an existing point for replacing the tail of the vector. You can either keep the same angle and length or change them.

INDICATE POINT	Use the graphics cursor to select an existing point as the tail of the vector.
1.ANGLE = n.nnnn 2.LENGTH = n.nnnn	Enter new values for the angle and length of the vector.

If you use 15.2.6.11 PT AT ANGLE TO LINE/VECTOR to define the vector, then you can modify the vector by changing the angle or the length or both.

1.ANGLE = n.nnnn 2.LENGTH = n.nnnn	Enter new values for the angle and length of the vector.

If you enter [, the system will prompt you to indicate a two-dimensional curve for replacing the old one used as the base for the angular measurement. Then, you can either keep the same angle and length or change them.

`INDICATE 2-D CURVE`	Use the graphics cursor to select an existing line or vector to use as the base for the angular measurement.
`1.ANGLE = n.nnnn` `2.LENGTH = n.nnnn`	Enter new values for the angle and length of the vector.

NOTE

Vectors defined by these menus cannot be modified: 2 Points (15.2.6.3), Surface Unit Normal (15.2.6.4), Cross 2 Vectors (15.2.6.6), Normalize Vectors (15.2.6.7), Intersection of 2 Planes (15.2.6.9), Sum or Diff of 2 Vectors (15.2.6.10), and Reverse Surface Normal (15.2.6.13). If vectors defined by these menus are selected for modification, control will be returned to the Select Entry Mode menu.

15.2.6.13 Reverse Surface Normal

With this choice, you can create and display a unit surface normal vector. You can also reverse the direction of all surface normals associated with a specific surface. This is useful because some Numerical Control manufacturing operations require that the normal vectors point away from the material side of a surface. You can create a surface without regard to its normals and then confirm or change them later.

```
SURFACE NORMAL MODE
1.DISPLAY VECTOR
2.REVERSE NORMAL
```

Enter:

1 To define and display a unit surface normal vector on a selected surface.

INDICATE SURFACE	Use the graphics cursor to select a surface. Selecting the surface with the graphics cursor on the surface causes the system to define the unit surface normal vector at the graphics cursor location. Selecting the surface with the graphics cursor off the surface causes the system to define the vector at the surface parameters u,v=(0.5,0.5).

2 To reverse one or all of the normals associated with a surface.

REVERSE NORMAL MODE 1.INDICATE SURFACE 2.REV. LAST VECTOR	Enter: 1 To use the graphics cursor to indicate the surface for which all normals are to be reversed. The system indicates this surface by displaying a unit normal on the surface at u,v=(0.5,0.5). 2 To indicate that the vector to be reversed is the one previously defined with 1.DISPLAY VECTOR under the SURFACE NORMAL MODE prompt. The system deletes the vector and replaces it with a new reversed vector at that location.

15.2.7 Bezier Curve

With this choice, you can define or modify a Bezier curve, which is a parametrically-defined polynomial whose shape approximates a control polygon (refer to figure 3-9). In general, the polygon point does not lie on the curve, except for the first and the last polygon point. The number of polygon points can range between 2 and 15 and is referred to as the order of the curve. The degree of the polynomial curve is one less than the order.

The polygon points can be looked upon as exerting force on the curve and pulling the curve in a certain shape. You can define and modify the curves using the polygon points. If you move one polygon point, the shape of the entire curve is affected. The first two polygon points control the tangent of the curve at the starting point, thus not moving these points results in no change of tangent at the beginning of the curve. The same is true for the end of the curve involving the last two polygon points.

Other methods of defining and modifying this curve are available which do not involve the polygon points, for instance approximating an order point set or deformation. You can, however, always inspect the polygon points graphically and numerically. In the graphic representation, the polygon points are joined by dotted lines (refer to figure 3-9). You can use 14 DATA VERIFY (described in the ICEM Design/Drafting Basic Construction manual) to obtain the numerical value of each polygon point.

The menu described in this section is:

```
BEZIER CURVE
1.DEFINITION
2.MODIFICATION
```

The choices in this menu are described next.

15.2.7.1 Definition

With this choice, you can define a Bezier curve:

```
DEFINITION
1.POLYGON
2.INTERPOLATION/APPROXIMATION
3.BLENDING
4.CONVERSION
5.DUPL&TRUNC/EXTEND
```

The choices in this menu are described next.

15.2.7.1.1 Polygon

With this choice, you can define a Bezier curve by specifying the polygon points.

DEFINITION MODE 1.2-D 2.3-D	Select the two- or three-dimensional mode referring to the definition of a space curve. In two-dimensional mode, zt is the current depth.
CONTROL POINT ENTRY MODE 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT	Enter: 1 To use the graphics cursor to indicate positions. 2 To enter coordinates. They are either in model space or in transform space according to the definition space modal. (You cannot enter zt during two-dimensional mode for transform space.) 3 To select existing points.

You can enter up to 15 positions, coordinates, and/or points. Clear the last entry by entering [. Enter] for another selection of the control point entry mode. Entering an additional] initiates computation of the curve.

15.2.7.1.2 Interpolation/Approximation

With this choice, you can approximate an ordered set of points. Up to 64 points can be selected. You can also specify several conditions or constraints on the curve. These conditions can be classified as follows:

- Conditions at a specified point in the set

  You may impose up to three independent conditions of this kind at a specified point.

  - Constrain the curve to pass through a specified point.
  - Constrain the curve to have a specified tangent at a specified point.
  - Constrain the curve to have a specified radius of curvature or a curvature of zero degrees at a specified point.

  The constraints are independent of each other in that a radius of curvature can be specified without defining a tangent.

  You can also specify end conditions for the curve. They should only be used if the first and the last point of the given point set coincide.

- Global conditions

  Global conditions are applied to the entire curve. For example, the degree of the curve you specify defines the freedom the curve has for approximating the point set. If the degree is low, this freedom is restricted; if the degree is high, the curve follows the given point set more closely. A high degree of freedom may produce undesirable oscillations in the curve.

  Another global condition is the parameter distribution. Each point in the ordered point set is associated with a parameter value between 0 and 1, inclusive. There are three distributions available:

  - The equidistant distribution, where the nth point in the set is assigned the value of (n-1)/(N-1). N is the total number of points.
  - The chordal distribution, where the nth point in the set is assigned the accumulated sum of the distances of points P_1P_2 to $P_{n-1}P_n$ with regard to the total length of the distances.
  - The mean distribution, where the nth point in the set is assigned a value that is the mean of the equidistant and the chordal distribution.

The local conditions and global conditions are not independent of each other. The degree of a curve and the imposed constraints affect each other. If the degree of the curve is k, then initially the order or degrees of freedom the curve has is (k+1). Defining constraints at specified points reduces the degrees of freedom, since the curve must pass through the given point. A tangent constraint removes one degree of freedom, since the curve must have a specified tangent at a given location. A curvature constraint removes two degrees of freedom, since internally two constraints are imposed on the curve, one being a tangent constraint. If you specified a tangent constraint for the same point, imposing a curvature constraint removes only one degree of freedom.

The number of removed degrees of freedom is referred to as the minimal degree. Keying in the minimal degree allows the curve to fulfill all the given constraints. If you specify a degree that is higher than the minimal degree, additional freedom is obtained and is used to approximate the points that have no constraints. The higher the degree, the better the curve approximates the points. The degree, however, cannot be raised indefinitely. It is bounded by two limits, one being the number of variables available in the curve definition (14), the other being the number that is computed by adding together the minimal degree and the number of points in the set that have no constraint attached.

The prompts guide you in the curve definition. First, all points are selected. In addition to the usual control point entry mode selection, a fourth option is available: you can select any existing system curve that is used to define a series of points extending from one end to the other end of the curve. After the point selection is completed, you can specify local constraints. Only the tangent direction is used as a constraint. The correct magnitude is determined by the program, since magnitude depends on the parametrization and is not an intrinsic geometric property.

You will remain in this section until all constraints are defined or may skip this section to immediately proceed with the definition of the global condition. When initially defining a curve, you can only specify the degree of the curve in accordance with the given limits. The parameter distribution will be defaulted to be the mean distribution, since experience has shown that in many examples this distribution renders curves that are more fair than the curves with the other two parameter distributions.

The curve computation is carried out in an iterative way. The approximation scheme used is the least-squares method. In a first iteration, the magnitudes of imposed tangents are computed; in a second iteration, the imposed curvature is taken into account. In the last step, the final curve is computed. If no constraints are given, however, only the last step is performed.

After computation of the curve, you are asked if the information used to compute the curve (the points and the additional constraints) should be retained, so that it is possible to later modify this curve using this information. If you allow the system to store that information, the curve can then be modified using the Constraint Modification menu (refer to 15.2.7.2.2 CONSTRAINTS later in this chapter).

The Constraint Modification menu allows you to select a curve that was created by interpolation and approximation (refer to 15.2.7.2.2 CONSTRAINTS) and to retrieve the information that is used to define the curve. You can inspect the conditions and modify them. In addition to the options available at definition time, there are two more modification types possible:

- Parameter distribution
- Smoothing the polygon

You can select another parameter distribution and recompute the curve. Smoothing the polygon refers to the following process: The curve is computed to minimize certain distances, thereby varying the polygon points freely. This leads to a polygon that is not acceptable (refer to figure 3-9). When you decide to smooth the polygon, a factor is entered that is used as a weight. Additional equations are set up that have the effect of straightening the polygon to a certain extent. This means, however, that the points are not as well approximated as before and you may be prompted to increase the degree of the curve to get acceptable results (refer to figures 3-10 and 3-11).

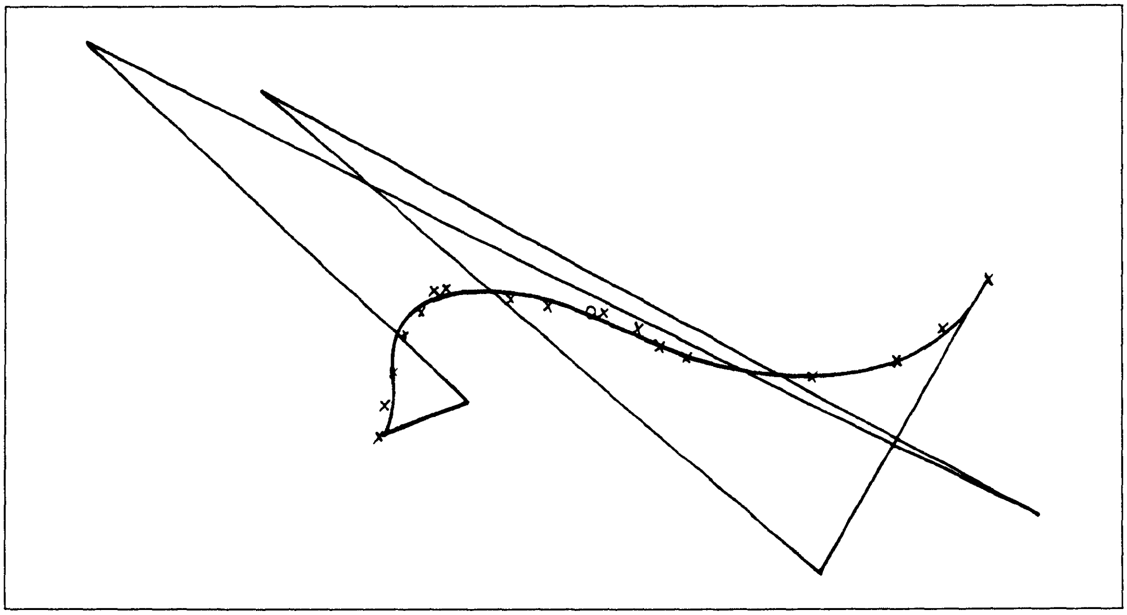


Figure 3-9. Polynomial Curve (Smoothing Factor of 0.0)

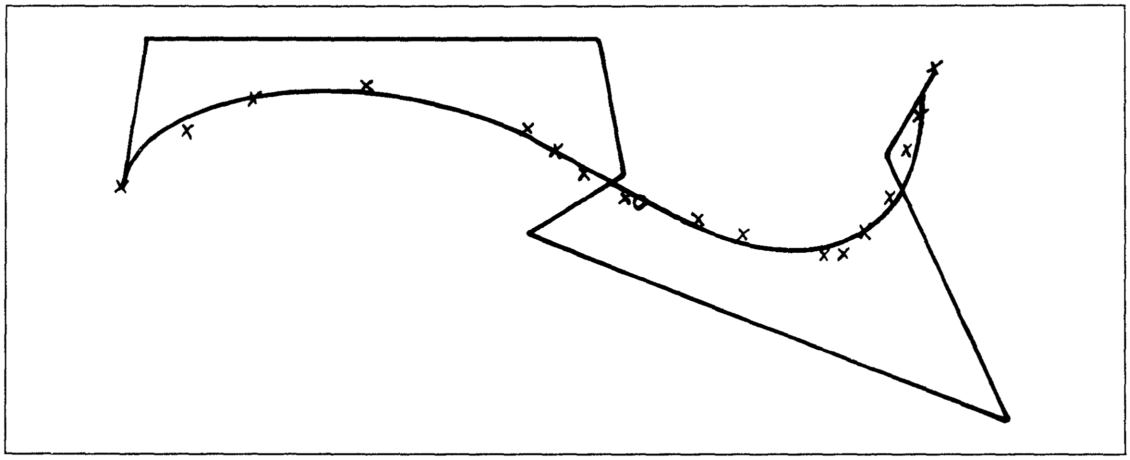


Figure 3-10. Polynomial Curve (Smoothing Factor of 0.1)

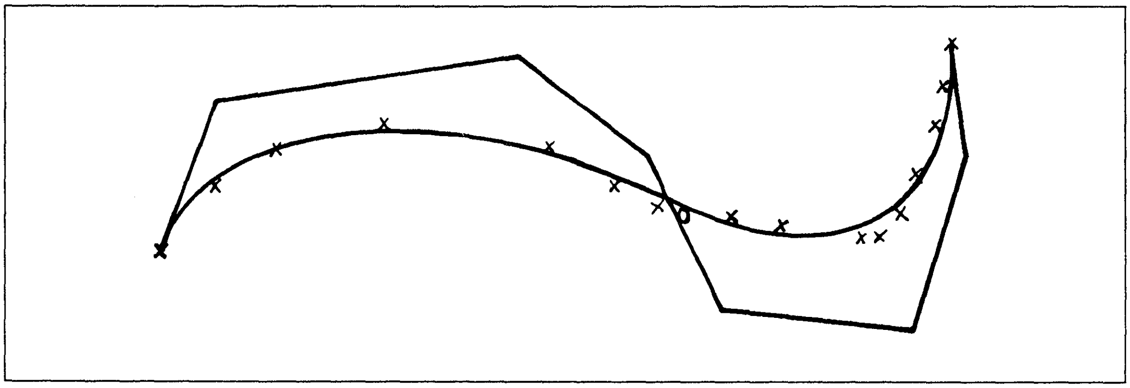


Figure 3-11. Polynomial Curve (Smoothing Factor of 0.2)

The system first prompts you to select either 2-D or 3-D definition mode and then displays:

DEFINITION MODE 1.2-D 2.3-D	Select the two- or three-dimensional mode referring to definition of a planar or space curve. In two-dimensional mode, zt is the current depth.
CONTROL POINT ENTRY MODE 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT 4.EXISTING CURVE	Enter: 1 To use the graphics cursor to indicate position. 2 To enter coordinates. 3 To select existing points. 4 To select an existing curve.

If you select 4.EXISTING CURVE, the system displays:

NUMBER OF POINTS = nn	Enter the number of points to be generated on the curve. The maximum number of points is 64. The first (and last) point coincides with the curve end(s). The points are arranged in ascending order, starting with the endpoint that was closest to the cursor position when the curve was selected.

After all points have been defined, you can proceed to the following section where constraints at specified points can be defined. You remain in this section until all the constraints are defined.

CONSTRAINT DEFINITION 1.CURVE THRU POINT 2.TANGENT AT A POINT 3.CURVATURE AT A POINT 4.END CONDITIONS	Enter: 1 To select a point. The curve is forced through that point. 2 To prescribe a tangent at a point. 3 To prescribe the curvature at a point. 4 To impose end conditions for the curve.

If you select 1, 2, or 3 from the Constraint Definition menu, the system displays:

CONTROL POINT SPECIFICATION 1.SCREEN SELECT 2.SPECIFY POINT NUMBER 3.FIRST POINT 4.LAST POINT	Enter: 1 To identify a point by screen position. The point nearest to the given location is selected. 2 To identify a point by its number. 3 To select the first point. 4 To select the last point.

If you select 2.TANGENT AT A POINT from the Constraint Definition menu, the system displays:

TANGENT DEFINITION MODE 1.KEY-IN 2.SELECT CURVE END	Enter: 1 To enter the components of a vector. The vector is displayed. 2 To select one end of an existing curve. The tangent of the endpoint nearest to the cursor position is computed and displayed as a vector connected to the point in question.
REVERSE DIRECTION OF TANGENT?	Enter: Y To reverse the direction. N To accept the direction.

If you select 3.CURVATURE AT A POINT from the Constraint Definition menu, the system displays:

CURVATURE DEFINITION MODE 1.ENTER RADIUS 2.ZERO CURVATURE 3.SELECT CURVE END	Enter: 1 To enter a radius of the circle of curvature. 2 To impose a curvature of zero. 3 To select one end of an existing curve. The curvature at the endpoint closest to the cursor position is imposed.

If you select 4.END CONDITIONS from the Constraint Definition menu, the system displays:

END CONDITIONS 1.CLOSE CURVE 2.SET TANGENTS EQUAL 3.SET CURVATURE EQUAL	Enter: 1 To make the start and end of the curve meet. 2 To impose tangents at both ends having the same direction. 3 To impose tangents at both ends of the curve having the same direction and curvature.

Terminate the constraint section by entering] after the CONSTRAINT DEFINITION prompt.

You next enter more information concerning the curve. The system displays:

MINIMUM DEGREE = nn MAXIMUM DEGREE = mm	A curve degree is computed that is minimal in that all constraints can be met. If no constraints are imposed, m + 1 is the number of points given. However, mm will not be greater than 14.
1.DEGREE = mm	You can accept the value mm or enter an appropriate value between nn and mm including these two values.
SAVE CONDITION SET?	Enter: Y To save the condition set (points and imposed constraints) in the data base. N To indicate the condition set is not saved for further modification.

The curve is then computed and defined in the data base. Additional modifications using the condition set are possible in 15.2.7.2.2 CONSTRAINTS if the condition set was saved.

The system may display the following error message when you are defining the points and constraints, and during computation of the curve. The system returns you to DEFINITION MODE if two coincidental points are created.

Message	Description
TWO CONSECUTIVE POINTS ARE IDENTICAL	In the given point set, two consecutive points coincide.

While defining the curve, the constraints for a given point can only be imposed once. The following error messages indicate they cannot be removed. Removal of a constraint can be carried out in the Modification menu. The system displays:

Message	Description
POINT CONSTRAINT ALREADY DEFINED	Point already has a point constraint.
TANGENT CONSTRAINT ALREADY DEFINED	Point already has a tangent constraint.
CURVATURE CONSTRAINT ALREADY DEFINED	Point already has a curvature constraint.
END CONDITION ALREADY DEFINED	End conditions are already present.
TANGENT DEGENERATED	The tangent has zero length.
ZERO LENGTH TANGENT	The curvature radius is zero, which is not allowed.

The following error messages are printed when an inconsistency in the number of contraints is detected. In this case, all constraints are deleted and you must start over with constraint definition.

```
TOO MANY POINT CONSTRAINTS

TOO MANY TANGENT CONSTRAINTS

TOO MANY CURVATURE CONSTRAINTS

TOO MANY CONDITIONS FOR MAXIMAL DEGREE
```

Admissible numbers are: up to 15 point constraints, up to 8 tangent constraints, and up to 4 curvature constraints.

The following error message is printed when an inconsistency exists between the number of constraints and the provided degree:

```
DEGREE TOO HIGH - UNDERDETERMINATION
```

You are prompted to enter a lower degree.

15.2.7.1.3 Blending

With this choice, you can connect two arbitrary curves (line, arc, conic, two-dimensional spline, three-dimensional spline, machining curve, point set, composite curve, Bezier curve) by using a Bezier curve meeting certain constraints at one or both ends.

The system prompts you to identify the first curve:

`INDICATE CURVE`	Select the first curve at the end where the Bezier curve should start.

`CONTINUITY MODE`
`1.TANGENT`
`2.TANGENT AND CURVATURE`
	Enter:
	1 To obtain tangent continuity.
	2 To obtain tangent and curvature continuity.

Then the system asks you to identify the second curve:

`INDICATE CURVE`	Select the second curve at the point where the Bezier curve should end.

`CONTINUITY MODE`
`1.TANGENT`
`2.TANGENT AND CURVATURE`
	Enter:
	1 To obtain tangent continuity.
	2 To obtain tangent and curvature continuity.

15.2.7.1.4 Conversion

With this choice, you can approximate curves (lines, arcs, conics, two-dimensional splines, three-dimensional splines, machining curves, point set, composite curve, Bezier curves) by using a Bezier curve. The approximation is an interpolation if the curve is a line.

INDICATE CURVE — Select a curve.

CURVE EXTENT
1.ENTIRE CURVE
2.PART OF CURVE

Enter:

1 To approximate the entire curve.

2 To approximate only part of the curve.

If you entered 2.PART OF CURVE, the system displays:

START PARAMETER
1.SCREEN POSITION
2.KEY-IN
3.EXISTING POINT
4.SELECT CURVE END

Enter:

1 To indicate a position near the desired point at which the new curve starts. If the position does not lie on the curve, the shortest two-dimensional normal is dropped from the position to the curve.

2 To enter the parameter value to define the start point. The system displays the parameter values of the start and endpoints of the curve. You can enter a value between those two values to define the point within the curve, or beyond them to define the point on the extended curve.

3 To select an existing point near the desired point at which the new curve starts. If the point does not lie on the curve, the shortest three-dimensional normal is dropped from the point to the curve.

4 To indicate one end of an existing curve.

Then you are prompted for the END PARAMETER having the same choices.

In all cases, the Bezier curve generated is degree 14 and there is only one curve. If the existing curve is an ellipse and the opening angle is larger than 180^{o}, two curves are generated in order to more closely match the ellipse.

15.2.7.1.5 Dupl&Trunc/Extend

With this choice, you can define a Bezier curve that is identical to part of the Bezier curve (segment) or that extends a Bezier curve (extension).

INDICATE BEZIER CURVE	Select a Bezier curve.
START PARAMETER 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT 4.SELECT CURVE END	Enter: 1 To indicate a position near the desired point at which the new curve starts. If the position does not lie on the curve, the shortest two-dimensional normal is dropped from the position to the curve. 2 To enter the parameter value within the range from -1 to 2 to indicate where the starting point is to be. The parameter values 0 to 1 define points within the existing curve. The parameter values from -1 to 0, or from 1 to 2, define points on the extended curve. 3 To select an existing point near the desired point at which the new curve starts. If the point does not lie on the curve, the shortest three-dimensional normal is dropped from the point to the curve. 4 To indicate one end of an existing curve.

Then you are prompted for the END PARAMETER having the same choices.

15.2.7.2 Modification

With this choice, you can modify a Bezier curve. The menu described in this section is:

```
MODIFICATION
1.POLYGON
2.CONSTRAINTS
3.DEFORMATION
4.DEGREE
5.SEGMENT
6.PARAMETER
```

The choices in this menu are described next.

15.2.7.2.1 Polygon

With this choice, you can modify the polygon of the Bezier curve interactively. Moving one point affects the shape of the entire curve. Modification of the polygon point affects only the xt-, yt-coordinates of that point (the zt-coordinate is kept constant) whenever screen position is used. In all other cases, the zt-coordinate can be changed.

INDICATE BEZIER CURVE	Select the Bezier curve.

After indicating the curve, the polygon of the curve is displayed.

INDICATE POLYGON POINT	Select a polygon point with the graphics cursor.

MODIFICATION CONTROL MODE
1.SCREEN POSITION
2.KEY-IN
3.EXISTING POINT
4.DELTA

Enter:

1 To move the polygon point to the given position. The current zt-coordinate of the point remains fixed.

2 To enter new values. According to the setting of the definition space modal, the coordinates of the polygon point are displayed in model or transform space.

3 To select an existing point that replaces the polygon point.

4 To enter delta values. According to the setting of the definition space modal, the delta values are interpreted as being in model or transform space.

After the modification entry is completed, a new curve is defined and displayed. The MODIFICATION CONTROL MODE prompt appears again only after you enter [. You remain in the same control mode so that you can make modifications using the same polygon point. Enter] two times to return to the INDICATE POLYGON POINT prompt.

The curves created at successive stages of the modification process do not remain in the data base. Only the last curve created is stored. The original curve is blanked during the process.

If you enter [in response to the INDICATE POLYGON POINT prompt, the system displays:

REPLACE OLD ENTITY?

Enter:

Y To replace the original curve with the last curve created.

N To retain the original curve. The last curve created is deleted.

15.2.7.2.2 Constraints

With this choice, you can retrieve the information in the condition set if a Bezier curve was defined using a condition set (points and possibly imposed constraints) and the condition set was saved. The CONSTRAINTS choice also lets you:

- Remove constraints
- Add constraints
- Change the degree
- Change the parametrization
- Smooth the polygon

It is also possible to delete the condition set to get free space in the data base. (Refer to 15.2.7.1.2 INTERPOLATION/APPROXIMATION earlier in this chapter.)

INDICATE BEZIER CURVE	Select the Bezier curve.

After selection of the curve, a complete list of the condition set is presented.

A printout example follows:

```
COMPLETE LIST OF CONDITION SET
NUMBER OF POINTS = 9
DEGREE OF CURVE  = 7
PARAMETER        : MEAN
SMOOTHING FACTOR = 0.0000
CLOSURE          : NO
1.........FIX TAN
3.........    TAN
5.........        CUR
9.........FIX TAN
```

The global conditions are listed first. They describe the number of points used, the degree of the curve, and the parameter distribution. The polygon smoothing factor and the closure condition are listed.

Then each point that carries constraints is listed. For example, the first point is fixed and carries a tangent, and the third point carries a curvature.

To list the tangent and curvature values, select 4.LIST CONSTRAINT when the Constraint Modification menu appears.

```
CONSTRAINT MODIFICATION
1.LIST CONDITION SET
2.IMPOSE CONSTRAINT
3.REMOVE CONSTRAINT
4.LIST CONSTRAINT
5.CHANGE DEGREE
6.CHANGE PARAMETER
7.SMOOTH POLYGON
8.DELETE CONDITION SET
```

Enter:

1 To list the complete condition set.

2 To impose a constraint.

3 To remove a constraint.

4 To list a constraint.

5 To change the degree of the curve.

6 To change the parametrization.

7 To introduce a polygon smoothing factor.

8 To delete the condition set.

] initiates computation of the curve with the actual condition set.

If you select 1 from the Constraint Modification menu, a complete list of the condition set is displayed.

If you select 2 or 3 from the Constraint Modification menu, the system displays:

```
IMPOSE/REMOVE CONSTRAINT
1.CURVE THRU POINT
2.TANGENT AT A POINT
3.CURVATURE AT A POINT
4.END CONDITIONS
```

Enter:

1 To impose/remove a fixed point condition.

2 To impose/remove a given tangent.

3 To impose/remove a curvature condition.

4 To impose/remove a closed curve condition.

Selection of the point follows the same rules as described in 15.2.7.1.2 INTERPOLATION/APPROXIMATION.

If you select 2.TANGENT AT A POINT from the Impose/Remove Constraints menu to be imposed, the system displays:

```
TANGENT DEFINITION MODE
1.KEY-IN
2.SELECT CURVE END
```

Enter:

1 To enter the coordinates of a tangent according to the setting of the definition space modal.

2 To select any curve. The tangent at the end nearest to the cursor location is computed.

The tangent direction is displayed in the current work view and the system displays:

```
REVERSE DIRECTION OF TANGENT?
```

Enter:

Y To reverse the tangent.

N To use the given tangent direction.

If a curvature is to be imposed, the system displays:

CURVATURE DEFINITION MODE 1.ENTER RADIUS 2.ZERO CURVATURE 3.SELECT CURVE END	Enter: 1 To enter the radius of curvature. 2 To impose zero curvature. 3 To select any curve. The curvature at the end nearest to the cursor location is computed.

If a curvature is imposed at a point, the point does not have to be fixed or carry a tangent constraint.

If you select 4.END CONDITIONS from the Impose/Remove Constraint menu, the system prompts you to impose/remove the appropriate conditions:

END CONDITIONS 1.CLOSE CURVE 2.SET TANGENTS EQUAL 3.SET CURVATURE EQUAL	Enter: 1 To impose/remove the conditions that the starting and end points of the curve meet. 2 To impose/remove the condition that the starting and end points of the curve have the same tangent. 3 To impose/remove the condition that the starting and end points of the curve have the same tangent and curvature.

If you select 4.LIST CONSTRAINT from the Constraint Modification menu, the system prompts you to specify the point. The coordinate data of that point in model and transform space is printed. If the point carries a tangent constraint, the tangent direction is listed in model and transform space. If the point has a curvature imposed, the curvature value (if it is zero) or the curvature radius, is listed.

If you select 5.CHANGE DEGREE from the Constraint Modification menu, the system displays:

1.DEGREE = nn	Enter the appropriate degree ($1 \leq n \leq 14$).

If you select 6.CHANGE PARAMETER from the Constraint Modification menu, the system displays:

PARAMETER 1.EQUIDISTANT 2.CHORDAL 3.MEAN	Enter: 1 To use an equidistant parametrization. 2 To use a chordal parametrization. 3 To use the mean value of an equidistant and a chordal parametrization.

If you select 7.SMOOTH POLYGON from the Constraint Modification menu, the system displays:

`1.FACTOR = n.nnnn`	Enter a positive factor. This factor is used as a weight to smooth the polygon of the Bezier curve. The default value is zero.

For a discussion of how the choice of the degree, the selected parametrization, and the factor for smoothing the polygon affects the computation, refer to INTERPOLATION/APPROXIMATION (15.2.7.1.2).

If you select 8.DELETE CONDITION SET from the Constraint Modification menu, the system displays:

`DELETE CONDITION SET?` Enter:

Y To delete the condition set.

N To save the condition set.

After deleting the condition set, you return to the INDICATE BEZIER CURVE prompt.

If you enter], the system recomputes the curve with the actual condition set and displays the new curve. The system then displays:

`CONTINUE WITH SAME CURVE?` Enter:

Y To allow further modification.

N To continue to the next prompt.

You can create several curves in successive modification. Only the last modified curve is accessible. The original curve is blanked during the process.

`REPLACE OLD ENTITY?` Enter:

Y To replace the original entity with the last one created.

N To retain the original curve. The last curve created is deleted.

If the old entity is replaced, the system displays:

`SAVE CONDITION SET?` Enter:

Y To save the condition set.

N To delete the condition set.

15.2.7.2.3 Deformation

With this choice, you can modify the shape of an existing Bezier curve by working directly with the curve. Two operational modes are available for this operation: pulling and lifting.

In both operations, a start location (a location on the curve) is moved to an end location, which must not lie on the curve. By moving the start location to the end location, the entire curve is deformed so that the end location lies on the curve. The end points of the curve will always remain fixed during the deformation process. Optionally, the tangents at one or both end points are unchanged.

In the pulling operation, you specify the start and end locations. When specifying the start location, you can specify a location near the curve, and the system finds the correct start location by dropping a two-dimensional or three-dimensional normal onto the curve.

In the lifting operation, you only specify an end location. The system computes the start location by dropping a two-dimensional or three-dimensional normal from the end location onto the curve.

To obtain the desired result, repeat the process with a series of end locations not too far off the curve.

DEFORMATION MODE 1.2D-PULL 2.2D-LIFT 3.3D-PULL 4.3D-LIFT	Enter the deformation mode you prefer.

The difference between two- and three-dimensional mode is that the position the curve is to pass (end location) through has the same depth as the corresponding position on the curve (start location), that is, a curve planar in the work view remains planar.

TANGENCY CONSTRAINT 1.NO CONSTRAINT 2.ONE END 3.BOTH ENDS	Enter: 1 To free both tangents at the end. 2 To keep the tangent at one end of the curve. 3 To keep the tangents at both ends of the curve.
INDICATE BEZIER CURVE	Select the curve. If a tangency constraint is given at one end, that end is selected which is closest to the cursor position.

DEFORMATION CONTROL MODE 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT	Enter:

1 To select the cursor as the principal entry mode. A two-dimensional normal is dropped.

2 To select data entry as the principal entry mode.

3 To select the identification of points as the principal entry mode. A three-dimensional normal is dropped.

According to the deformation mode selected, the system prompts once lifting or twice (pulling) for each modification. You remain in the same deformation control mode to allow successive deformations. The DEFORMATION CONTROL MODE prompt appears after entering [.

Entering a] completes the deformation process. The curves created at the successive stages do not remain in the data base. Only the last curve created is stored. The original curve is blanked during the process.

REPLACE OLD ENTITY? Enter:

Y To replace the original curve with the last curve created.

N To retain the original entity. The last curve created is deleted.

15.2.7.2.4 Degree

With this choice, you can increase or decrease the degree of the Bezier curve. The system displays:

INDICATE BEZIER CURVE	Select the Bezier curve.
1.DEGREE = nn	The degree of the curve is displayed. You can enter any degree n with $1 \leq n \leq 14$.

Decreasing the degree changes the shape of the curve. The shape of the curve does not change when the degree is increased.

REPLACE OLD ENTITY? Enter:

Y To replace the original curve with the last curve created.

N To retain the original entity. The last curve created is deleted.

15.2.7.2.5 Segment

With this choice, you can split an existing Bezier curve into two or more curves.

INDICATE BEZIER CURVE	Select the curve.
SEGMENT MODE 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT 4.DIVIDE INTO N SEGMENTS	Enter: 1 To use the cursor for the approximate point where the curve is to be segmented. A two-dimensional normal is dropped. 2 To enter the parameter where the curve is to be segmented. 3 To select a point near or on the curve where the curve is to be segmented. A three-dimensional normal is dropped. 4 To allow the curve to be segmented into n partial curves.

If you select 1.SCREEN POSITION from the Segment Mode menu, the system displays:

INDICATE POSITION	Indicate the approximate location where the curve is to be segmented. A two-dimensional normal is dropped. The curve is segmented at this point.

If you select 2.KEY-IN from the Segment Mode menu, the system displays:

1.PARAMETER = n.nnnn	Enter a parameter where the curve is to be segmented. The parameter range is 0 to 1. The default value is 0.5.

If you select 3.EXISTING POINT from the Segment Mode menu, the system displays:

INDICATE POINT	Select a point. If the point does not lie on the curve, a three-dimensional normal is dropped. The curve is segmented at this point.

If you select 4.DIVIDE INTO N SEGMENTS from the Segment Mode menu, the system displays:

NUMBER OF SEGMENTS = nn	Enter the number of segments into which you want to divide the original Bezier curve. N may range from 2 to 256, inclusive. The default value is 2.

15.2.7.2.6 Parameter

With this choice, you can determine the parameter flow of the Bezier curve and, optionally, reverse it.

`INDICATE BEZIER CURVE`	Select the Bezier curve.

The system displays a vector on the curve that points in the tangent direction.

`REVERSE PARAMETER FLOW?`	Enter: Y To reverse the parameter flow; for example, the start of the curve is the endpoint. N To leave the parameter flow unchanged.

Menu 15.3: Surfaces 4

Using Menu 15.3

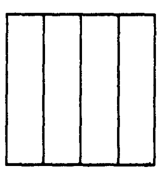
The 15.3 Surfaces menu allows you to create various types of surfaces ranging from planes to more complicated geometric entities such as spheres, cylinders, cones, 3-D tabulated cylinders, and projected surfaces.

The menu described in this chapter is:

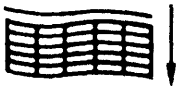
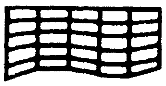
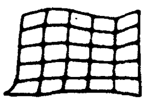
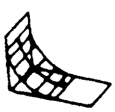
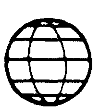
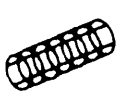
```
SURFACES
 1.PLANE
 2.SURFACE OF REVOLUTION
 3.3-D TABULATED CYLINDER
 4.RULED SURFACE
 5.DEVELOPABLE SURFACE
 6.CURVE MESH SURFACE
 7.FILLET SURFACE
 8.OFFSET SURFACE
 9.SPHERE
10.CYLINDER
11.TORUS
12.CONE
13.COMPOSITE SURFACE
14.SURFACE SEGMENT
15.PROJECTED SURFACES
16.CURVE DRIVEN SURFACE
17.Reserved for Future Use
18.SURFACE VIEWING
```

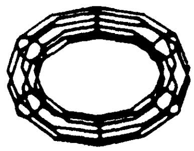
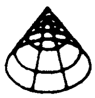
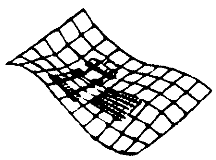
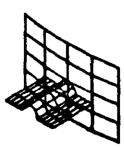
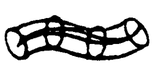
The Surfaces menu offers the following functions. An example of each surface is presented below its name and menu number.

Menu Choice	Function Description
15.3.1 PLANE	Creates a plane of infinite width and length.



Menu Choice	Function Description
15.3.2 SURFACE OF REVOLUTION	Creates a surface by rotating a curve around a line.

Menu Choice	Function Description
15.3.3 3-D TABULATED CYLINDER 	Creates a surface by sweeping a curve through space in the direction indicated by a line or vector.
15.3.4 RULED SURFACE 	Creates a surface by sweeping straight lines between the ends of two curves.
15.3.5 DEVELOPABLE SURFACE 	Creates a ruled surface that can be laid flat.
15.3.6 CURVE MESH SURFACE 	Creates a surface through a framework of intersecting fixed and variable curves.
15.3.7 FILLET SURFACE 	Creates a surface of constant radius between two surfaces along their intersection.
15.3.8 OFFSET SURFACE 	Creates a surface that is a normal offset from a selected surface.
15.3.9 SPHERE 	Creates a sphere in the form of points located at a specified distance from a center point.
15.3.10 CYLINDER 	Creates a right circular cylinder in the form of points located at a specified distance from a center axis line.

Menu Choice	Function Description
15.3.11 TORUS 	Creates a doughnut-shaped torus or toroid in the form of a circle revolved around a line.
15.3.12 CONE 	Creates a cone by revolving a line about an axis.
15.3.13 COMPOSITE SURFACE 	Creates a surface composed of any set of surfaces connected along their common edges.
15.3.14 SURFACE SEGMENT 	Creates a surface that is a segment of an existing surface.
15.3.15 PROJECTED SURFACES 	Creates several ruled surfaces simultaneously by projecting contiguous curves to a depth or onto a surface.
15.3.16 CURVE DRIVEN SURFACE 	Creates a surface by driving a boundary curve along a center line curve.
15.3.17 Reserved for Future Use	
15.3.18 SURFACE VIEWING	Creates surface faceting and allows modification of surface paths.

The remainder of this chapter describes the choices in menu 15.3.

15.3.1 Plane

With this choice, you can create planes that are infinite in length and width. In a view looking straight down onto the plane, the system displays each plane as a rectangular grid. This grid is for display purposes only.

The following menu is described in this section:

```
PLANE
1.COEFFICIENTS
2.THRU 3 NONCOLLINEAR PTS
3.THRU PT AND PARALLEL TO PLANE
4.PARALLEL TO PLANE AT A DISTANCE
5.THRU PT AND PERPTO VECTOR
6.THRU 2 PTS AND PERPTO PLANE
7.THRU PT AND PERPTO 2 PLANES
8.BETWEEN 2 LINES
```

In each plane operation, the system first requests specific parameters and then requests the center and a corner of the new plane. The requests for specific parameters are individually described for each operation. The requests for the center and a corner are:

INDICATE CENTER	Use the graphics cursor to indicate the location of the center of the plane.
INDICATE CORNER	Use the graphics cursor to indicate the location of one of the corners of the plane.

A plane is defined as the set of all points (x,y,z) that satisfy the equation Ax + By + Cz + D = 0. The coefficients (A,B,C) are the coordinates of a unit vector that is perpendicular to the plane. The absolute value of the D coefficient is the shortest (perpendicular) distance from the origin to the plane. D is greater than zero if the origin is on the side of the plane that the unit normal vector (A,B,C) points toward. D is negative if the unit normal points away from the origin. D is equal to zero if the plane passes through the origin.

In equation form, this is

$$D = -(A,B,C) \cdot (Px,Py,Pz)$$

where (A,B,C) is the plane's unit normal, (Px,Py,Pz) is a vector from the origin to a point on the plane, and . is the vector dot product.

For example, a plane parallel to the yz-plane and passing through the point P (Px,Py,Pz) = (1,2,3) is perpendicular to the x-axis. The unit normal is (A,B,C) = (1,0,0) and the D coefficient is:

$$D = -(1,0,0) \cdot (1,2,3)$$

$$D = -(1*1 + 0*2 + 0*3) = -1$$

Thus, the equation for this plane is:

$$Ax + By + Cz + D = 1x + 0y + 0z + (-1) = 0$$

The choices in the Plane menu are described next.

15.3.1.1 Coefficients

With this choice, you can create a plane using the coefficients for the equation of a plane Ax + By + Cz + D = 0. The coefficents are those described previously. You can define the plane using coefficients either in the current work view or model space. Refer to menu 1.15.3 in the ICEM Design/Drafting Introduction and System Controls manual to set the display to transform or model coordinates.

```
DEFINITION SPACE
1.TRANSFORM SPACE
2.MODEL SPACE
```

Enter:

1 To define the plane using transform coordinates.

1.COEFFICIENT AT = n.nnnn 2.COEFFICIENT BT = n.nnnn 3.COEFFICIENT CT = n.nnnn 4.COEFFICIENT DT = n.nnnn	Enter or accept the coefficients relative to the current work view to define the plane using the equation ATx + BTy + CTz + D = 0. By entering [, you can back up to the DEFINITION SPACE prompt, regardless of the modal setting, and temporarily change to model space without affecting the modal setting.

2 To define the plane using model coordinates.

1.COEFFICIENT A = n.nnnn 2.COEFFICIENT B = n.nnnn 3.COEFFICIENT C = n.nnnn 4.COEFFICIENT D = n.nnnn	Enter or accept the coefficients relative to model space to define the plane using the equation Ax + By + Cz + D = 0. By entering [, you can back up to the DEFINITION SPACE prompt, regardless of the modal setting, and temporarily change to transform space without affecting the modal setting.

The system then prompts for the center and a corner of the plane. Refer to menu 15.3.1 earlier in this chapter for a description of these prompts.

15.3.1.2 Thru 3 Noncollinear Pts

With this choice, you can create a plane that passes through three points.

INDICATE POINT	Use the graphics cursor to select three existing points that do not form a straight line.

The system then prompts for the center and a corner of the plane. Refer to menu 15.3.1 for a description of these prompts.

If these points are collinear, they do not define a plane. The system displays:

```
THESE POINTS DO NOT DEFINE A PLANE
```

15.3.1.3 Thru Pt and Parallel to Plane

With this choice, you can create a plane that passes through an existing point and is parallel to an existing plane.

INDICATE POINT	Use the graphics cursor to select a point.
INDICATE PLANE	Use the graphics cursor to select the parallel plane.

The system then asks for the center and a corner of the plane. Refer to menu 15.3.1 for a description of these prompts.

15.3.1.4 Parallel to Plane at a Distance

With this choice, you can create a plane that is parallel to, and a specified distance from, an existing plane.

INDICATE PLANE	Use the graphics cursor to select the parallel plane.
DISTANCE = n.nnnn	Enter the distance between the existing plane and the new plane. A positive distance defines the plane on one side of the existing plane; a negative distance defines it on the opposite side.

The system then prompts for the center and a corner of the plane. Refer to menu 15.3.1 for a description of these prompts.

15.3.1.5 Thru Pt and Perpto Vector

With this choice, you can create a plane that passes through an existing point and is perpendicular to an existing vector.

INDICATE POINT	Use the graphics cursor to select a point.
INDICATE VECTOR	Use the graphics cursor to select the perpendicular vector.

The system then prompts for the center and a corner of the plane. Refer to menu 15.3.1 for a description of these prompts.

15.3.1.6 Thru 2 Pts and Perpto Plane

With this choice, you can create a plane that passes through two points and is perpendicular to an existing plane.

INDICATE POINT	Use the graphics cursor to select two points that do not lie on the same normal to the existing plane.
INDICATE PLANE	Use the graphics cursor to select the perpendicular plane.

The system then prompts for the center and a corner of the plane. Refer to menu 15.3.1 for a description of these prompts.

15.3.1.7 Thru Pt and Perpto 2 Planes

With this choice, you can create a plane that passes through an existing point and is perpendicular to two planes.

INDICATE PLANE	Use the graphics cursor to select the two planes.
INDICATE ENTITY	Use the graphics cursor to select a point.

The system then prompts for the center and a corner of the plane. Refer to menu 15.3.1 for a description of these prompts.

15.3.1.8 Between 2 Lines

With this choice, you can create a plane that contains two existing lines. There are two ways to describe the location and the orientation of the display of the plane: by selecting the first line or by indicating a center and a corner.

BETWEEN 2 LINES DISPLAY METHOD 1.ALONG FIRST LINE 2.CENTER AND CORNER	Enter: 1 To create a plane that is displayed with one edge of its rectangular grid along the first selected line. 2 To create a plane that is displayed at the position and orientation defined by the indicated center and corner positions.

If you select 1.ALONG FIRST LINE from the Between 2 Lines Display Method menu, the plane is displayed as a square. The square's side length is equal to the magnitude of and is positioned along the first selected line. The second line is required to complete the definition of the plane. The system prompts:

INDICATE LINE	Select the line that will become an edge of the plane's rectangular grid. The two lines used to define the plane can be either parallel or intersect at one point.
INDICATE LINE	Select a second line that will complete the definition of the plane.
INDICATE SIDE	Place the graphics cursor in the current work view on the side of the first line. The plane will be displayed on the indicated side of the first line.

See figure 4-1 for an example of creating a plane with a normal vector.

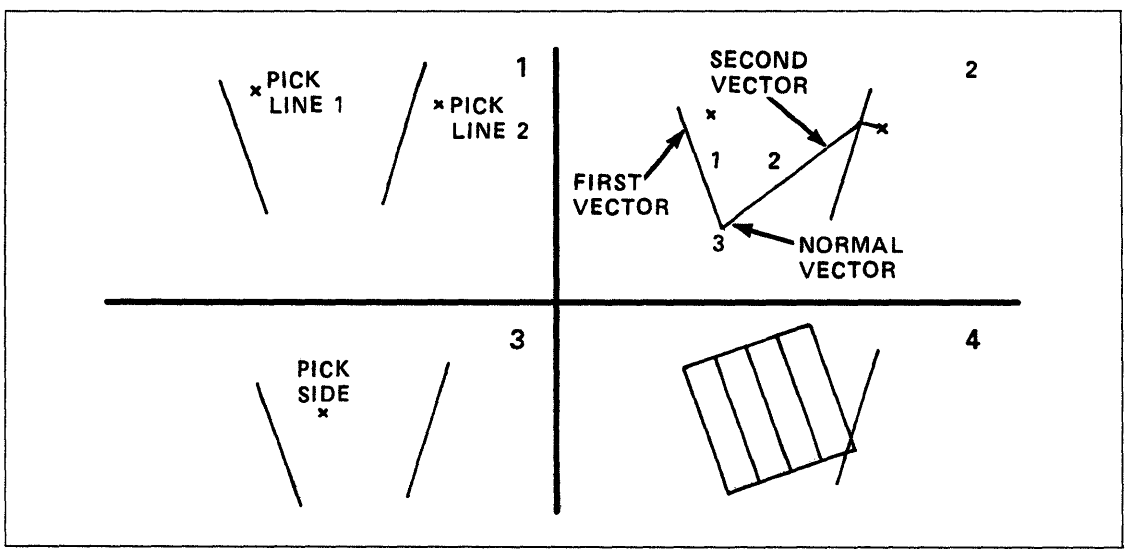


Figure 4-1. Creating a Plane With a Normal Vector

NOTE

You may want to control the sense (location of the arrowhead) of the plane's normal vector. To use this advanced feature, you must define the normal vector of the plane as the vector cross product of the first and second vectors. To define the first vector, use the graphics cursor to select the head end of the vector that runs along the first line. To define the second vector, select a position on the second line for the head end of a vector from the tail of the first line's vector. The position on the second line will be the 2-D normal point from the second pick position.

If the first line selected is of infinite length, the system displays:

```
CANNOT SELECT AN INFINITE LINE
```

If either line has a length of zero, the system displays:

```
LINES CANNOT HAVE ZERO LENGTH
```

If the lines are collinear or they are nonintersecting and nonparallel, the system displays:

```
THESE LINES DO NOT DEFINE A PLANE
```

If you select 2.CENTER AND CORNER from the Between 2 Lines Display Method menu, the system displays:

`INDICATE LINE`	Select the first line to define a plane. The two lines used to define the plane can be either parallel or have, at most, one point in common.
`INDICATE LINE`	Select a second line that will complete the definition of the plane.
`INDICATE CENTER`	Indicate the location of the center of the plane.
`INDICATE CORNER`	Indicate the location of one corner of the plane.

If the selected lines are both of zero length or are collinear or are nonintersecting and nonparallel, the system displays:

```
THESE LINES DO NOT DEFINE A PLANE
```

15.3.2 Surface of Revolution

With this choice, you can define a surface by rotating a curve about a line (figure 4-2).

INDICATE CURVE	Use the graphics cursor to select the generatrix of the surface, that is, the curve to be rotated about a line to define the surface. This curve can be any single-entity curve or a composite curve.
INDICATE LINE AND DIRECTION	Use the graphics cursor to select the line to be used as the axis of rotation. Place the graphics cursor near one end of the line thereby specifying the direction of angular rotation. Rotation is counterclockwise when looking along the line from the indicated end to the opposite end. For example, if the line is horizontal and you indicate the right end, rotation is out of the screen with 0° at the top of the screen and 180° at the bottom.
ROTATION ANGLES 1.START ANG.= nn.nnnn 2.END ANGLE = nn.nnnn	Enter (in degrees) the starting and ending angles through which the generatrix is to rotate around the line. A rotation angle of zero degrees begins along the generatrix. Start and end angles of 0° and 360°, respectively, produce a closed surface. Default angles are 60° and 120°, which produce an open surface. The angles may be positive or negative.

Figure 4-2 illustrates a surface of revolution.

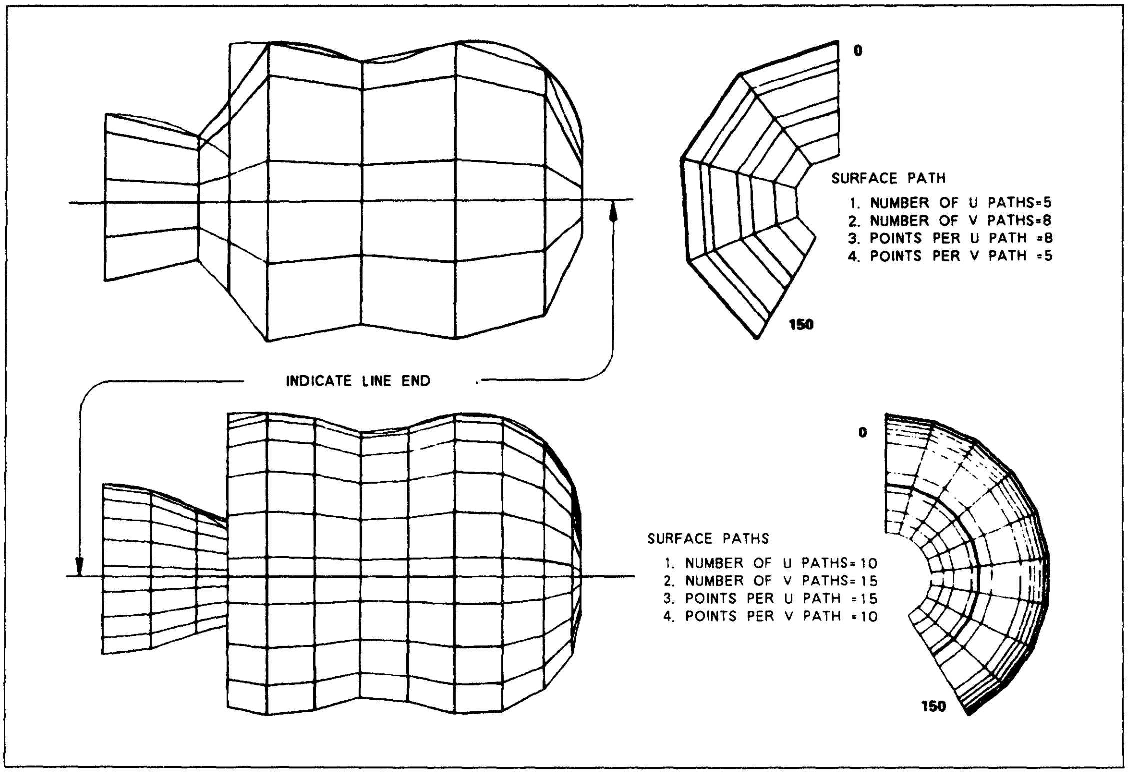


Figure 4-2. Surface of Revolution

15.3.3 3-D Tabulated Cylinder

With this choice, you can define a surface by sweeping a curve through space in the direction indicated by a line or vector (figure 4-3).

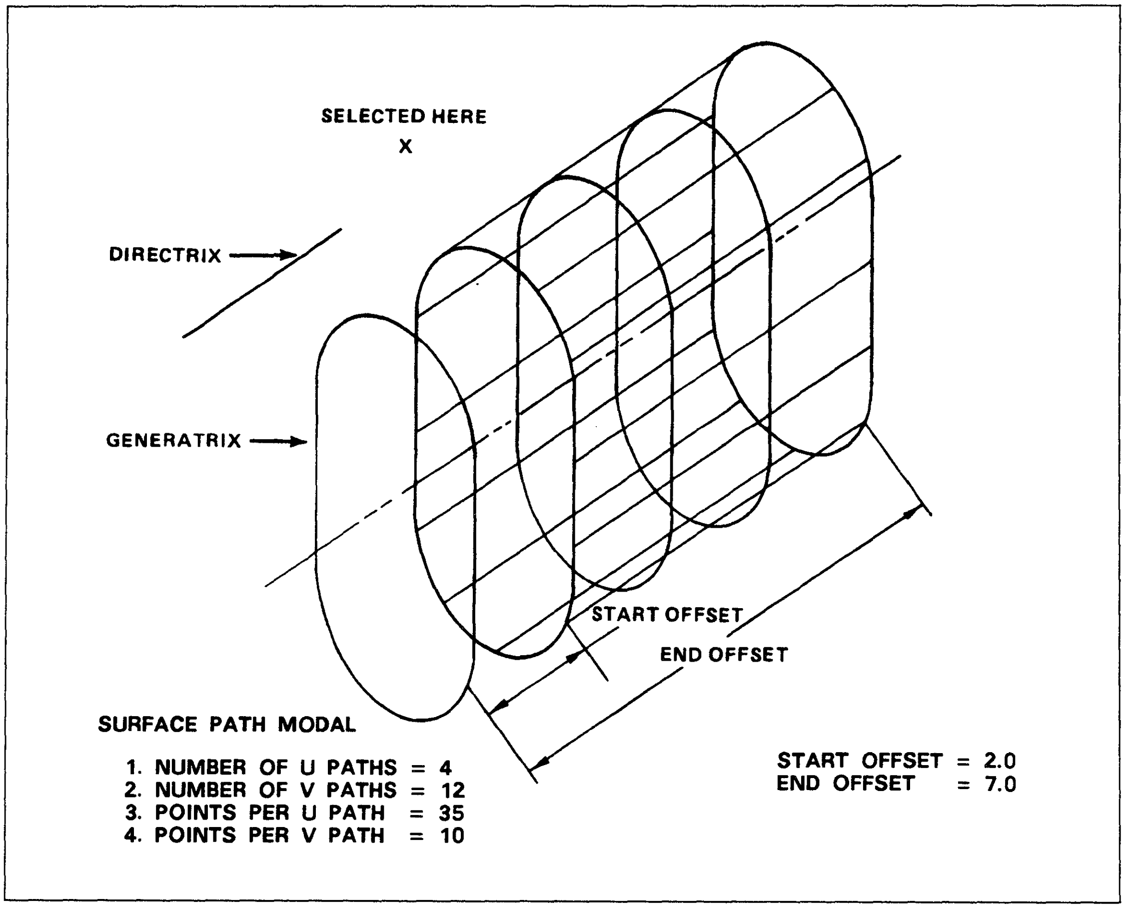


Figure 4-3. 3-D Tabulated Cylinder

INDICATE CURVE	Use the graphics cursor to select the generatrix of the surface, that is, the curve that is swept through space to define the surface. This curve can be any single-entity curve or a composite curve.

Select the method to enter the direction to sweep the generatrix.

```
SWEEP DIRECTION
1.EXISTING LINE
2.EXISTING VECTOR
3.ENTER VECTOR
```

Enter:

1 To select an existing line.

INDICATE LINE END	Use the graphics cursor to select the directrix of the surface. The surface sweeps in the direction defined by the directrix. Place the graphics cursor near the end of the line toward which to sweep the surface. The surface is independent of the length and position of the directrix.

2 To select an existing vector.

INDICATE VECTOR	Use the graphics cursor to select a vector to be used as the directrix of the surface. The surface sweeps in the direction defined by the vector from tail to head. The surface is independent of the length and position of the vector.

3 To enter the transform or model coordinates. Refer to 1.15.2 DEFINITION SPACE MODAL in the ICEM Design/Drafting Introduction and System Controls manual to set the display to either transform or model coordinates. The system displays:

```
DEFINITION SPACE
1.TRANSFORM COORDINATES
2.MODEL COORDINATES
```

Choose the coordinate system in which to specify the vector direction.

Enter:

1 To enter the direction to sweep the generatrix using the transform coordinate system.

VECTOR COMPONENTS 1.IT = n.nnnn 2.JT = n.nnnn 3.KT = n.nnnn	Enter the desired it, jt, and kt vector values.

2 To enter the direction to sweep the generatrix using the model coordinate system.

VECTOR COMPONENTS 1.I = n.nnnn 2.J = n.nnnn 3.K = n.nnnn	Enter the desired i, j, and k vector values.

3 After selecting the direction, define the offset amounts to sweep the generatrix.

SWEEP OFFSETS 1.STARTING OFFSET = n.nnnn 2.ENDING OFFSET = n.nnnn	Enter the starting and ending offsets from the generatrix to specify the position and the range of the surface. Positive values are in the direction given by the directrix from the generatrix; negative values are in the opposite direction as the directrix. For example, if the start offset is negative and the end offset is positive, the surface extends on both sides of the generatrix.

15.3.4 Ruled Surface

With this choice, you can create a surface by sweeping straight lines from the ends of one curve to the corresponding ends of another curve.

INDICATE CURVE	Use the graphics cursor to select the first curve (place the graphics cursor near one end of the curve). This curve can be any single-entity curve or a composite curve.
INDICATE CURVE	Use the graphics cursor to select the second curve (the selection must be placed near the end that corresponds to the indicated end of the first curve).

15.3.5 Developable Surface

With this choice, you can create a developable surface.

Developable surfaces are the subset of ruled surfaces (refer to menu 15.3.4) that can be laid flat. The system finds the edge lines as follows:

- The tangent vector at the start of the first curve is determined. This vector is compared with the tangent vector at the start of the second curve.
- If the vectors are coplanar, the edge line is between the points of tangency.
- If the the vectors are not coplanar, a new point is tried along the second curve.
- In all cases, the entire first curve is used; however, portions of the second curve cannot be used.

After the edge lines have been found, another check is made. This check ensures the surface can be laid out flat with no bumps.

`INDICATE CURVE`	Use the graphics cursor to select the first curve (place the graphics cursor near one end of the curve).
`INDICATE CURVE`	Use the graphics cursor to select the second curve (the graphics cursor must be placed near the end that corresponds to the indicated end of the first curve).
`TOLERANCE (DEGREES) = n.nnnn`	Enter (in degrees) the tolerance acceptable in determining the coplanar condition for the edge lines. A smaller tolerance generates a truer developable surface.

If the surface cannot be defined, the system displays:

```
A DEVELOPABLE SURFACE CANNOT BE DEFINED
BETWEEN THE CHOSEN CURVES
```

15.3.6 Curve Mesh Surface

With this choice, you can create a surface on a framework of two intersecting groups of curves, called fixed curves and variable curves. All fixed curves must intersect all variable curves, and all variable curves must intersect all fixed curves. Three types of surfaces are available with differing mathematical bases for the interpolation.

```
CURVE MESH SURFACE FORM
1.WITH TWIST VECTORS
2.LINEARLY BLENDED
3.LINEARLY BLENDED FROM POINT MESH
```

Enter:

1 To use twist vectors and parabolic approximation to define the curve mesh surface. Continue to INDICATE CURVES.

2 To correspond to the commonly used Coons' interpolation method to define the curve mesh surface. This technique is recommended in most circumstances, since it is less susceptible to ripples in the surface. Continue to INDICATE CURVES.

3 To build the surface from a given set of points using linear blending as in 2.LINEARLY BLENDED. Continue to POINT MESH.

Indicate Curves

```
FIXED CURVES
INDICATE CURVES
```

Use the graphics cursor to select the fixed curves in an ordered manner. Fixed curves can be lines, arcs, conics, two-dimensional splines, composite curves, point sets, three-dimensional splines, or machining curves. Fixed curves cannot be selected end-to-end. Curves that lie end-to-end should be redefined as a composite curve before selection as a fixed curve. You must select at least 2, but not more than 28, fixed curves.

```
VARIABLE CURVES
INDICATE CURVES
```

Use the graphics cursor to select the variable curves in an ordered manner. Variable curves, like fixed curves, can be of any type. Variable curves that lie end-to-end should be redefined as a composite curve before selection as a variable curve. You must select at least 2, but not more than 30, variable curves.

Point Mesh

1.POINTS PER U PATH = nnn
2.POINTS PER V PATH = nnn

The screen must display a lattice of m times n (m * n) points, which may be at different depths. These points will serve to generate a framework of two intersecting groups of three-dimensional splines and then a linearly blended curve mesh surface. You decide which set of curves is to be the U paths and which the V paths and, accordingly, type m or n for each of these sets of paths (for example, if there are m U paths, there must be n V paths). The result is:

1.PTS/U-PATH = m
2.PTS/V-PATH = n

Valid entries for m and n range from 2 to 30. The maximum number of points allowed is 509 ($m * n \leq 509$).

After the number of U and V paths are selected, enter] to display:

U-PATH NO.1
INDICATE POINT

Select the first point of the first U path, then proceed in order until all points in the first U path have been selected.

The system then automatically displays:

U-PATH NO.2
INDICATE POINT

Select the first point of the second U path, then continue in the order in which the points for the first U path were selected.

This procedure is repeated until all U paths have been selected. The system then automatically displays the mesh of three-dimensional splines and, shortly thereafter, the linearly blended curve mesh surface through the given points.

15.3.7 Fillet Surface

With this choice, you can create a fillet surface.

A fillet surface is a constant radius, transitional surface created between two surfaces along their intersection. This surface is generated between the two points of contact of a sphere rolling along the groove formed by two surfaces.

After you answer the prompts in this menu, the system defines the fillet surface using a three-phase process. First, it generates a surface intersection curve using the tolerance and step size as in 15.2.3 SURFACE-SURFACE INTERSECTION CURVE. This results in a series of points.

Second, the system generates three machining curves using each point in the surface intersection curve to locate a hypothetical sphere with a specified radius. The points that make up these machining curves correspond to the center of the sphere, the tangent of the sphere with the drive surface, and the tangent of the sphere with the check surface.

Third, the system defines the fillet surface using the three machining curves. Defining the fillet surface with machining curves allows you to generate a toolpath using the absolute tool motion feature of numerical control. This menu provides various options for displaying the machining curves. (Refer to Using Menu 15.2 in chapter 3 for a description of the machining curve.)

`INDICATE SURFACE`	Use the graphics cursor to indicate the drive surface (usually the smaller surface).
`INDICATE SURFACE`	Use the graphics cursor to indicate the check surface.

The system displays the indicated machining curves after generating the fillet surface and after you repaint the display.

`DISPLAY DEFINITION CURVES`
`1.ALL 3`
`2.CENTER`
`3.EDGES`
`4.NONE`
	Enter:
	1 To display all three machining curves.
	2 To display only the center machining curve.
	3 To display the machining curves on the drive surface and on the check surface.
	4 To display none of the machining curves.

The intersection of the drive surface and the check surface creates four quadrants in which the fillet surface can be defined.

```
QUADRANT MODE
1.NORMALS
2.POINT
3.POSITION
```

Enter:

1 To define the fillet surface in the quadrant toward which both surface normals point. Use 15.2.6.13 REVERSE SURFACE NORMAL to reverse or verify the direction of the normals.

2 To use the quandrant determined by dropping a normal point onto each surface from an existing point. These three points determine the quadrant.

INDICATE POINT	Use the graphics cursor to select an existing point.

3 To use the quadrant determined by dropping a normal point onto each surface from a graphics cursor location and the current depth.

INDICATE POSITION	Use the graphics cursor to select an existing quadrant.

If the system cannot find one or both surface normal points, it displays:

```
CANNOT FIND QUADRANT
```

The system redisplays the QUADRANT MODE prompt for the selection of another mode, point, or position.

Because the definition of a fillet surface is based on the intersection of the drive surface and the check surface, the intersection must be well defined. If one of the surfaces intersects the other along an edge (such as a surface projected to a surface), the system might not define the fillet surface correctly. To prevent this, you can either redefine one surface so that it pierces the other surface or enter Y to the following prompt:

FOLLOW EXISTING SURFACE INTERSECTION CURVE?	Enter: Y To define the three machining curves using the points in the surface intersection curve. N To define the three machining curves using a tolerance and step size.

If you enter Y, the system displays:

`INDICATE MACHINING CURVE`	Use the graphics cursor to select a machining curve for the fillet surface to follow. The tolerance of this machining curve is used as the tolerance of the fillet surface.
`RADIUS = n.nnnn`	Enter the fillet surface radius.

If you enter N, the system displays:

```
1.RADIUS    = n.nnnn
2.TOLERANCE = n.nnnn
3.STEP SIZE = n.nnnn
```

1 Enter the radius of the fillet surface.

2 Enter the tolerance, which is the normal distance between the drive surface and the midpoint of a line segment joining two consecutive points along the surface intersection. A positive tolerance specifies that the step size is the maximum allowable step size. The minimum allowable positive tolerance is 0.005 mm (0.0002 in). A negative tolerance specifies that the tolerance is not used to adjust the step size, but that instead, the step size is a constant.

3 Enter the step size, which is the approximate distance between two consecutive points. The minimum allowable step size is 0.254 mm (0.01 in). If you use very small tolerances, a significant amount of computer time and space is necessary to generate the curves. You should use a tolerance only when necessary and use a reasonably large step size to conserve both computer time and space.

After the surface intersection curve is defined, select the mode in which to start the search for the initial point of the fillet surface.

```
START MODE
1.EDGE INTERSECTION
2.EXISTING POINT
3.POSITION/DEPTH
4.POSITION/PIERCE
```

Enter:

1 To use one or all edge intersections of the two surfaces. Continue to EDGE INTERSECTION.

2 To find the initial intersection point by dropping a normal point onto each surface from an existing point. Continue to EXISTING POINT.

3 To find the initial intersection point by dropping a normal point onto each surface from a specified location and the current depth. Continue to POSITION/DEPTH.

4 To find the initial intersection point by piercing the drive surface with a vector. Continue to POSITION/PIERCE.

The choices in this menu are described next.

Edge Intersection

The system tests all eight edges of the two surfaces and locates up to 50 points on the edge of one surface where it intersects the other surface. If the system finds only one unique point, it defines and displays the fillet surface starting at that point. If more than one fillet surface is possible, the system displays:

MULTIPLE EDGE INTERSECTIONS 1.USE ALL 2.INDICATE POSITION	Enter: 1 To define and display all unique fillet surfaces starting at the intersection points. 2 To use the graphics cursor to indicate a screen position. The system uses the edge intersection nearest this position to define a fillet surface.

The system searches the edges of a surface by dividing the edge into a number of line segments and testing each segment for an intersection with the other surface. Initially, the number of segments used for the search is the same as the number of line segments in the surface display (refer to 15.3.18.2 MODIFY SURFACE PATHS in this manual). For a very large or long surface, this may be too coarse a search. You have two means by which this search may be controlled. First, if you suspect that an edge intersection may be difficult to find, you can use 15.3.18.2 MODIFY SURFACE PATHS to increase the number of segments in the display. (In general, if a short segment displayed on the edge of a surface appears to intersect the other surface, this point is found.) Second, if the system cannot find an edge intersection, you can tell the system to use twice as many segments as it is currently using. This may be repeated if necessary.

If the system does not find an edge intersection, it displays:

NO EDGE INT FOUND RATIO = n.nnnn DOUBLE SEARCH?	Enter: Y To tell the system to try again using twice the current number of segments in its search (2 times n). N To return to the START MODE prompt to select another method.

Existing Point

INDICATE POINT	Use the graphics cursor to select an existing point.

The system drops normal points onto the drive surface and the check surface from the selected point. It then defines the initial point as close as possible to the two normal points. If the system cannot find an initial intersection point, it displays:

```
CANNOT FIND START PT
```

The system redisplays the START MODE prompt for the selection of another start option.

Position/Depth

INDICATE POSITION	Use the graphics cursor to indicate a location.

The system drops normal points onto the drive surface and the check surface from the specified location and current depth. It then defines the initial intersection point as close as possible to the two normal points. If the system cannot find an initial intersection point, it displays:

```
CANNOT FIND START PT
```

The system redisplays the START MODE prompt for the selection of another start option or another position.

Position/Pierce

INDICATE POSITION	Use the graphics cursor to indicate a location over the drive surface near a possible intersection point.

The system pierces the drive surface with a vector perpendicular to the screen and through the specified location. It then defines an initial intersection point as close as possible to the pierce point. If the system cannot find an initial point, it displays:

```
CANNOT FIND START PT
```

The system redisplays the START MODE prompt for the selection of another start option or another position.

After locating the initial intersection point, the system defines the fillet surface.

If the points of tangency between the sphere and the drive surface and between the sphere and the check surface are well defined points within the surface boundaries, the system creates the fillet surface without any adjustment.

If the points of tangency between the sphere and one surface or between the sphere and both surfaces are outside the surface boundaries (that is, the sphere center extends beyond one or both surfaces), the system creates the fillet surface by adjusting the position of the sphere center, thereby modifying the fillet to fit the surfaces. This means that the fillet surface radius is constant, but that the surface can extend between two surfaces, between a surface and an edge, or between two edges (refer to figure 4-4).

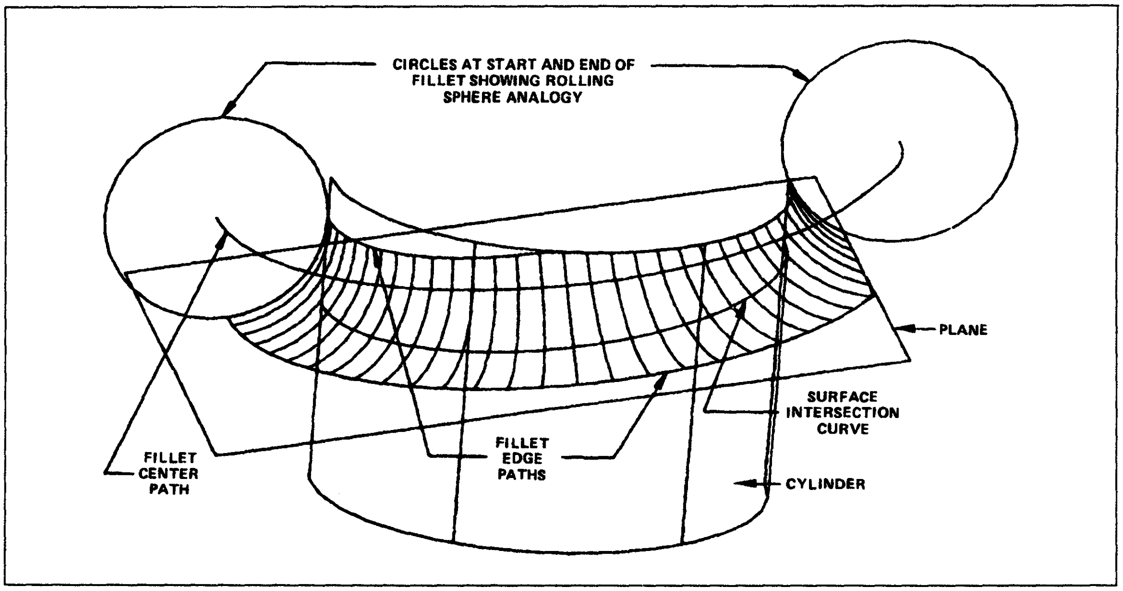


Figure 4-4. Fillet Surface

15.3.8 Offset Surface

With this choice, you can create a surface that is a normal offset from a selected surface (base surface). The offset surface is on the side of the positive unit normal vector to the base surface. You can change the normal direction in 15.1.6.13 REVERSE SURFACE NORMAL. If the system creates the offset surface on the wrong side of the base surface, delete the offset surface and recreate it with a negative offset distance.

`INDICATE SURFACE`	Use the graphics cursor to indicate the base surface.
`OFFSET DISTANCE = n.nnnn`	Enter the normal distance between the base surface and the offset surface.

15.3.9 Sphere

With this choice, you can create a sphere. This surface is the set of points located at a specified distance from a center point. 15.1.3 SURFACE PATHS is used to construct the sphere with the desired number of surface paths.

```
CENTER POINT
1.SCREEN POSITION
2.KEY-IN
3.EXISTING POINT
```

Indicate the center point of the sphere.

Enter:

1 To use the graphics cursor to indicate a position.

2 To enter the xt-, yt-, and zt-coordinates.

3 To use the graphics cursor to select an existing point.

After you select a center point for the sphere, the system displays:

```
FULL EQUATOR?
```

Enter:

Y To construct the sphere with a full 180° equator. The equator rotates about the vertical axis in the view of definition and is measured from the right to the left.

N To construct the sphere with other than a 180° equator.

If you enter N, the system displays:

```
1.STARTING ANGLE = n.nnnn
2.ENDING ANGLE   = n.nnnn
```

Enter the starting angle and the ending angle of the equator. Default angles are 0° and 180°.

After you define the equator, the system displays:

```
FULL CIRCUMFERENCE?
```

Enter:

Y To construct the sphere with a full 360° circumference. The circumference rotates about the horizontal axis in the view of definition and is measured from the top out of the screen.

N To construct the sphere with other than a 360° circumference.

If you enter N, the system displays:

```
1.STARTING ANGLE = n.nnnn
2.ENDING ANGLE   = n.nnnn
```

Enter the starting angle and the ending angle of the circumference. Default angles are 0° and 360°.

After you define the circumference, the system displays:

```
1.RADIUS = n.nnnn
```

Enter the sphere radius. Default is 6.35 mm (0.25 in).

The sphere in figure 4-5 has an equator of 30° to 170° and a circumference of 0° to 270°.

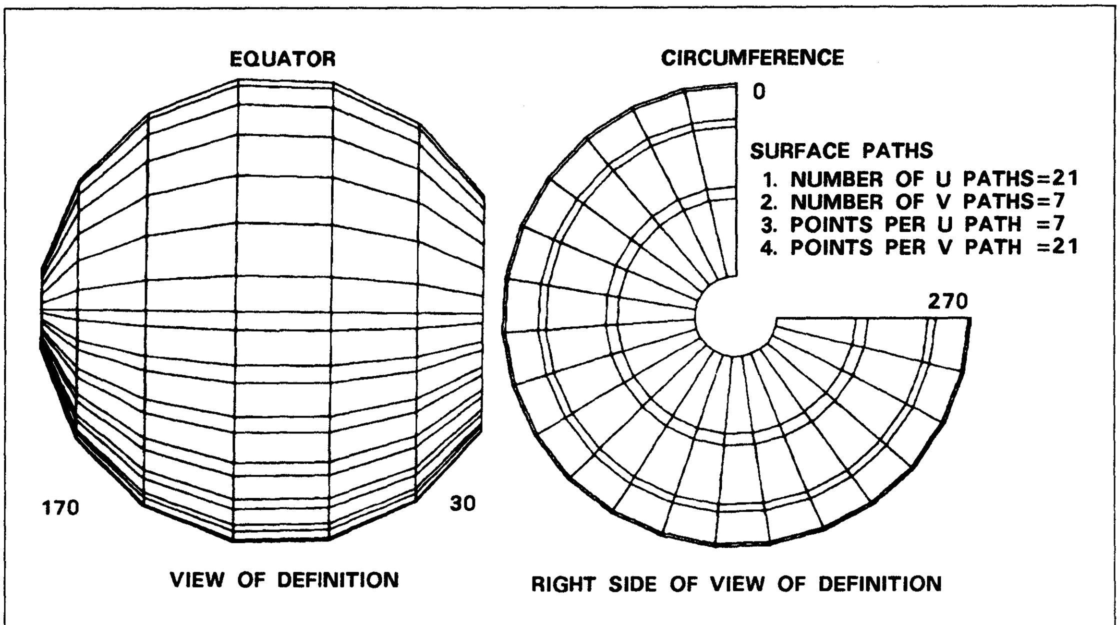


Figure 4-5. Sphere

15.3.10 Cylinder

With this choice, you can create a right circular cylinder. This surface is the set of points at a specified distance from a center axis line. A cylinder may be generated using one of two methods. With the first method, you specify an axis, radius, and start/end angles. With the second method, you specify an arc and a height. The system generates an axis normal to the arc and through its center. The cylinder is defined using the radius and angles describing the arc. 15.1.3 SURFACE PATHS can be used to construct the cylinder with the desired number of surface paths.

CYLINDER DEFINITION 1.AXIS 2.EXISTING ARC	Enter: 1 To specify an axis, radius, and starting and ending angles. Continue to AXIS. 2 To specify an arc and a height. Continue to EXISTING ARC.

The choices in this menu are described next.

Axis

If you select 1.AXIS from the Cylinder Definition menu, the system displays:

```
AXIS FORM
1.EXISTING LINE
2.EXISTING POINTS
```

Enter:

1 To specify an existing line as the axis.

`INDICATE LINE`	Use the graphics cursor to select a line as the axis.

2 To use points to define the axis.

Indicate the starting point of the axis.

```
AXIS START
1.SCREEN POSITION
2.KEY-IN
3.EXISTING POINT
```

Enter:

1 To use the graphics cursor to indicate a location.

2 To enter the xt-, yt-, and zt-coordinates.

3 To use the graphics cursor to select an existing point.

Indicate the ending point of the axis. If you enter 1 or 2 in response to the AXIS START prompt, the system displays:

```
EXTENT
1.SCREEN POSITION
2.KEY-IN
```

Enter:

1 To use the graphics cursor to indicate a location.

2 To enter the xt-, yt-, and zt-coordinates.

Indicate the ending point of the axis. If you enter 3 in response to the AXIS START prompt, the system displays:

```
EXTENT
1.EXISTING POINT
2.DELTA
```

Enter:

1 To use the graphics cursor to select an existing point.

2 To enter the delta xt-, delta yt-, and delta zt- coordinates.

The system defines and displays the cylinder.

After you define the axis, the system displays:

`1.RADIUS = n.nnnn`	Enter the radius of the cylinder. Default is 127 mm (5 in).
`2.STARTING ANGLE = n.nnnn` `3.ENDING ANGLE = n.nnnn`	Enter the starting angle and the ending angles of the cylinder. These angles are measured (in degrees) counterclockwise from the start of the axis. Default angles are 0° and 360°.

Existing Arc

If you select 2.EXISTING ARC from the Cylinder Definition menu, the system displays:

INDICATE ARC	Use the graphics cursor to select the arc to be used for the base of the cylinder.
1.HEIGHT = n.nnnn	Enter the height of the cylinder from the arc.

The system defines and displays the cylinder.

15.3.11 Torus

With this choice, you can create a torus. This doughnut-shaped surface is generated by revolving a circle about a line. Indicate the center point of the torus. 15.1.3 SURFACE PATHS can be used to construct the torus with the desired number of surface paths.

CENTER POINT 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT	Enter: 1 To use the graphics cursor to indicate a location. 2 To enter the xt-, yt-, and zt-coordinates. 3 To use the graphics cursor to select an existing point.

After you select a center point for the torus, the system displays:

1.INNER RADIUS = n.nnnn 2.OUTER RADIUS = n.nnnn	Enter values for the inner radius and the outer radius. The difference between these values is the circle diameter.

The system revolves the circle about a line passing horizontally through the specified point.

15.3.12 Cone

With this choice, you can create a cone by revolving a line about an axis, using one of two methods. With the first method, you specify an axis, half angle, radius of small end, and starting/ending angles. With the second method, you specify an arc, height, and radius of second end. The system generates an axis normal to the arc and through its center. The cone is defined using the radius and angles describing the arc.

```
CONE DEFINITION
1.AXIS
2.EXISTING ARC
```

Enter:

1 To specify an axis, half angle, radius of small end, and starting and ending angles.

2 To specify an arc, height, and radius of second end.

The choices in this menu are described next.

Axis

If you select 1.AXIS from the Cone Definition menu, the system displays:

```
AXIS FORM
1.EXISTING LINE
2.EXISTING POINTS
```

Enter:

1 To specify an existing line as the axis.

```
INDICATE LINE
```

Use the graphics cursor to select a line as the axis. The cone is created with the small end at the selected end of the line.

2 To use points to define the axis.

After you indicate the starting point of the axis (the small end of the cone is at this point), the system displays:

```
AXIS START
1.SCREEN POSITION
2.KEY-IN
3.EXISTING POINT
```

Enter:

1 To use the graphics cursor to indicate a location.

2 To enter the xt-, yt-, and zt-coordinates.

3 To use the graphics cursor to select an existing point.

Indicate the endpoint of the axis (the base of the cone is at this point). If you enter 1 or 2 in response to the AXIS START prompt, the system displays:

EXTENT 1.SCREEN POSITION 2.KEY-IN	Enter: 1 To use the graphics cursor to indicate a location. 2 To enter the xt-, yt-, and zt-coordinates.

Indicate the ending point of the axis (the base of the cone is at this point). If you enter 3 in response to the AXIS START prompt, the system displays:

EXTENT 1.EXISTING POINT 2.DELTA	Enter: 1 To use the graphics cursor to select an existing point. 2 To enter the delta xt-, delta yt-, and delta zt-coordinates.

After you define the axis, the system prompts you to define the cone's half angle, small radius, start angle, and end angle:

1.HALF ANGLE = n.nnnn	Enter the angle between the axis and the outside of the cone. This angle must be less than 90°. Default is 1°.
1.SMALL RADIUS = n.nnnn 2.STARTING ANGLE = n.nnnn 3.ENDING ANGLE = n.nnnn	Enter: 1 To enter the cone radius at the starting point of the axis. 2 To enter (in degrees) the starting angles. 3 To enter (in degrees) the starting and ending angles.

Both starting and ending angles are measured counterclockwise looking down the axis from the starting point to the ending point. Zero degree is in the plane of the screen; positive angles rotate out of the screen. Default angles are 0° and 360°.

The system defines and displays the cone (figure 4-6).

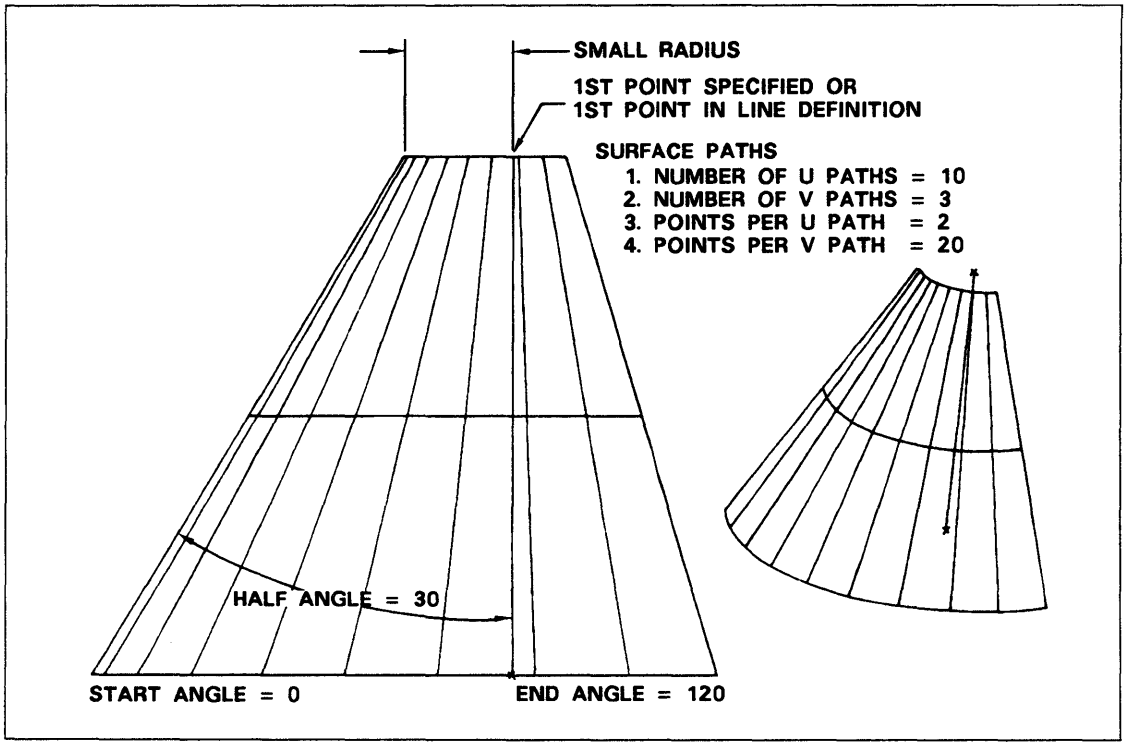


Figure 4-6. Cone

Existing Arc

If you select 2.EXISTING ARC from the Cone Definition menu, the system displays:

`INDICATE ARC OR CIRCLE`	Use the graphics cursor to select the arc to be used for the base of the cone.
`1.SECTION RADIUS = 0.0` `2.HEIGHT = n.nnnn`	Enter the radius of the opposite end of the cone and the cone height.
`INDICATE DIRECTION OF AXIS`	Use the graphics cursor (and depth, if necessary) to indicate on which side of the arc the cone should extend.

The system defines and displays the cone.

15.3.13 Composite Surface

With this choice, you can create a composite surface from a set of surfaces. The surfaces must be connected to each other by common edges; they can be any type of surface except another composite surface.

In certain applications, such as numerical control and fillet surfaces, it is important that surface normal directions be properly aligned. A composite surface does not have its own normals; the normals used are those defined for the individual surfaces that make up the composite surface. Since the direction of a surface normal cannot be reversed once it becomes part of a composite surface, you should make sure that all normals are pointing in the correct direction before defining the composite surface. To view or reverse a normal, refer to 15.2.6.13 REVERSE SURFACE NORMAL.

`INDICATE SURFACE`	Use the graphics cursor to select surfaces.
`INDICATE EDGE`	Use the graphics cursor to indicate the edge that is common to the just-selected surface and the surface that is to be selected next. Upon selection of the last surface, indicate the edge on the opposite side of the previous connection.

The system returns to the INDICATE SURFACE prompt for the selection of up to 150 surfaces. Surfaces should be selected in order and in the same axis direction (horizontal or vertical); figure 4-7 illustrates the correct selection sequence for contiguous, one-directional surface edges. Upon termination of the selection procedure, the system defines the composite surface in one direction.

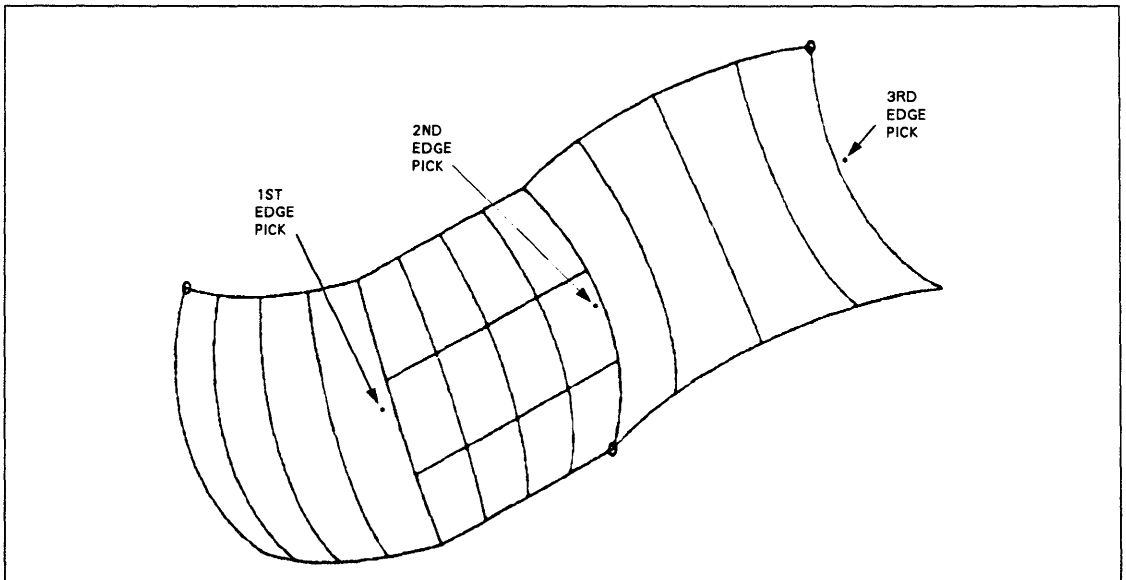


Figure 4-7. Composite Surface

15.3.14 Surface Segment

With this choice, you can create a surface that is a segment of an existing surface.

INDICATE SURFACE	Use the graphics cursor to indicate the base surface (the surface from which the surface segment is to be created).
1.U-MINIMUM PARAMETER = n.nnnn 2.V-MINIMUM PARAMETER = n.nnnn 3.U-MAXIMUM PARAMETER = n.nnnn 4.V-MAXIMUM PARAMETER = n.nnnn	Enter the parameters for the range of the base surface.

The system redisplays the range parameters prompt so that you can select more segment surfaces. If the base surface is to remain unchanged after definition of the segment surfaces, entering] returns you to the next higher menu. Entering [indicates selection is complete and allows you to change the status of the base surface. Indicate the status of the base surface.

BASE SURFACE 1.BLANK 2.DELETE 3.KEEP	Enter: 1 To blank the base surface. 2 To delete the base surface. 3 To keep the base surface unchanged.
INDICATE SURFACE	Use the graphics cursor to indicate which surface is blanked, deleted, or kept.

15.3.15 Projected Surfaces

With this choice, you can create several ruled surfaces simultaneously by projecting a curve or group of curves to a depth or onto a surface. The system defines one ruled surface for each generating curve. Refer to menu 1.15.3 in the ICEM Design/Drafting Introduction and System Controls manual to set the display to transform or model coordinates. These are the three basic Projected Surfaces parameters:

Tolerance

A projected surface calculated by breaking a curve or set of curves into a series of points that are then individually projected to the depth or onto the surface. The tolerance is applied to the resulting projected points to ensure the accuracy of the resulting curves. The tolerance must be greater than zero.

Draft Angle

The draft angle is measured as an offset angle in the projection direction. A projected surface is produced by projecting a curve or set of curves to a depth or on to a surface at this angle. The first part of the projected surface is located on the same side of the first entity selected that is specified in response to the INDICATE SIDE prompt. A zero degree angle is parallel to the projection direction. Allowable angles are zero degrees up to, but not including, 90 degrees.

The application of this angle in the projection operation is based on a side indication entered by the user after the side(s) to be projected have been selected. The side of the projection is then fixed for all the selected curves based on the order of their selection.

Depth

The depth is the zt value of a plane that is parallel to the current workplane. If 1.PROJECT TO DEPTH is chosen, the curve is projected to this plane at the specified draft angle. This is not a relative displacement.

```
PROJECTION TYPE
1.PROJECT TO DEPTH
2.PROJECT TO SURFACE
```

Enter:

1 To generate projected surfaces to a specified depth.

2 To generate projected surfaces to a specified surface.

15.3.15.1 Project to Depth

If you select 1.PROJECT TO DEPTH from the Projection Type menu, the system displays:

```
PROJECTION PARAMETERS
1.TOLERANCE = n.nnnn
2.ANGLE     = n.nnnn
3.DEPTH     = n.nnnn
```

Enter the desired tolerance, draft angle, and depth or accept the default values with].

If you enter an out-of-range tolerance value, the system returns you to the Projection Variables menu and displays:

```
TOLERANCE MUST BE GREATER THAN ZERO
```

If you enter an out-of-range draft angle value, the system displays this message before it returns you to the Projection Parameters menu:

```
DRAFT ANGLE VALUES CAN RANGE FROM ZERO
UP TO 90 DEGREES
```

15.3.15.3 Project to Surface

If you select 2.PROJECT TO SURFACE from the Projection Type menu, the system displays:

```
PROJECTION DIRECTION
1.ALONG ZT AXIS
2.ENTER VECTOR
3.EXISTING LINE OR VECTOR
```

Enter:

1 To cause the projection to occur along the zt axis as it would when projecting to a depth.

2 To allow you to enter the components of the desired projection vector.

3 To allow you to project the selected curves along an existing line or vector onto the surface.

If you select 1.ALONG ZT AXIS from the Projection Direction menu, the system continues to the DRAFT PARAMETERS prompt.

If you select 2.ENTER VECTOR from the Projection Direction menu, the system displays:

```
DEFINITION SPACE
1.TRANSFORM COORDINATES
2.MODEL COORDINATES
```

Enter:

1 To enter the vector using the transform coordinate system.

2 To enter the vector using the model coordinate system.

If you select 1.TRANSFORM COORDINATES from the Definition Space menu, the system displays:

```
VECTOR COMPONENTS
1.IT = n.nnnn
2.JT = n.nnnn
3.KT = n.nnnn
```

Enter the desired it, jt, and kt vector values.

If you select 2.MODEL COORDINATES from the Definition Space menu, the system displays:

```
VECTOR COMPONENTS
1.I = n.nnnn
2.J = n.nnnn
3.K = n.nnnn
```

Enter the desired i, j, and k vector values.

NOTE

The entered values for either coordinate system must define a vector of nonzero length. If a vector of zero length is defined, the system displays:

```
VECTOR CAN NOT BE ZERO LENGTH
```

You are then returned to the VECTOR COMPONENTS prompt.

If you select 3.EXISTING LINE OR VECTOR from the Projection Direction menu, the system displays:

INDICATE ENTITY	Use the graphics cursor to select any displayed line or vector.

After you have defined the projection direction, the system displays:

PROJECTION PARAMETERS 1.TOLERANCE = n.nnnn 2.ANGLE = n.nnnn	Enter the desired tolerance and draft angle or accept the default values with].

If you enter an out-of-range tolerance value, the system displays this message before it returns you to the Projection Parameters menu:

```
TOLERANCE MUST BE GREATER THAN ZERO
```

If you enter an out-of-range draft angle value, the system displays this message before it returns you to the Projection Parameters menu:

```
DRAFT ANGLE VALUES CAN RANGE FROM ZERO
UP TO 90 DEGREES
```

After you have defined the projection parameters, the system displays:

INDICATE SURFACE	Use the graphics cursor to select the surface on which to project the selected curve(s).

For both types of projection, the system displays:

SCREEN SELECT MODE 1.SINGLE 2.CHAIN ⋮	Use the entity selection procedure to select as many as 240 curves for projection on to the previously selected surface or to the specified depth.

If you select composite curves, the curves are expanded into their constituent curves. This may push the total curve count over the maximum limit of 240 curves. If this occurs, the system displays:

```
TOO MANY CURVES FOR ONE PROJECTION
```

You are then returned to the Screen Select Mode menu to reselect the curves.

If the draft angle is greater than zero degrees, the system displays:

INDICATE SIDE	Use the graphics cursor to indicate the side of the first entity selected on which the projected surface is to be generated.

The system now performs the projection operation. If you selected to project onto a surface and none of the selected curves touch the surface, the system displays:

```
NO CURVES PROJECTED
```

15.3.16 Curve Driven Surface

With this choice, you can create a surface by driving one curve (boundary curve) along another curve (centerline curve). An example of a curve-driven surface is pipe ducting, where a circle follows a curve through space.

```
BOUNDARY CURVE FORM
1.ARC OF SPECIFIED RADIUS
2.PREDEFINED CURVE
```

Enter:

1 To use an arc of specified radius, starting angle, and ending angle as the surface boundary.

1.RADIUS = n.nnnn 2.STARTING ANGLE = n.nnnn 3.ENDING ANG = n.nnnn	Enter the radius, the starting angle, and the ending angle of the arc to be driven along the centerline curve. Default radius is 6.35 mm (0.25 in). Default angles are 0° and 360°.
INDICATE CURVE	Use the graphics cursor to indicate the curve that is to have the boundary curve driven along it. The allowable entities for the centerline curve are lines, arcs, conics, two-dimensional splines, composite curves, point set curves, and machining curves. If you selected a predefined curve as the boundary curve, you should select the centerline curve at the end nearest the boundary curve. Using an entity name for selecting the centerline curve might not produce the desired surface.

2 To select an existing curve as the surface boundary.

INDICATE CURVE	Use the graphics cursor to indicate the curve that is to be driven along the centerline curve. Define this curve at either the start or the end of the centerline curve.
INDICATE CURVE	Use the graphics cursor to indicate the curve that is to have the boundary curve driven along it. The allowable entities for the centerline curve are lines, arcs, conics, two-dimensional splines, composite curves, point set curves, and machining curves. You should select the centerline curve at the end nearest the boundary curve. Using an entity name for selecting the centerline curve might not produce the desired surface.

The system defines the curve-driven surface by maintaining a constant distance between the centerline curve and the moving boundary curve.

15.3.17 Reserved for Future Use

15.3.18 Surface Viewing

With this choice, you can enhance surface viewing.

15.3.18.1. Surface Facetting

With this choice, you can display surface entities in shaded color (facetting).

All facets that are part of the same surface are displayed in the same color. To change the color of a facet, you must change the color of the surface entity before using the facetting feature.

SURFACE SELECTION 1.SCREEN SELECT 2.REGION 3.ALL DISPLAYED ⋮	Use the entity selection procedure to select the entities to be facetted.

Before the system displays the facets of the selected entities, the system erases all previously selected facets. When facet painting is complete, the system returns to the surface selection menu. If you enter [or], you will return to the next higher menu level. The system then erases the facets and repaints the screen display.

15.3.18.2 Modify Surface Paths

With this choice, you can modify the number of paths and points per path for specified surfaces. If you select only one surface, the surface path values for that surface will be displayed for modification. If you select more than one surface, the surface path modal values will be displayed.

NOTE

15.1.3 SURFACE PATH MODALS are used to set the number of paths for a new surface.

The system displays:

SELECT SURFACES TO BE MODIFIED.	Select the surface(s) whose surface path(s) you wish to modify.
SURFACE PATHS 1.NUMBER OF U PATHS = R 2.NUMBER OF V PATHS = S 3.POINTS PER U PATH = T 4.POINTS PER V PATH = U	Change any of the values. R, S, T, and U are the current surface path values for the surface or the current surface path modal settings.

After you change or accept these surface path and point values, the selected surfaces will be modified.

Since all four edges of the surface must be displayed for viewing purposes, the minimum values for the number of u and v paths is two. The minimum number of points per u and v paths is three. If you change the number of u or v paths to be less than two, the system automatically sets these values at two. If you change the number of points per u or v paths to less than three, the system automatically resets these values to three.

Some surface types have set (default) values for surface display that cannot be changed. The following surface types have default values of 3 per u path and/or v path: plane, tabulated cylinder, and ruled surface.

Menu 15.4: Solids 5

Menu 15.4: Solids 5

Using Menu 15.4

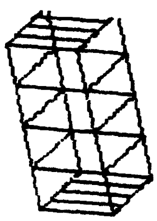
The Solids menu allows you to create several important types of solid figures including hexahedrons, spheroids, circular rods, toroids, and ellipsoids.

The following menu is described in this chapter:

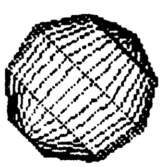
```
SOLIDS
1.HEXAHEDRON
2.SPHEROID
3.CIRCULAR ROD
4.TOROID
5.ELLIPSOID
```

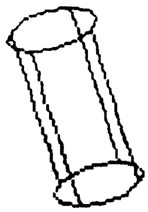
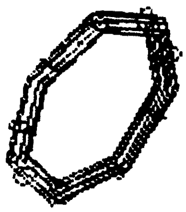
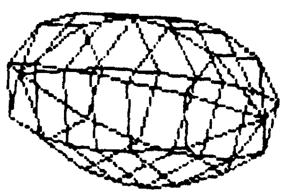
The Solids menu offers the following functions. An example of each solid is presented below its name and menu number.

Menu Choice	Function Description
15.4.1 HEXAHEDRON	Creates a solid bounded by orthogonal planes.



15.4.2 SPHEROID	Creates a solid sphere.



Menu Choice	Function Description
15.4.3 CIRCULAR ROD 	Creates a solid cylinder by revolving a line about a parallel line.
15.4.4 TOROID 	Creates a solid torus.
15.4.5 ELLIPSOID 	Creates a solid by rotating an ellipse 360° about its major axis.

With the Solids menu you can create various types of solids. 15.1.3 SURFACE PATHS can be used to construct the solids with the desired number of surface paths.

The remainder of this chapter describes the choices in menu 15.4.

15.4.1 Hexahedron

With this choice you can create a hexahedron, which is a solid bounded by orthogonal planes. Indicate the center point of the hexahedron.

CENTER POINT 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT	Enter: 1 To use the graphics cursor to indicate a position. 2 To enter the xt-, yt-, and zt-coordinates. 3 To use the graphics cursor to select an existing point.

After you select a center point for the hexahedron, the system displays:

1.DXT = n.nnnn 2.DYT = n.nnnn 3.DZT = n.nnnn	Enter either the distances along the x-, y-, and z-axes from the center point to the hexahedron faces, or the normal distances from the center point to the orthogonal planes bounding the hexahedron.

15.4.2 Spheroid

With this choice, you can create a solid sphere. Indicate the center point of the spheroid.

CENTER POINT 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT	Enter: 1 To use the graphics cursor to indicate a position. 2 To enter the xt-, yt-, and zt-coordinates. 3 To use the graphics cursor to select an existing point.

After you select a center point for the spheroid, the system displays:

FULL EQUATOR?	Enter: Y To construct the spheroid with a full 180° equator. The equator rotates about the vertical axis in the view of definition and is measured from the right to the left. N To construct the spheroid with other than a 180° equator.

If you enter N, the system displays:

1.STARTING ANGLE = n.nnnn 2.ENDING ANGLE = n.nnnn	Enter the starting angle and the ending angle of the equator. Default angles are 0° and 180°.

After you define the equator, the system displays:

FULL CIRCUMFERENCE?	Enter: Y To construct the spheroid with a full 360° circumference. The circumference rotates about the horizontal axis in the view of definition and is measured from the top out of the screen. N To construct the spheroid with other than a 360° circumference. If you enter N, the system displays:
1.STARTING ANGLE = n.nnnn 2.ENDING ANGLE = n.nnnn	Enter the starting angle and the ending angle of the circumference. Default angles are 0° and 360°.

After you define the circumference, the system displays:

1.RADIUS = n.nnnn	Enter the radius of the spheroid. Default is 6.35 mm (0.25 in).

15.4.3 Circular Rod

With this choice, you can create a solid cylinder. This circular rod is generated by revolving a line about a parallel line. The system displays this menu:

```
1.EXISTING LINE
2.EXISTING POINTS
```

Enter:

1 To specify an existing line as the axis.

INDICATE LINE	Use the graphics cursor to select a line as the axis.

2 To use points to define the axis.

1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT	Indicate the starting point of the axis. Enter: 1 To use the graphics cursor to indicate a location. 2 To enter the xt-, yt-, and zt-coordinates. 3 To use the graphics cursor to select an existing point.

If you enter 1 or 2, the system displays:

1.SCREEN POSITION 2.KEY-IN	Indicate the ending point of the axis. Enter: 1 To use the graphics cursor to indicate a location. 2 To enter the xt-, yt-, and zt-coordinates.

If you enter 3, the system displays:

1.EXISTING POINT 2.DELTA	Indicate the endpoint of the axis. Enter: 1 To use the graphics cursor to select an existing point. 2 To enter the delta xt-, delta yt-, and delta zt-coordinates.

After you define the axis, the system displays:

`1.STARTING ANGLE = n.nnnn` `2.ENDING ANGLE = n.nnnn`	Enter (in degrees) the starting angle and the ending angle of the circular rod. These angles are measured counterclockwise from the start of the axis. Default angles are 0° and 360°.
`3.RADIUS = n.nnnn`	Enter the radius of the circular rod. Default is 50.8 mm (2 in).

15.4.4 Toroid

With this choice, you can create a solid torus. Indicate the center point of the toroid.

`CENTER POINT`
`1.SCREEN POSITION`
`2.KEY-IN`
`3.EXISTING POINT`
	Enter:
	1 To use the graphics cursor to indicate a location.
	2 To enter the xt-, yt-, and zt-coordinates.
	3 To use the graphics cursor to select an existing point.

After you select a center point for the toroid, the system displays:

`1.INNER RADIUS = n.nnnn` `2.OUTER RADIUS = n.nnnn`	Enter values for the inner radius and the outer radius. The difference between these values is the circle diameter.

The system rotates the circle about a line that passes horizontally through the specified point.

15.4.5 Ellipsoid

With this choice you can create an ellipsoid which is the solid generated by rotating an ellipse 360° about its major axis.

`INDICATE CONIC`	Use the graphics cursor to select an ellipse.

Menu 15.5: Cross Section Slice 6

Menu 15.5: Cross Section Slice 6

Using Menu 15.5

The Cross Section Slice menu allows you to create a point set curve along the intersection of a surface and a plane. The points in the point set curve are stored as xt- and yt-coordinates in a view parallel to the plane. The zt-coordinates stored with the curve corresponds to the plane depth in that view. Figure 6-1 shows a cross section slice example.

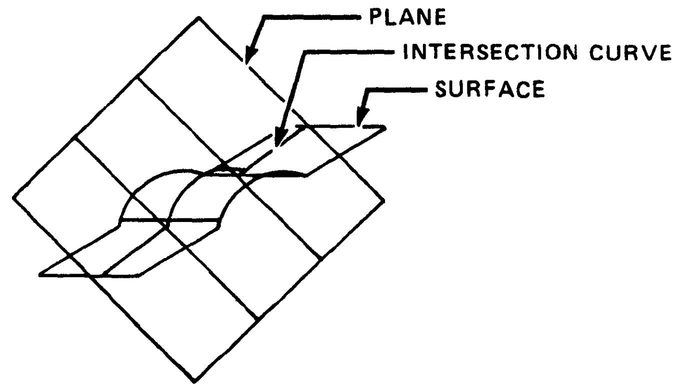


Figure 6-1. Cross Section Slice Example

The system displays these prompts:

`CROSS SECTION SLICE` `INDICATE SURFACE`	Use the graphics cursor to select the surface to be sliced.
`INDICATE PLANE`	Use the graphics cursor to select the slicing plane.

If you select two infinite planes, the system defines and displays an infinite line. If no intersection exists between the two selected planes, the system displays:

```
PLANES ARE PARALLEL
```

You then return to the INDICATE SURFACE prompt. For other cases, the system requests a method to generate the curve.

```
START MODE
1.EDGE INTERSECTION
2.EXISTING POINT
3.POSITION/DEPTH
4.POSITION/PIERCE
```

Enter:

1 To generate all unique curves starting at an edge intersection of the surface with the plane. Continue to EDGE INTERSECTION.

2 To find the initial point by dropping the normal point onto each surface from an existing point. Continue to EXISTING POINT.

3 To find the initial point by dropping a normal point onto each surface from a specified location and the current depth. Continue to POSITION/DEPTH.

4 To find the initial point by piercing the drive surface with a vector. Continue to POSITION/PIERCE.

For choices 1 through 4, the system prompts for a tolerance and a step size:

```
1.TOLERANCE = n.nnnn
```

1 Enter the tolerance, which is the normal distance between the surface and the midpoint of a line segment approximation. A positive tolerance specifies that the step size is the maximum allowable step size. The minimum allowable positive tolerance is 0.005 mm (0.0002 in). A negative tolerance specifies that the tolerance is not used to adjust the step size, but that instead, the step size is a constant.

```
2.STEP SIZE = n.nnnn
```

2 Enter the step size, which is the approximate distance between each point on the intersection curve. The minimum allowable step size is 0.254 mm (0.01 in). If you use very small tolerances, a significant amount of computer time and space is necessary to generate the curves. Use a tolerance only when necessary and use a reasonably large step size to conserve both computer time and space.

When you specify a positive tolerance, the system uses the specified step size as an initial value to find the second point on the curve. Before the system stores this second point, it calculates the tolerance for the associated line segment. If this distance is not within 10 percent of the specified tolerance, the step size is adjusted and a new second point is found. This process is repeated for each new point along the curve, always using the previous step size. If the step size adjustment ever exceeds the specified step size, the system uses the specified step size as a maximum value.

In some cases, you might not be concerned about the tolerance of the plane slice curve, because you only want to know if an intersection exists, want only a rough estimate of the shape of the curve, or already know the intersection is a straight line. Entering a negative tolerance indicates that the tolerance check is not to be done and that the step size is to be used as a constant to conserve computer time.

Edge Intersection

The system defines and displays all possible curves. The system searches the edges of a surface by dividing the edge into a number of line segments and testing each segment for an intersection with the other surface. Initially, the number of segments used for the search is the same as the number of line segments in the surface display (refer to 15.3.18.2 MODIFY SURFACE PATHS). For a very large or long surface, this search may be too coarse. You have two means to control this search. First, if you suspect that an edge intersection may be difficult to find, you can use 15.3.18.2 MODIFY SURFACE PATHS to increase the number of segments in the display. (In general, if a short segment displayed on the edge of a surface appears to intersect the other surface, this point is found.) Second, if the system cannot find an edge intersection, you can tell the system to use twice as many segments as it is currently using. This may be repeated if necessary.

If no edge intersection points are found, the system displays:

`NO EDGE INT FOUND`
`RATIO = n.nnnn`
`DOUBLE SEARCH?`
	Enter:
	
	Y To tell the system to try again using twice the current number of segments in its search (2 times n).
	
	N To return to the START MODE prompt to select another method.

Existing Point

`INDICATE POINT`	Use the graphics cursor to select an existing point.

The system drops normal points onto the plane and the surface from the selected point. It then defines the initial point as close as possible to the two normal points. If the system cannot find an initial point, it displays:

```
CANNOT FIND START PT
```

The system redisplays the START MODE prompt for the selection of another start option.

Position/Depth

INDICATE POSITION	Use the graphics cursor to indicate a location.

The system drops normal points onto the plane and the surface from the specified location and current depth. It then defines the initial point as close as possible to the two normal points. If the system cannot find an initial point, it displays:

```
CANNOT FIND START PT
```

The system redisplays the START MODE prompt for the selection of another start option or another position.

Position/Pierce

INDICATE POSITION	Use the graphics cursor to indicate a location.

The system pierces the surface to be sliced with a vector perpendicular to the screen and through the specified location. It then defines a starting point as close as possible to the pierce point. If the system cannot find an initial point, it displays:

```
CANNOT FIND START PT
```

The system redisplays the START MODE prompt for the selection of another start option or another position.

With all four modes, after locating the starting point (or initial points) the system defines and displays the curve (or curves). If a view exists parallel to the slicing plane, this operation is complete. If no view exists parallel to the slicing plane, the system creates one and informs you.

NEW VIEW NO. IS n, ACKNOWLEDGE	Enter] to acknowledge new view n.

The system then returns to the INDICATE SURFACE prompt.

Menu 15.6: Surface Development 7

Menu 15.6: Surface Development 7

Using Menu 15.6

The Surface Development menu enables you to perform two functions related to the development (flattening, unwrapping) of thin sheet material.

The menu described in this chapter is:

```
DEVELOPMENT FUNCTIONS
1.DEVELOPABLE SURFACE LAYOUT
2.DEVELOPABLE FEATURE LAYOUT
```

A description of these two functions follows:

Menu Choice	Function Description
1.DEVELOPABLE SURFACE LAYOUT	Creates a flat pattern from existing surfaces including all developable surfaces, cylinders, cones, and tabulated cylinders.
2.DEVELOPABLE FEATURE LAYOUT	Wraps or unwraps selected entities to or from a surface.

The remainder of this chapter describes the choices in menu 15.6.

15.6.1 Developable Surface Layout

With this choice, you can generate a flat pattern from an existing surface. Surfaces that can be laid flat are all developable surfaces, cylinders, cones, and tabulated cylinders. Cylinders and cones are solved analytically and define lines and arcs. Other surfaces result in lines and point set curves. The layout begins at the starting end of the first curve chosen when the surface was created.

INDICATE SURFACE	Use the graphics cursor to select the surface.
SECTIONS = nn	Enter the number of rectangular-like sections to define the surface layout. The number must be greater than zero and less than 100. Default is 29.
POINT MODE 1.SCREEN POSITION 2.KEY-IN 3.EXISTING POINT	Indicate the screen position for the bottom left of the layout. Enter: 1 To use the graphics cursor to indicate a position. 2 To enter the xt-, yt-, and zt-coordinates. 3 To use the graphics cursor to select an existing point.

15.6.2 Developable Feature Layout

With this choice, you can wrap or unwrap selected entities to or from a surface.

This function is complementary to 15.6.1 DEVELOPABLE SURFACE LAYOUT, and makes it possible for features on surfaces such as holes, cutouts, and lettering to be unwrapped with the surface to the layout. It is primarily intended to assist in the manufacturing of sheet metal parts which are to be formed by single curvature bending around a jig. The features developed can be cut from the flat sheet prior to forming. Also provided is the ability to take flat shapes from the layout and wrap them onto the surface.

A typical use of this function would be the positioning of labels on bottles. It can also be used in sheet metal design to determine whether or not a simple shape on the flat sheet, such as a circular hole, will retain its shape on the surface within a specified tolerance. Likewise, any point or curve type can be mapped in either direction between the surface and its layout.

A point is created if points are being mapped. Three-dimensional splines are created when other entities are mapped.

DEVELOPABLE FEATURE LAYOUT INDICATE ENTITY	Select a developable surface and the two curved edges of the corresponding layout.

The surface must have been created by using 15.3.5 DEVELOPABLE SURFACE and the layout must have been generated by 15.6.1 DEVELOPABLE SURFACE LAYOUT. The three entities can be selected in any order.

DIRECTION OF MAPPING 1.UNWRAP FEATURE (SURFACE TO LAYOUT) 2.WRAP FEATURE (LAYOUT TO SURFACE)	Indicate which function to perform.
INDICATE CURVE	Indicate the desired point or curve to map to the surface or its layout, depending on the choice from the Direction of Mapping menu.

No check is made by the system to determine if the point or curve lie in the surface (in the case of unwrapping), or in the layout (in the case of wrapping). If the point or curve is not in the surface (or layout), it is internally projected onto the surface (or layout) before performing the unwrapping (or wrapping). This is of value if the feature stands out from the surface.

Appendixes

Glossary A

A

Alphanumeric

The letters of the alphabet (A through Z) and the digits (0 through 9).

ANSI

American National Standards Institute. English (U.S. customary) dimensions are specified in feet and/or inches. Metric dimensions are specified in millimeters.

Approximation

A process of computing a curve that interpolates the given point set.

Attention Indicator

A small oval that the system displays on an entity to indicate that the entity has been selected.

B

Bezier Curve

A polynomial represented (in terms of Bernstein's basis functions) so that the associated coefficients can be viewed as the vertices of a polygon. The first and last vertices lie on the curve. The other vertices do not lie on the curve but define the general shape of the curve.

Blanking

A process applied to entities in the current part where the entities remain in the part, but are not displayed on the screen. Contrast with Deleting.

Blending

The process of connecting two entities (two end points of a curve) by using a Bezier curve (matching tangents and curves).

C

Chain Select

The process of selecting a series of contiguous curves.

Chamfer

A straight line that joins two lines, arcs, or curves. It is generally used to change a sharp corner into a blunt one. Contrast with Fillet.

Contiguous Curves

Adjacent curves that share a common endpoint or that have endpoints within a short distance of each other.

Coordinate Axes

The lines that form a right-handed, three-dimensional frame of reference. The model axes (x, y, and z) and the transform axes (xt, yt, and zt) are referenced from a shared origin point in which x=0, y=0, and z=0.

Coordinate Space

A space created by the position (rotation and translation) of the coordinate axes. There are two types of coordinate space: model and transform.

Coordinates

The position of a point in relation to the x-, y-, and z- or xt-, yt-, and zt-axes.

Curve

A line, arc, conic, or two-dimensional spline. In some cases, it also includes points, three-dimensional splines, strings, composite curves, machining curves, and Bezier curves.

Curve Projection

A function in which a curve or set of curves is mapped onto a surface or to a specified depth by breaking down the curves into points and mapping those points.

D

Data Base

An integrated set of files, tables, arrays, and other data structures.

Data Capture

The system capability to save as a variable either a default parameter value or a previously entered parameter value displayed on the screen.

Default Value

A value used by the system if] is entered in response to a data entry prompt. It is also the initial value assigned to a modal until it is changed.

Degree (of a Polynomial)

The highest exponent of the polynomial.

Deleting

The removal of entities from the data base. Contrast with Blanking.

Depth

The current value of the workplane along the z- or zt-axis of the workspace.

DIN

Deutsche Industrie Norme (German industry standards).

Directrix

A fixed curve with which a generatrix maintains a given relationship in generating a geometric figure.

Dormant Entity

An entity created by the system for the purpose of defining another entity. Dormant entities cannot be displayed or manipulated. All dormant entities have sequence numbers.

Draft Angle

A draft curve parameter that controls the angle at which the selected curve(s) are projected, measured as an offset from the projection direction.

Dragging

The movement of selected entities around the screen in order to view their position at any given point. The entities are drawn in the accepted location.

E

English (U. S. customary) Mode

The condition in which all entities are defined in English (U. S. customary) units (inches and/or feet).

Entity

The representation of a geometric construction in the ICEM DDN data base. Examples are points, lines, arcs, and spheres.

Entity Type Numbers

System-assigned code numbers used to identify specific entity types such as line, group, and copious data.

F

Fillet

An arc that joins two lines, arcs, or curves. It begins and ends at a point of tangency on each of the two lines.

Font

The method used to represent a line or curve in a display. Examples of fonts are solid, dashed, phantom, and centerline.

Function Control Key/Square

A keyboard/tablet character that has a special function in ICEM DDN such as menu and entity selection.

G

Generatrix

A point, line, or surface whose motion generates a line, surface, or solid.

GPL

A graphics programming language. See the ICEM Design/Drafting GPL manual.

Graphics Cursor

The cursor symbol used to locate or define entities by screen position. The graphics cursor can be crosshairs or some other cursor symbol.

GRAPL

A graphics programming language. See the ICEM Design/Drafting GRAPL Programming Language manual.

Group

A collection of entities selected by the user to be defined as a group in the data base.

H

Hidden Lines

Line segments that are obscured from view in display of a three-dimensional object.

I

ICEM DDN

Control Data's integrated computer-aided engineering and manufacturing software application for design, drafting, and numerical control.

ICEMDDN

The name of the direct access file that contains ICEM DDN.

Implicit Point

A point automatically generated by the system at curve endpoints, midpoints, intersections, and circle centers.

Interpolation

A process of computing a curve that satisfies all the conditions imposed; for example, a curve that passes through all given points.

L

Local File

Any file that is currently associated with a job.

M

Menu

A list of options used to perform operations in ICEM DDN.

Message

Text written by the system to display information, errors, or warnings. See also Prompt.

Mirror

An operation that enables you to produce a copy of a set of entities in the part by rotating them 180 degrees around a selected line.

Modal

A user-assigned status or value that controls the operation of ICEM DDN. Examples are modals for setting system decimal places and for activating the grid.

Model Coordinate Space

The space created by the position of the x-, y-, and z-axes. This space is displayed as the front view, view 1. See Transform Coordinate Space.

N

Normal

Perpendicular.

Normal Vector

A vector that is perpendicular to the tangent vector for a planar curve or perpendicular to the tangent plane for a surface at a point on the curve or surface.

NOS

Network Operating System.

O

Offset Curve

A curve that is a specified distance from an existing curve. The offset curve has the same slope along its length as the existing curve.

Operating System

The set of system programs that controls the execution of user programs and commands.

Operation Complete --]

A keyboard/tablet operation that completes the current operation.

Operation Reject -- [

A keyboard/tablet operation that rejects the previous operation. The system returns to the preceding prompt.

P

Parameter

A variable whose values determine the operation or characteristics of a system.

Part Name

A series of alphanumeric characters that identify a part or drawing used in conjunction with a sheet number.

Permanent File

A file saved by the system between terminal sessions.

Piecewise Linear

A curve divided into segments, each of which is a linear segment.

Piecewise Polynomial

A curve divided into segments, each of which is a polynomial.

Point

A geometric construction located by x-, y-, and z-coordinates.

Point Set Curve

A piecewise linear or polynomial approximation to an actual curve. It is a series of points stored as transform coordinates.

Pointer Number

The number assigned to an entity that describes the location in the data base of that entity's parameters.

Polygon Points

The coefficients of a Bezier curve.

Prompt

A screen display requiring user action. See also Message.

R

Region Select

A process of selecting all the entities within or outside a specified region.

Rescale

To change the scale factor and redraw a part using the new scale so all entities in the part are displayed.

Rotate

To revolve a construction about an axis.

S

Scalar

In any context where both vectors and real numbers are present, scalar refers to a real number.

Scale Factor

Ratio of the current display with respect to the data base.

Sequence Number

A unique sequential number associated with each entity.

Sheet Number

A subdivision of a part name. Sheets can be numbered from 0 to 9,999 with a part name.

Single Select

A process of selecting individual entities one at a time.

Spline

A piecewise polynomial constructed with specified smoothness between pieces.

Standard Views

The system-generated coordinate systems from which the user can view a display of the part surfaces from these view points: front, back, top, bottom, right, left, isometric, and auxiliary.

Surface Normal Point

Given a 3-D point, a surface normal point is the point found by dropping a normal vector from the 3-D point onto a second surface. The surface normal point is the point on the second surface closest to the 3-D point.

Surface of Revolution

Surface produced by the rotation of a curve around an axis through a specified angular displacement.

Surface Parameters

A pair of numbers (u,v) that specify positions on a surface.

Surface Paths

A modal that allows users to define the surface of objects along two parameters of flow, u paths (horizontal) and v paths (vertical).

System

The ICEM DDN software system.

T

Tabulated Cylinder

Surface generated by the translation of a curve along a specified direction with upper and lower limits on the distance of translation.

TAPE3

A local file on which parts are filed and from which parts are retrieved.

Temporary File

A nonpermanent file associated with a job only during job processing.

Text

Alphanumeric characters and symbols used in notes, labels, and dimensions.

Transform Coordinate Space

The space created by the position of the xt-, yt-, and zt-axes. This space is displayed in any view other than view 1. See Model Coordinate Space.

Transformation of Coordinates

The mapping of the xt-, yt-, and zt-coordinates into the x-, y-, and z-coordinates.

Translate

To relocate entities on the screen display by assigning them different origin coordinates.

U

U Path

A curve on a surface in one of two parametric directions. See V Path.

Unblank

A procedure for making visible entities that have been blanked by the user.

V

V Path

A curve on a surface in one of two parametric directions. See U Path.

Vector

A directed line segment. Two directed line segments are regarded as defining the same vector if they have the same magnitude and direction.

View

A display of coordinate space.

View Apparent

The view distortion that occurs when a 3-D drawing is displayed on a 2-D screen. Points that appear close together may be far apart due to differences in depth.

W

Work Plane

A specific plane in workspace on which two-dimensional entities are constructed.

Work View

The view in which screen input is accepted.

Workspace

The space in which entity construction and specification occurs.

Z

Z-Clip

The specification of parameters for the display of a drawing. Elements outside of the z-clip window are not displayed. No change occurs to those elements not displayed.

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Comments (continued from other side)

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