

SURFCAM Help File Contents

This help file is intended as a quick reference for menu selections and pop-up parameters. The user's manual provides a detailed description of all SURFCAM operations and functions. The help screen displayed describes the current menu or pop-up parameter line highlighted.

Analyze Menu

The *Analyze* menu enables you to display coordinate information about elements, locations and distances on the screen.

NOTE: When *Lathe* is selected the locations are displayed using the *Lathe Input* and *Lathe Views* described in the SURFCAM.CFG file.

NOTE: Distances and locations are displayed using the current coordinate system. When the *Coords* parameter is set to *View* in the *Status* menu, the locations and distances displayed are relative to the current construction coordinate system.

Angle The Angle option on the Analyze menu enables you to display the true and projected angles between any two lines. The true angle is the angle as measured in 3D space from one line to the other. the projected angle is the angle as it appears on the screen.

The difference between the true and projected angles is more easily understood by viewing the axes gnome. Although all 3 axes are (by definition) 90 degrees from each other in 3D space, the 2D image on the screen indicates the different angles (depending on their orientation).

Distance The Distance option in the Analyze menu enables you to display the distance between any two locations (as indicated from the location menu).

Min/Max This option analyzes the distance between two surfaces or splines.

Elements The Elements option on the Analyze menu enables you to display detailed information about any single element.

NOTE: You may single out, or mask, elements to analyze by type and color. Masking makes element selection easier when trying to analyze a complex 3D image in which many elements are close or overlapping. Refer to the Status window section for a complete description of the Mask function.

Location The Location function on the Analyze menu enables you to display the X, Y, and Z coordinates of a specified location. This option uses the Location menu.

Time Choosing the Time option will display the current date and time (from your computer's internal clock) in a small pop-up.

Status The Status option allows you to determine the amount of memory space that has been used by (and is available in) SURFCAM's database and display list.

Version This option allows you to access SURFCAM version and options information.

Tangency This option allows you to analyze the edges of a surfaces or splines for the minimum and maximum angles for tangency.

Surf Pnts The Surf Pnts option allows you to analyze points on a surface for surfaces tolerances.

Curvature This option analyzes a surface or spline for the minimum radius.

Files / CADL / CDL Write Menu

This option allows you to save all active elements, or selected elements, into a CDL file.

Chain Menu

The *Chain Menu* is displayed at various times.

The following parameters may or may not appear on your menu depending on the operation in which you are currently engaged.

Single

This option is used to select a single element or a series of single elements. It avoids extra steps in the selection process. If a series of elements are selected, they are not chained together. After you make your selections and click *Done*, each separate element you have selected is acted upon by the SURFCAM feature that is using the chaining menu.

Chain

This option is used to chain several connected elements together. To use *Chain*, select a point near the beginning of the first element followed by a point near the end of the last of the connected elements you wish to chain. After selecting the beginning point, the *Single* and *Chain* buttons are de-activated and the *Done* button is changed to the *Close* button.

Another chaining method is to use *Close* to chain all of a set of connected elements together after first selecting a point near the beginning of the first element. An alternate way to chain all the connected elements together is to click the beginning point a second time.

In some situations, when chaining, SURFCAM encounters branch points or forks in the road and cannot determine the next element to chain. When this occurs, the *Chain next* menu is displayed and you are prompted with *Branch point found, select next element or end to finish*. Select the next element to chain or select the *End* button.

Vector

The vector chaining method identifies profiles that have a vector attached to them. The vector defines the direction to offset the tool path.

To create a tool path using vectors, select *Vector* from the chaining menu. SURFCAM will then display the *Selection* menu to select the vectors. Use any one of the selection options to select the vectors representing the profiles to machine. SURFCAM will highlight the profiles. Select *Done* when all of the desired profiles have been highlighted. Select *Done* again and SURFCAM will prompt for an INC filename.

NOTE: Refer to the **Create Vector** section in the **Reference** chapter of the manual to create a vector.

Auto

When you select *Auto*, SURFCAM displays the *Select* menu and you are prompted to *Select contours(s) to be chained, then Done*.

Plunge

Use this option to specify plunge locations. The system will automatically move the tool to the closest user-defined plunge point and feed to the cutting depth, then to the beginning point of the cycle's pass. If plunge points are not specified, plunge positions will be automatically calculated. The system disregards any Z axis coordinate since the cycle determines the depth to which to plunge.

To define a plunge point, select *plunge* from the menu. The system will display the location menu to define the plunge point coordinate. Select the appropriate locations and select *Done*. Select *Done* again and the system will prompt for an INC

filename.

NOTE: This may be done before or after a cycle has been defined.

Chain Opt.

When this option is selected, the system displays a pop-up containing the *Planar*, *Tangent* and *Angle Tolerance* parameters.

Done

Selecting *Done* completes the chaining session and signals SURFCAM to proceed to the next step in its current processing.

Create / Surface / Blend

Select the cross sections that represent the edges of the surface. The edge of an existing surface can also be selected by picking it. If the edge of a surface has been picked, the resulting surface will be tangential to the original surface within the specified tolerance. At least two edges must be selected.

- 2 edges* A ruled surface is created between the two edges.
- 3 edges* The edges must be touching each other. A three-sided surface will be created with a very small fourth side created between the first and second edges picked.
- 4 edges* A surface is created between the surface edges picked.

Chain Next Menu

End

A branch point has been found in the chaining process. You should select the next element or select the *End* option to finish.

Create / Surface / Composite Menu

The Composite option does not create any new geometry. This option allows you to trim and link a group of existing surfaces into a single Composite surface.

The surfaces that used to create a composite surface are trimmed, one at a time, to fillet edge splines, surface intersection splines, projected splines, and/or cutter intersection splines.

The following notes apply to composite surfaces.

- NOTE: The Backup menu option can be used at any time during the creation of a composite surface as an Undo editing feature. Each time Backup is clicked, the previous surface that was added to the composite will be removed, and returned to the original untrimmed state.
- NOTE: Splines that have been used to create a composite surface cannot be trimmed, broken or optimized. This limitation preserves the integrity of the composite surface. Prior to creating a composite, splines can be trimmed or broken to establish the correct trim boundary.
- NOTE: Once a composite surface has been created, it is a single entity. When you delete any element that belongs to the composite, the entire composite will be deleted. This preserves the integrity of the composite surface.
- NOTE: Once a composite surface has been created, it is possible to add additional surfaces to it or to remove the last surface that was added to it. This may be accomplished by selecting Create Surface Composite and pick the existing composite surface. SURFCAM prompts you for confirmation. You may either add more surfaces by picking a new surface to add or remove the last surface by selecting Backup before selecting any other function. This will undo the trimming on the last surface added to the composite. You may continue removing surfaces from the composite by choosing Backup.

Convert Menu

This menu allows you to Read or Write the desired file type.

Read You are prompted to select the desired file to read into SURFCAM.

Write You may output the desired file type with this selection.

Files / Backplot Menu

Choosing INC from the Files menu presents another menu that can be used for backplotting and merging INC (tool motion) files.

NOTE: When merging files, choosing Backup or Main from the Files INC menu will automatically execute the Done option. This means that you must use the Merge option to resume merging a file after leaving the menu.

Read The Read option is what is commonly referred to as a backplot. This allows you to read in and display tool motion. This is generally done for tool path verification purposes.

Start new Choosing this is the first step in merging existing INC files into a new master INC file. When you select Start new, SURFCAM prompts for a filename of the new INC file to create. This is the filename to use for the files to be merged into. After entering a filename, you must use the Merge option.

Merge This option can be used two different ways, either with or without the Start new option above, to append information in two or more INC files. Like the Start new option above, SURFCAM prompts for a filename when you select Merge from the menu. The two methods to use this option are explained below.

Without Start new The first filename entered is not displayed on the screen. It is identified as the file to merge files into.

Upon specifying subsequent files to merge, by choosing Merge again, SURFCAM displays the tool motion contained in the incoming INC file and adds the incoming tool motion commands to the end of the file that was specified in the first Merge operation.

With Start new Since the Start new operation causes a new file to be created, upon giving the filename, SURFCAM displays the tool motion contained in the incoming INC file and copies the incoming tool motion commands into the started file.

Once an INC file is started by either the Start new or Merge procedures, the Merge procedure may be repeated to add together as many INC files as desired.

NOTE: When finished merging all the desired INC files, use the Done option below.

Done This option is used when finished merging INC files. The master file is saved and you may exit the INC menu by pressing Backup or Main.

While the INC file is being read, the tool motion is displayed on the screen if the X, Y, Z space of the tool motion is visible on the screen.

Create / Line / Angle Line Menu

This menu allows you to enter the desired line angle information and location.

Info change This option displays a pop-up for you to enter the line angle and length.

Location The Location menu is displayed for you to select the line location point.

Undo If this option appears in the menu, you may use it to delete the line you just created.

Create / Line Menu

The Line option allows you to create lines with a variety of methods. The menu below is displayed when this option is selected.

<i>Tangent</i>	This option allows you to create a line Tangent to elements or locations. Locations may be selected by clicking the Location button to display the Location menu.
<i>Endpoints</i>	This option allows you to create a line by selected the desired locations for the endpoints of the line. The Location menu is displayed for you to select the endpoint locations.
<i>String</i>	The String option is used to create a series of single line elements that are in an end-to-end orientation. This is done simply by entering locations. SURFCAM joins all the locations with lines, from the first point to the next.
<i>Horizontal</i>	This option is used to create lines that are oriented horizontally in the cview.
<i>Vertical</i>	This option is used to create lines that are oriented vertically in the cview.
<i>Both</i>	This option is used to create lines that are oriented horizontally and/or vertically in the cview.
<i>Angle</i>	The Angle option gives you access to an exceptionally flexible method of line creation. There are three different ways, listed below, to use this option to create a line of a desired length at a specified angle.
<i>Location or Point</i>	The line is created at the given angle starting at the location or point specified.
<i>Arc</i>	The line is created tangent to the arc along the angle specified.
<i>Line and Location</i>	The line selected is the angle reference. The angle specified is relative to the line.
<i>Cross prod</i>	This option allows you to create a line that is the cross product of two lines. A Cross Product is the term commonly used for a line that is normal (perpendicular) to the plane described by two other lines.
<i>Offset</i>	This option allows you to offset a line by a distance, to a location, or tangent to an arc or circle.
<i>Rectangle</i>	This option is used to create a rectangle by selecting two points that are diagonally opposite corners of the rectangle.
<i>Undo</i>	This option is available after an element is created. Selecting this option deletes the last element created.
<i>Location</i>	Select this option to display the Location menu when creating a line Tangent.

Create / Points Menu

The *Points* option is used to create a number of points. In addition to creating one point at a time by using the *Project* function, you can instantly create a series of points using other options on the *Create Points* menu.

- | | |
|------------------|---|
| <i>Nodes</i> | You may create a series of points at the nodes of a selected spline. |
| <i>Error</i> | This option creates a series of points along a selected spline such that a string of lines drawn between the points will not deviate from the original spline more than the error amount you specified. |
| <i>Distance</i> | This option creates a series of points at a given distance along a chain of elements. |
| <i>Project</i> | This allows you to create points (and lines) on splines and surfaces by projecting locations onto them. The location can be projected through the <i>Cview</i> or normal to the surface or spline. |
| <i>Rectangle</i> | This option creates a series of points in a rectangular array pattern. Define the array along the X and Y axes. |
| <i>Circular</i> | This option creates a series of points around an arc. |

NOTE: The number of points must be greater than zero.

Create / Arc Menu

The Arc function on the Create menu displays a submenu of different ways that you can create arcs.

- 3 Points* This option allows you to create an arc by specifying three points on the screen. These three points are selected using the Location menu. Each desired point may be selected with a different method from the Location menu.
- Center/Dia* When using this method of arc creation, SURFCAM displays a pop-up for you to enter the diameter, start angle and end angle of the arc.
- Center/Rad* This option works exactly the same as the Center/Dia option described immediately above except that SURFCAM prompts you for the Radius instead of the diameter.
- 2 pnts Dia* This option allows you to create an 180 degree arc by specifying two end points. The points are selected using the Location menu. Each desired point may be selected with a different method from the Location menu. The points selected define the diameter of arc. The arc is created in the counterclockwise direction from the first point to the second point.
- Ctr/st/end* This option allows you to create an arc by specifying the center of the arc, the start point and a point defining the angle of the arc. The Location menu is displayed for you to select the required locations.
- The first location selected is the center of the arc. The distance between the second location selected and the center defines the radius of the arc and the start point of the arc. The third location selected defines the angle between the start and end point of the arc. The last location is not required to be a point on the arc. When the locations are selected, the arc is created in the counterclockwise direction from the start point to the third point projected on the arc.
- Offset* This option allows you to offset an arc by a distance, to a location, or tangent to another entity.
- Tangent 2* You may create an arc tangent to two elements. When this option is selected the Arc menu is displayed for you to select the arc radius and trim options.
- Tangent 3* You may create an arc tangent to three elements. When this option is selected you are prompted to select the three elements that are to be tangent to the arc. The elements selected is restricted to points, lines, arcs and circles.
- Undo* Deletes the last arc that was created.

Arc Options

These options apply to creating arcs and fillets.

<i>Change rad</i>	Change the arc radius.
<i>Trim</i>	Indicate the number of elements neighboring the arc to be trimmed. You may select 0, 1, or 2. This option toggles between the choices when it is clicked.
0	This option creates the arc without trimming any of the elements.
1	The first element selected is trimmed to the arc. The other element is not trimmed.
2	Both elements are trimmed to the arc.
<i>Sweep</i>	Change the sweep of the fillet from less than 180 degrees to greater than 180 degrees.
<i>Location</i>	This option displays the Location menu to select arc locations.
<i>Flip arc</i>	Flip an existing arc from less than 180 degrees to greater than 180 degrees or vice versa.

The following options are displayed ONLY after the arc has been created.

<i>Other</i>	Display other tangent possibilities.
<i>Reverse</i>	Reverse the trimming of the elements, and optionally reverse, or flip, the arc itself.
<i>Undo</i>	This allows you to return the elements to the original configuration and remove the created arc. All elements are untrimmed to the original size.

Create / Surface / Composite / Bounds Menu

Select the boundary curve and select Done when complete. You may use connect or no connect for the boundary curve. This option toggles between the two selections.

Connect When multiple tool containment splines exist, you may connect the splines. Selecting the Connect option creates an invisible boundary between the beginning point of the current spline and the ending point of the previous spline.

No Connect This option does not connect multiple boundary lines.

Select Menu

The Select menu allows you to select specific elements displayed on the screen. This menu is used by a variety of system functions.

When this menu is displayed, the last selection method used is active. The selection method is highlighted in white.

- | | |
|-------------------|--|
| <i>Single</i> | With this option you can chain all of a set of connected elements with a single click on any part of any one of the connected elements. |
| <i>Within</i> | This option lets you use a rubber band box to identify where chaining will begin. The chaining starts at the end points of any element completely enclosed in the box and extends in both directions until all connected elements are chained. Several separate groups of connected elements can be chained at the same time if at least one complete element from each group is completely enclosed in the box. |
| <i>Intersect</i> | This option is similar to <i>Within</i> except that the element whose end points are to be the start of the chaining need only intersect the rubber band box, not be completely enclosed by it. |
| <i>Visible</i> | This option will cause all groups of connected elements that are visible in the screen work area to be automatically chained at one time. |
| <i>MultiSelOn</i> | This button is set to <i>On</i> so you can make multiple selections. You cannot change this setting. |
| <i>Done</i> | Returns you to the SURFCAM feature menu that is using the <i>Chaining</i> menu. |

Display Menu

The Display menu enables you to manipulate the image. These functions may be selected by clicking the desired icon at any time.

<i>Fit</i>	This option is used to shrink or expand the graphic image to fit the screen viewing area.
<i>Rotate</i>	You may change the viewing angle of the graphic display. This is considered dynamic rotation. The elements displayed are graphically rotated while the right mouse button is pressed and the mouse is being moved. The arrow keys may also be used to rotate the display.
<i>Zoom</i>	You may expand the graphic image to a defined region. This is accomplished with the use of a rubber-band box that defines the area to fit to the screen display.
<i>Unzoom</i>	This option allows you to shrink the graphic image by half the current screen scale.
<i>Pan</i>	You may slide the graphic image by using the arrow keys or by holding down either the right or the left mouse button. This function uses the current screen scale.
<i>Last View</i>	Use this option to display up to the twenty most recent screen displays. This includes view changes, as well as any display function.
<i>Next View</i>	You may display the previous twenty screen displays in the reverse order of <i>Last View</i> .
<i>Redraw</i>	This option is used to refresh the graphic viewing area. This is useful after deleting elements. The <i>Redraw</i> function displays all active elements for the visible layers.
<i>Shade</i>	This option may be used to shade the current drawing. The <i>Shade</i> menu is displayed for you to select the desired shading and shade the drawing.

Files / DXF Write Menu

This option allows you to save all active elements, or selected elements, into a DXF file in any directory.

Rapid Frequency

You may select *per Pass* or *per Region* for this parameter.

per Pass This causes the tool to lift and reposition each time it reaches the hole.

per Region This causes the tool to lift and reposition only when it has to. SURFCAM will force the tool to remain in contact with the current surface until it is completely machined before lifting.

Edit Menu

The *Edit* menu allows you to change the attributes of elements. You may also edit an INC file created during any machining sequence.

Color The *Color* option allows you to change the color of one or more elements using the standard *Select* menu options (*Single*, *Within*, *Intersect*, *Visible*, and *MultSelOn/Off*).

NOTE: If the *MultSelOn/Off* option is set to *Off*, selecting an element changes its color immediately. You may continue selecting elements, one at a time, to change them to the same color.

If the *MultSelOn/Off* option is set to *On* you can select a number of elements which will be highlighted as they are selected. To change the color of these elements, click on *Done*.

Flip Element The *Flip Element* option will create the complement of an existing arc or vector. This option displays the other portion of the circle not displayed with the arc, or it flips the vector arrow so it points in the opposite direction.

Trim/Break This option allows you to trim or break elements.

Increment The *Increment* option is used to specify locations at even increments along an element.

Splines SURFCAM allows you to change the attributes of splines including changing the locations of the nodes, the tangent vector directions and magnitudes.

Surfaces The *Surfaces* option allows you to change the attributes of a surface.

Poly lines The *Polyline* option results in another submenu that allows you to access options to add, delete, or move nodes within a polyline, or to create individual line elements that resemble a chosen polyline.

NC This option is used to graphically edit the contents of tool motion or INC files. An INC file is made up of a series of records that can contain movement information, rapid moves, feed moves, arc moves, etc., and other information that is needed to produce a complete NC program.

The *Edit NC* function enables you to easily edit tool motion graphically by moving, inserting, or deleting tool motion on the screen. Since an INC file also contains tooling and machine tool parameters, the *Edit* option allows you to easily insert, delete and/or alter this information as well.

Tangency This option allows matching tangents between two surfaces or splines. This option modifies the second element selected to touch the first.

Join You may join two surfaces or splines with this option. The two selected elements are deleted and the joined element is created.

Extend *Extend* allows you to extend or trim the length of a surface or spline. A surface can be extended or trimmed in multiple directions. Either end of a spline can be extended or trimmed.

NOTE: If you use the *curvature continuity* or *full* options, extending a spline is useful only if the distance to be extended is relatively small compared to the distance between the last two

nodes of the spline. If the *Tangent* option is used, then the spline can be extended to any length without problems.

Text/Dim This option displays the *Edit Text Menu* which allows you to make changes in the text and dimensions contained in your drawing.

Cntl_pts This option allows you to modify surfaces or splines by changing the location of their control points.

Gap Tolerance

Edit / Spline / Node/Tvect Menu

Choosing Node/Tvect from the menu will result in another menu that gives control over the node and tangent vector information for splines.

Analyze This option may be used to display the location and tangent vector information for a spline node.

Move Node This option allows you to move a node to another location.

Tanvect This allows you to change the direction of a tangent vector by indicating a new location for the tangent vector to point towards.

NOTE: This does not change the magnitude of the tangent vector, only its direction.

NOTE: Unless the direction the spline is running is known, there is a 50% chance that the effect of the editing may not make the spline take the desired shape. This is because splines run in only one direction, from one end to the other. If the spline was running in the direction other than the one assumed, the editing may have changed the direction of the tangent vector by more than 90 degrees, and an unexpected loop or curve may have appeared.

Line Tangt The Line Tangt function creates a line that represents the tangent vector at the spline node selected. The length of the line created is equal to the magnitude of the tangent vector.

It can be seen that the tangent vector at the left end of the spline points along the spline, and the one at the other end points away from the spline. This indicates that the spline has a direction, from the start point, the left end, towards the end point, the right end.

Magnitude This option allows you to change the magnitude of a tangent vector. Changing the magnitude of a tangent vector increases or decreases its effect on the spline.

Copy Tanv Similar to the Copy Mag. option above, the Copy Tanv option allows you to assign the direction of a tangent vector at one spline node to another spline node.

Copy Mag The Copy Magnitude function allows you to give one tangent vector the same magnitude value as another simply by clicking the nodes.

Untrim Yes / No When you select Untrim on the menu, the display on the menu will change from Untrim Yes to Untrim No. While set to Yes, any trimmed spline selected will be temporarily untrimmed. The spline will return to its trimmed state either when this is set back to No, when another spline is selected, or when you exit the spline editing menus.

Edit / Surface Menu

The Surfaces option on the Edit menu allows you to change the attributes of a surface.

- Side* This option allows you to change a surface arrow to the other side of the surface.
- Direction* A surface arrow can be changed using this option so that it points in the other direction. If it is currently pointing along the cross sections, using this option will cause the arrow to point across the cross sections (and vice versa).
- Corner* The location of a surface arrow can be changed so that it causes cutting to start at a different corner of the surface.
- Arrow* This option allows you to toggle the surface arrow to make it visible or invisible simply by selecting the surface(s). Selecting surfaces with an invisible arrow causes the arrow to reappear.
- Decompose* This option is used to untrim the surfaces that make up a composite surface and return them to the state they were in before they were trimmed into the composite surface.
- Twist Vecs* You may change a surfaces twist vectors with this option. When the surface is selected a pop-up is displayed for you to select the desired twist vectors for the surface. The surface is recalculated with the new twist vectors.
- Polygon* This option is used to create a polygon on an existing NURBS or parametric surface.
- Display* This option allows you to select the number of curves to display in the U and V direction. When the value of zero is entered, the surface is displayed with flow splines at the actual surface patch boundaries.
- Align* This option is used when dealing with a large number of surfaces to be planar cut whose arrows may not be on the desired surface side. During a planar cut, all arrows should be pointing to the side of the surfaces to be machined.
- Edit Surface Align allows you to specify either a location or a vector (direction) that is used as a reference for setting the surface side by aligning the arrows on one or more surfaces automatically.
- Untrim* This option allows you to untrim a previously trimmed surface. A single surface may also be untrimmed from an existing composite surface with this option.

Menu Item 24 (Unused)

Files Menu (Unused)

Files Menu

The Files menu enables you to load information from disk files into SURFCAM, save work into disk files, display the contents of disk directories, and other tasks that involve information stored in disk files. During these tasks, SURFCAM often requires that you enter the name of a file.

<i>Get</i>	The Get function on the Files menu enables you to load information from a SURFCAM DSN file into SURFCAM.
<i>Append</i>	The Append function on the Files menu enables you to load information from a SURFCAM DSN file into SURFCAM without erasing the existing data.
<i>Save</i>	The Save function allows you to save the current DSN file. You may use the existing name of the file or change the file name.
<i>CADL</i>	The CADL function on the Files menu allows you to read or write information to or from a CADKEY Advanced Design Language (CDL) file format.
<i>DXF</i>	The DXF function on the Files menu enables you to read or write information from or to a Data Exchange Format (DXF) file format.
<i>Plot</i>	The Plot option allows you to create a 2D DXF file from a 3D drawing or output a variety of Pen plotter files.
<i>Inc</i>	Choosing INC from the Files menu presents another menu of options that can be used for backplotting and merging INC (tool motion) files.
<i>Digitized</i>	The Digitized function on the Files menu enables you to read in three different types of files.
<i>STL</i>	This option allows creation of STL type files. These files are used to input into a rapid prototype system. A closed surface model must be created to produce these files. For example all six sides of a cube must be surfaced prior to creating an STL file. Both binary and ASCII STL files are supported.
<i>New</i>	The New option allows you to clear the screen of all elements and begin a new drawing. The layers and views are also cleared. The pop-up values are not returned to the default values.

File Plot Menu (TekSOFT)

File / Plot

This menu allows you to create a Flat DXF or Plotter file.

Flat DXF This option allows you to create a Flat DXF file of the desired entities.

Plot You may create a pen plotter file using this option. The output file type is determined by the Plotter parameter set in the SURFCAM.CFG file. The Plot file created is written to the PLTPath directory specified in the SURFCAM.CFG file. The output file uses the PLT extension for the default. The plotter file is oriented for landscape printing.

Plot Select Menu

This menu allows you to select the desired entities to plot.

Select This option allows you to select specific entities by displaying the Select menu to choose the desired entities.

On Screen This option selects all entities displayed on the current screen.

Done This option is selected when all entities have been selected.

Location Menu

The Location menu is used by a variety of Main menu selections. This menu allows you to select specific locations in 3D space. When this menu is displayed the last selection method used is active. The selection method flashes as described above. You must place the crosshairs close enough to the location for SURFCAM to recognize the location. When the crosshairs are not close enough, or the desired element is masked, the error message Eligible element not found is displayed.

- | | |
|------------------|---|
| <i>Sketch</i> | You may select any point on the screen by clicking the left mouse button. The point is located at the depth on the specified in the current construction view. |
| <i>Point</i> | When this option is selected SURFCAM only allows you to select an existing point. |
| <i>Endpoint</i> | This option allows you to select the endpoint of any element type. SURFCAM selects the endpoint of the element closest to the location clicked by you. |
| <i>Center</i> | You may select the center of an arc, circle, or line with this option. |
| <i>Intersect</i> | You may select a location at the intersection of two elements. You are prompted to select each element. It is not necessary for the two elements to physically or graphically intersect. When the two elements are located on different planes, the intersection is projected based on the current CView. |
| <i>Relative</i> | This option allows you to define a location based on a known location. When this option is selected, you are prompted to enter the X, Y, and Z distance from the known location. This distance is based on the World or View coordinate system as set in the Status window. You are then prompted to select for the known location. |
| <i>Keyboard</i> | When this option is selected, a pop-up is displayed. You may enter any X, Y, and Z coordinate in 3D space. This location is based on World or View coordinates as set in the Status window. |
| <i>Quadrant</i> | This option allows you to select one of four points on a circle. These points are 90 degrees apart starting at the 3 o'clock position. To select one of these four points, click on any point of the circle that is within 45 degrees of the point you want. You can also select any of the same four points associated with an arc if they are within 45 degrees of an end point of the arc. |
| <i>Undo</i> | This option becomes available after an element is created. Selecting <i>Undo</i> deletes the last element created. |
| <i>Done</i> | Choose this option after you have finished selection locations. |

Location Menu

The Location menu is used by a variety of Main menu selections. This menu allows you to select specific locations in 3D space. When this menu is displayed the last selection method used is active. The selection method flashes as described above. You must place the crosshairs close enough to the location for SURFCAM to recognize the location. When the crosshairs are not close enough, or the desired element is masked, the error message Eligible element not found is displayed.

- | | |
|------------------|---|
| <i>Sketch</i> | You may select any point on the screen by clicking the left mouse button. The point is located at the depth on the specified in the current construction view. |
| <i>Point</i> | When this option is selected SURFCAM only allows you to select an existing point. |
| <i>Endpoint</i> | This option allows you to select the endpoint of any element type. SURFCAM selects the endpoint of the element closest to the location clicked by you. |
| <i>Center</i> | You may select the center of an arc, circle, or line with this option. |
| <i>Intersect</i> | You may select a location at the intersection of two elements. You are prompted to select each element. It is not necessary for the two elements to physically or graphically intersect. When the two elements are located on different planes, the intersection is projected based on the current CView. |
| <i>Keyboard</i> | When this option is selected, a pop-up is displayed. You may enter any X, Y, and Z coordinate in 3D space. This location is based on World or View coordinates as set in the Status window. |
| <i>Quadrant</i> | This option allows you to select one of four points on a circle. These points are 90 degrees apart starting at the 3 o'clock position. To select one of these four points, click on any point of the circle that is within 45 degrees of the point you want. You can also select any of the same four points associated with an arc if they are within 45 degrees of an end point of the arc. |
| <i>Undo</i> | This option becomes available after an element is created. Selecting <i>Undo</i> deletes the last element created. |
| <i>Done</i> | Choose this option after you have finished selection locations. |

Layers Menu

SURFCAM allows you the ability to organize the drawing by placing entities on different layers. These layers can be made visible, invisible, selectable, and unselectable.

SURFCAM supports the use of 256 layers numbered 0 - 255. You define the layer name and number using the procedures described below.

Make The Make function on the Layers menu enables you to create a new layer. When a layer is made, it can be used to hold elements that are created. Elements that have already been created can be changed from their current layer into the one created by using the Change option on the Layers menu.

NOTE: SURFCAM support 256 layers numbered 0 - 255.

Current The Current function on the Layers menu allows you to make a specific layer active for storing newly created elements. This layer becomes the current layer. The layer number is displayed in the Status window.

Set The Set function on the Layers menu enables you to change the settings that determine whether a layer is visible (V) or invisible (I) and whether the elements on this layer will be selectable (+) or not selectable (-) during upcoming tasks. The names of layers can also be changed.

Change The Change function on the Layers menu enables you to move elements from one layer to another by selecting them.

NOTE: If the layer chosen in step 2 is set to invisible, the elements will disappear as they are placed into the selected layer. They are displayed when the layer is set to visible.

Layers / Set Layers Menu

The Set function on the Layers menu enables you to change the settings that determine whether a layer is visible (V) or invisible (I) and whether the elements on this layer will be selectable (+) or not selectable (-) during upcoming tasks. The names of layers can also be changed.

- Visible or Invisible* You are prompted for a Layer Range. All layers that fall within the range specified will be set to Visible or Invisible, respectively.
- or -* You are prompted for a Layer range. All layers specified will be set to (selectable) or - (not selectable) respectively. When a layer is set to not selectable (-) the entities on that layer may not be selected. You may not select entities on that layer for any function that requires you to select an entity.
- Swap V & I* Choosing this causes all layers currently visible to be set to invisible, and vice versa.
- Swap & -* Like the choice above, all layers set to (selectable) will be changed to - (not selectable) and vice-versa.
- Done* Select this option when finished making changes to the layer table.

In addition to the menu above, SURFCAM also displays the Layers pop-up with a table listing all defined layers. This table allows you to manipulate the parameters for each layer quickly.

Use the layer table as follows.

- Page Display** To display the previous or next page of the table, you may click the mouse in the drawing window above or below the pop-up, respectively, or use the PgUp or PgDn keys, or the up or down arrow keys
- Column 1** This column displays the letter V or I for visible or invisible. When you click the letter displayed it is toggled between V and I.
- Column 2** This column displays the or - symbol to indicate the selectability of the layer. You may click the symbol for the layer to toggle between the options. This method is used to select individual layer selectability.
- Column 3** This column indicates the layer number. When you click this number the layer becomes the current layer. The Status window displays this layer number as the current layer.
- Column 4** This column displays the layer name. You may change the layer name by clicking this layer column. SURFCAM displays a pop-up with the layer name. You may edit the name and accept the change. The new name is entered into the layer column. The length of the layer name is restricted to the length of the pop-up.

Delete Menu

The Delete menu enables you to delete elements using the Delete selection menu. You may also delete views and layers. In addition to deleting, you may undelete (restore) the last element or all previous elements deleted. Masking may be used to ensure desired entities are not deleted.

<i>Single</i>	Deletes the single entity selected.
<i>Within</i>	Deletes all entities inside the rubber-band box.
<i>Intersect</i>	Deletes all entities intersected and inside the rubber-band box.
<i>Visible</i>	Deletes everything that appears in the work space.
<i>Views</i>	You may delete views from the view list using this option.
<i>Layers</i>	Layers may be deleted from the layers list with this option.
<i>Styles</i>	Styles may be deleted from the <i>Styles list</i> using this option.
<i>Undel Last</i>	The Undelete Last function on the Delete menu enables you to restore elements that have been deleted. When this option is selected, the most recently deleted element is restored.

This procedure may be repeated until all elements have been restored. This option is only valid for items that are deleted during the current SURFCAM session, or since a new file is loaded. When SURFCAM is exited, these items may not be undeleted.

NOTE: When the layer with the element is set to Invisible, the element will NOT appear on the screen until the layer is set to Visible.

NOTE: When layers have been deleted with elements in them, all deleted layers will be undeleted before SURFCAM undeletes any of the elements that were on the deleted layers.

<i>Undel All</i>	The Undelete All function on the Delete menu enables you to restore all elements that were previously deleted during the current SURFCAM session.
------------------	---

NOTE: When the layer with the element is set to Invisible, the element will NOT appear on the screen until the layer is set to Visible.

Options Menu

The options presented to you from this menu allows access to system parameters and settings. The defaults can be changed by editing the SURFCAM.CFG file.

- Display* Choosing Display from the Options menu displays a pop-up that allows changes to various parameters that affect the graphics screen.
- Tolerances* Selecting Tolerances from the Options menu allows you to change various tolerance values.
- Inches / Metric* Selecting this option from the menu toggles the unit of measure used during the SURFCAM session (Inches or Millimeters). No conversion of part geometry is carried out by SURFCAM, this effectively multiplies the tolerance values by a conversion factor to suit the tolerance values to the unit of measure that were selected. The default units can be set with the Units parameter in the SURFCAM.CFG file.
- Mill Axis / Lathe Axis / Lathe Rad / Lathe Dia*
This option allows you to select the *Mill, Lathe Radius, or Lathe Diameter* axes gnomes designations. Clicking on this button repeatedly changes from one to another of these options. The default may be changed by editing the SURFCAM.CFG file. This option is only available with the SURFCAM Lathe option.
- Password* This parameter is used to update the SURFCAM system.
- Create* Selecting Create displays a pop-up allowing you to enter options that apply to both splines and surfaces.

Planar Cut Side

Click the screen on the side of the direction locations desired for the cut.

Post Menu

The Post menu is specific for each NC Mode selected. The default selection at the beginning of the menu will list the available post processors in the POSTFORM file. The options listed in the same group are the individual post processors available in the Postform file. The next group displayed are the defaults for all NC Modes. These are described below:

Verify	This option allows you to graphically simulate the material removal process. See SURFCAM Verify chapter of the manual.
INC Merge	This option allows you to merge desired INC files.
INC Trans	This option allows you to translate an INC file.
APTCL	You may create an APTCL format file using this utility.
SetupSheet	You may create a work Setup Sheet with this option.

NOTE: The last five of the above are described in the SURFCAM Utilities chapter of the manual

These menus may be user-configured by changing the SURFCAM.PST file. This function is described in the **Post Processor** chapter of the manual.

2 axis operations may be processed for a variety of CNC machines. SURFCAM uses the MPOSTWIN.EXE file to generate NC code. Several standard POSTFORM files are supplied that may be added to the POSTFORM.M file. These files may be used as examples to create a user defined machine tool.

SURFCAM uses the INC files with the post processor to generate NC code for CNC machines. You may define the code format with the use of the POSTFORM.M file.

The SURFCAM 2 Axis system may be used for 4 axis wire EDM machines. To create NC code for these machines, use the EPOSTWIN.EXE file. The EPOSTWIN.EXE file requires the use of the POSTFORM.E file.

NC 3 Axis Menu

NC 5 Axis Menu

SURFCAM allows you to perform a variety of machining operations including normal to surface, lead angle, lag angle, swarf, contour, and project cutting. These features can be performed on parametric and NURB surfaces.

To activate the 5 axis capabilities, select *5 Axis* from the *NC Mode* menu. This allows you to choose the cutting options listed below.

<i>Cut</i>	This option allows you to perform 4 axis machine cutting.
<i>Project</i>	You can project an existing INC file onto a surface.
<i>Swarf</i>	This option allows you to cut a surface with the side of the tool.
<i>Contour</i>	You can perform a rotary cut along a chain of elements.
<i>Drill</i>	This option is used for Z axis canned cycles including drilling, reaming, tapping, and boring.
<i>Set Axis</i>	This option is used when one or more of the axes on the machine is rotary. It changes the tool path rapid movements from linear to cylindrical movements. The feature will not be available unless the <i>Rot 5 Axis</i> option in the SURFCAM.CFG file is set to indicate a rotary axis is being used. See the Configuration Utility chapter of the manual for information on how to make this setting.
<i>Options</i>	The 5 Axis cutting parameters may be set using this option.
<i>Edit</i>	The tool path may be edited using this option.
<i>Post</i>	This option displays the current Post processor menu.
<i>NC mode</i>	This option is used to select the desired NC menu.

To perform 5 axis cutting, SURFCAM calculates I, J, and K vectors to direct the tool shank in relation to the tool tip contact point. These vectors identify the tool shank orientation during the cutting process. The tool shank vectors may be adjusted by you with a lead angle and side angle inputs.

The cutting procedures described in this section are all performed in the top view. Machining in different views may be accomplished with construction views.

Create / Spline Menu

A spline is a single element that joins a series of points or locations.

<i>Points</i>	This option allows you to create a natural cubic spline by indicating the locations of the node points. When SURFCAM is used to create a spline in this manner, the resulting spline is called a natural spline which means that the curvature is zero at the ends of the spline.
<i>Elements</i>	Using this option, you can create a spline from a series of chained lines, arcs, and even other splines.
<i>Optimized</i>	This option creates an optimized spline by duplicating an existing spline, then removing the unneeded nodes from it. The new spline created will only use the nodes that are needed to hold a tolerance, specified by you, to the original spline, reducing the amount of data and time that is required to represent curves and surfaces.
<i>Ellipse</i>	An elliptical spline may be created using this option.
<i>Spri hlx</i>	This option allows you to create a spiral spline, helical spline, or helical spiral spline.
<i>Surf. flow</i>	When a surface is created SURFCAM creates surface flow splines. These create the boundaries of the surface. This option allows you to create additional surface flow splines in either direction along the surface.
<i>X-section</i>	The X-section (cross-section) spline is created by slicing surfaces (single, multiple, or composite) at defined intervals relative to the CView. The plane of the sections is perpendicular to the CView. When the CView is set to zero, the current view is used.
<i>Surf. Int</i>	This option allows you to create a spline along the intersection of two surfaces.
<i>Cutter Int</i>	A Cutter Int spline is offset from the intersection of two surfaces such that it defines the center of a cutter.
<i>Project</i>	This option allows you to project a spline onto a surface. The spline must be located in space relative to the location of the surface. The projections is accomplished either normal to the surface or through the construction view.
<i>Blend</i>	This option is used to create a smooth NURBS spline between two existing curves.

Create Menu

This menu allows you to draw the desired entities for the required part geometry. The entities listed below may be selected from the *Create* menu.

<i>Point</i>	You may create a point in the drawing using the <i>Location</i> menu. This menu is displayed when this option is selected.
<i>Points</i>	This option allows you to create multiple points. When selected the <i>Points</i> menu is displayed.
<i>Line</i>	The <i>Line</i> menu is displayed when this option is selected. You are prompted to select the type of line to create.
<i>Arc</i>	You may select this option to create arcs. The <i>Arc</i> menu is displayed for you to select the arc creation method.
<i>Circle</i>	This option is used to create circles. The <i>Circle</i> menu is displayed for you to select the circle creation method.
<i>Fillet</i>	The <i>Fillet</i> option is used to produce fillet radii between two elements or locations. The <i>Fillet</i> menu is displayed for you to create the fillet radius.
<i>Chamfer</i>	You may create a chamfer between two elements or locations with this option. The <i>Chamfer</i> menu is displayed for you to create the desired Chamfer.
<i>Spline</i>	You may select this option to create parametric or NURB splines. The <i>Spline</i> menu is displayed for you to select the desired spline type.
<i>Surface</i>	This option is used to select the desired surface type to create. The <i>Surface</i> menu is displayed for you to select the desired surface creation method and options.
<i>View</i>	You may use this option to create, change, or list views. The <i>View</i> menu is displayed for you to select the desired option or view creation method.
<i>Text/Dim</i>	This option provides access to a number of features to create text or dimensions. The <i>Text/Dim</i> menu is displayed for you to select the desired text or dimension creation method.
<i>Vector</i>	This option is used to define the offset side when creating a tool path in 2D.

Create / Surface Menu (Unused)

Create / Surface Menu

In SURFCAM, a surface is a mathematical representation of a part's outer or inner shape. Surfaces have a positive and a negative side to them, typically indicated by a surface arrow. The direction of the surface arrow indicates the side, and the cutter path direction of the surface to be machined. A surface is displayed using U and V splines. The area bounded by the U and V splines is a surface patch, and the splines are the patch boundaries.

<i>Points</i>	A surface created directly from point data or a linemesh into a surface.
<i>Cross sect</i>	A surface created by selecting a series of cross sections in 3D space that describe the shape of the desired surface.
<i>Drive curv</i>	A surface created by sweeping one or more 2D or 3D cross sections along a curve. Supports an optional twist-directing curve for additional control.
<i>Offset</i>	A surface created by offsetting normal to an existing surface.
<i>Fillet</i>	A surface that is a constant or variable radius fillet between two existing surfaces.
<i>Composite</i>	Trims and links together existing surfaces into a single composite surface.
<i>Primitives</i>	This option provides a set of different basic surface features such as a cylinder, plane, sphere, etc.
<i>Revolution</i>	A surface created by rotating a profile around an axis or vector.
<i>Extrude</i>	A surface created by extending a profile along a defined direction.
<i>Blend</i>	Creates a smooth, tangent surface between 2 - 4 curves or surfaces.
<i>Trimplane</i>	A flat surface that is automatically trimmed to the edge of a surface or curve.
<i>Options</i>	A pop-up window containing surface creation tolerances and options.

Main Menu

When the SURFCAM system is executed the *Main* menu is displayed. The *Main* menu allows you to select from the available system functions.

<i>Create</i>	The <i>Create</i> menu allows you to create many types of elements. Each choice on the <i>Create</i> menu matches a different type of element that SURFCAM can create, and each will cause a submenu to be displayed that lists the different ways that each element may be created.
<i>Edit</i>	The <i>Edit</i> menu allows you to edit, or change, the attributes of elements. You may also edit an INC file created during any machining sequence.
<i>Display</i>	The <i>Display</i> menu enables you to manipulate the image. It can be accessed from the <i>Main</i> menu, or by pressing the desired icon .
<i>Transform</i>	The <i>Transform</i> menu allows you to perform a variety of transformation functions on elements in order to move or copy them to another location or view.
<i>Delete</i>	The <i>Delete</i> menu enables you to delete elements using the <i>Delete</i> selection menu. You may also delete views and layers. In addition to deleting, you may undelete (restore) the last element or all previous elements deleted.
<i>Files</i>	The <i>Files</i> menu enables you to load information from disk files into SURFCAM, save work into disk files, display the contents of disk directories, and other tasks that involve information stored in disk files. During these tasks, SURFCAM often requires you to enter the name of a file.
<i>Layers</i>	SURFCAM allows you the ability to organize the drawing by placing entities on different <i>Layers</i> . These layers can be made visible, invisible, selectable, and unselectable. SURFCAM supports the use of 256 layers numbered 0 - 255.
<i>NC</i>	The <i>NC</i> function allows you to access the SURFCAM machining functions. When this option is selected SURFCAM displays the current NC machining mode menu. You may change the desired machining mode by selecting the <i>NC mode</i> option from the NC menu displayed.
<i>Analyze</i>	The <i>Analyze</i> menu enables you to display physical information about elements, locations and distances on the screen.
<i>Options</i>	The options presented to you from this menu allows access to system parameters and settings. The default can be changed by editing the SURFCAM.CFG file.
<i>Exit</i>	This option exits the current session.

Create / Surface / 4 Curve / Blending Menu

This option is used to create a surface by chaining together four boundary curves that describe the edges of the surface to be created.

In this type of surface construction method, there are two generator curves and two director curves. The two generator curves are the first two boundary curves selected. The director curves are the opposite two boundary curves.

When the corresponding curves have the same number of elements, you can synchronize the elements in the corresponding curves. This has the same effect as it does in the other surface types; the patch boundaries within the surface follow a spline between the corresponding break points.

Curve geometry

The geometry created for a 4 Curve surface may contain any combination of entities. The elements should be created in 3D space. These entities should have first order continuity. The elements of each curve should be tangent. All boundary curves that are used for this type of surface creation must connect at their ends.

Chaining Direction

The order that the curves are selected and the direction in which they are chained is critical to the success of the surface being built. The 4 curves must be chained in groups as if chaining cross sections. The first two curves selected are the Generator curves. They are selected in the same order as selecting Cross Section surface elements. The second two curves are the Director curves. These are also selected like the Cross Section surface; however, the First endpoint selected of the Director curves must be the first endpoint selected of the Generator curves.

NOTE: Each of the four boundary curves is chained separately. Since the boundary curves connect at their end points, the Close option should not be used when chaining each side.

When this option is selected a menu is displayed for you to select the desired blend type. These blend types allow you to select different forms of Parametric Coons Patch surface construction.

These options determine the type of blend between the 4 curves. The amount of surface variation is dependent upon the curves.

- | | |
|-----------------|---|
| <i>Linear</i> | This option produces a surface with a linear transition between the Generator and Director curves. |
| <i>Bi-cubic</i> | This option allows you to create a surface with a splined, Cubic, transition between the curves. This splined transition creates a surface with more curvature than the Linear blend. |
| <i>Flat Tan</i> | This blend type creates a surface with the tangent vectors of the curve group tangent to the edge splines of the opposite group. The tangent vectors of the Generator curves are tangent to the Director curve edge splines. The Director curves tangent vectors are tangent to the Generator curve edge splines. This blend also uses a splined transition between the curves. |

Create / View Menu

The view option allows you to define a specific orientation as a view. This makes it possible to quickly return to any view. It also allows you to create geometry and machine in a specific orientation.

When creating a view, SURFCAM prompts you to define the view number. This number may then be entered into the view parameter on the status menu to return to the specified orientation.

NOTE: The first 8 views are predefined, when creating new views, do not use numbers less than 9.

Because a viewing position in SURFCAM is normal (perpendicular) to a viewing plane, a view can be defined by describing the plane that is to be viewed. The options on the Create View menu, displayed below, allow different methods of creating a view.

- | | |
|-------------------|---|
| <i>3 Points</i> | This option is used to create a view by selecting three points. The first two points define the X axis of the view. The third point defines the direction of the view Y axis. |
| <i>Point/Line</i> | You may create a view by using an existing line and a desired point. The line defines the view X axis. The point defines the direction of the view Y axis. |
| <i>Normal</i> | The normal selection allows you to create a view using the X and Z axis. The view is created with the same method as 3 points. The difference is the axis orientation. The first two points specify the direction along the Z axis, and the third point orients the X axis. As with the 3 points option, SURFCAM prompts you for the location of the origin and a view name and number. |
| <i>Current</i> | This option is used to record the current screen orientation as a view. |
| <i>List views</i> | This option lists the current view numbers and names. You may edit the view names by clicking the mouse while it is on a view name, the right side of the box. A pop-up is displayed for you to enter or edit the view name selected. |

Edit / View Menu

This option lists the current view numbers and names. You may edit the view names by clicking the mouse while it is on a view name, the right side of the box. A pop-up is displayed for you to edit the view name selected.

Transform Menu

The Transform menu allows you to perform a variety of transformation functions on elements in order to move or copy them to another location or view. The menu below is displayed when this option is selected.

The Transform menu allows you to either Move or Copy entities for each of the options. You must select the desired function before selecting the desired option. The Move or Copy function flashes a different color to identify it as active.

<i>Distance</i>	The Trans Dist. function on the Transform menu enables you to copy or move one or more elements by specifying the X, Y, and Z distances to translate (move) the elements from their current position.
<i>Location</i>	The Trans Loc. function on the Transform menu enables you to copy or move one or more elements to another location by selecting a base location, then specifying the new location for that point. The selected elements are then moved or copied to the new location by translating (moving) them the absolute distance between the base location and the new location.
<i>Rotate</i>	The Rotate function on the Transform menu allows you to copy or move elements by rotating them about a specific location. The function requires you to specify the point of rotation and an angle of rotation.
<i>Scale</i>	The Scale function on the Transform menu enables you to copy or move elements by applying a scaling (shrinkage or enlargement) factor. This allows you to enlarge or shrink an element.
<i>Scale XYZ</i>	This option allows you to scale all geometry independently along the three axes. The Scale XYZ option allows you to enter different scaling factors for each axis. Negative scale factors can also be used.
<i>Mirror</i>	The Mirror function on the Transform menu enables you to copy or move element(s) in a mirror image fashion.
<i>To View</i>	The To View function on the Transform menu enables you to copy or move and rotate one or more elements from one view to another.
<i>Offset</i>	The Offset function on the Transform menu allows you to offset lines and arcs by a distance, to a location, or tangent to another entity.
<i>Copy</i>	This function sets SURFCAM into the copy mode. Transformations performed while in this mode will cause copies to be created of the element(s) being transformed.
<i>Move</i>	This function sets SURFCAM into the move mode. Transformations performed while in this mode will NOT create copies of the element(s) being transformed. The elements selected will moved to the new location.

Offset Menu

Distance This option allows you to offset an arc or line by the distance displayed on the prompt line. This distance is defined by using the Change off option. The default is set at .5 inches. The distance is the amount of change in the arc radius. Select the arc to offset and clicks the side of the selected arc for the new arc.

NOTE: When an inside arc is to be created, the distance should be less than the original arc radius.

Location The Location menu is displayed for you to select a location for the offset arc or line to intersect. The distance between the location selected and the center of the arc defines the radius of the offset arc.

Tangent This option allows you to create an offset arc tangent to a line or arc. You are prompted to select the arc to offset. You are then prompted to select the arc or line to which the arc is to be tangent. When the offset element is created the menu displays the Other option to select the different tangencies. A line can be offset tangent to an existing arc or circle.

Change off This option allows you to change the value used for Offset Distance. A pop-up is displayed for you to enter the desired distance value. This value is displayed in the SURFCAM prompt line at the top of the screen.

Create / Splines / NCurves

Create / NSurfaces Menu

Edit / NCurves Menu

Pick Menu

Select the nodes to be moved and click *Done* when complete. The *Location* menu is displayed for you to select the location to move from and to.

Edit / Nsurfaces Menu

NC Lathe Menu

SURFCAM may be used to perform various NC turning operations to include ID and OD Turning, Facing, Grooving, Threading, and Drilling. These operations are performed on 2D geometry. SURFCAM's standard lathe capabilities are listed below.

<i>Turn</i>	This option allows ID or OD machining.
<i>Face</i>	This option allows you to machining the part end.
<i>Groove</i>	You may machine a groove with this option.
<i>Thread</i>	Threading may be accomplished with this option.
<i>Drill</i>	Drilling is performed using this option.
<i>Options</i>	The Lathe Axis cutting parameters may be set using this option.
<i>Edit</i>	The tool path may be edited using this option.
<i>Post</i>	This option displays the current Post processor menu.
<i>NC mode</i>	This option is used to select the desired NC menu.

These operations may be performed on two axis lathes only. SURFCAM is compatible with any lathe axes orientation. The axes orientation is user configurable with SURFCAM and the post processor. Standard output for a variety of manufacturers may be generated with the sample post processors supplied. The Post processor section explains the procedure for configuring the different post processors. For display purposes, all reference to the tool orientation is based upon the turret being to the rear of the part. Front lathe machine code is produced by the post processor.

Inside and outside diameter turning operations may be performed with the operations listed above. The Drill operation can be used for off center-line drilling also.

NOTE: The actual axis orientation and direction may be overridden by the post processor.

Yes or No

Read the question in the pop-up and select *Yes* or *No*.

Tool Information

Tool Corner Radius

Specifies the tip radius of a tool. In the case of a ball end mill, this is the tool radius; for a bullnose cutter, this is the corner radius.

Tool Total Radius

Specifies the larger radius of a bullnose cutter. If the cutter total radius is greater than the tip radius, SURFCAM uses a bullnose cutter.

Offset Side

Specifies the three different types of tool motion listed below:

Left Causes a clockwise cutting tool to climb cut the contour with standard clockwise cutting tools. SURFCAM drives the tool along the left side of the contour.

Right Exactly opposite to the above choice, SURFCAM drives the tool along the right side of the contour. This results in a conventional cutting process if using standard clockwise cutting tools.

NOTE: SURFCAM allows you to machine either side of the contour. The side of the contour that gets machined depends on the relationship between the chaining direction and the type of cut parameter. SURFCAM always machines contours in the direction that they are chained, and that, in conjunction with the type of cut parameter, determines which side of the contour is machined. If you choose left to climb cut a pocket, the pocket must be chained in a counterclockwise direction, and any islands inside the pocket must be chained in a clockwise direction. Conversely, if you choose Right to conventional cut a pocket, the pocket must be chained in a clockwise direction, and islands inside the pocket must be chained in a counterclockwise direction. Chaining in the wrong direction will cause the tool to cut on the wrong side of the contour or islands.

On Contour This is used to drive the center line of the tool directly along the path of the chained elements, as would be done to cut a groove or slot.

NOTE: During on contour operations, gouge checking is not active. This allows you to perform engraving operations.

Curve tolerance

SURFCAM will break all contour moves along a spline into straight linear movements. When driving the tool around a spline curve, SURFCAM will adjust the length of the cutting moves to make sure that the edge of the contour is machined within this tolerance value.

On Sides In Z (Title)

Geometry at value

Enter the desired Z value for the geometry location.

Rapid Plane

Specifies an absolute location to move to for clearance above the part surface. This should include clearance above any clamps or obstructions.

Minimum Z Value

This parameter sets the most negative Z axis move allowed. The tool path will not move below this level. The tool path maintains this Z level along the contour until the end of the contour or a higher Z level is intersected.

Path

Angle

Enter the desired angle of the line.

Length

Enter the desired length of the line.

Line Information

Radius (Diameter)

Enter the desired radius (diameter) of the arc.

Start Angle

Enter the desired start angle for the arc.

End Angle

Enter the desired ending angle of the arc.

Arc Information

Max Deviation Error To Spline

Enter the maximum desired deviation error. A string of lines drawn between the points will not deviate from the original spline more than the error amount specified.

Distance

Enter the desired distance between the created points.

Enter New Fillet Radius

Enter the desired radius of the fillet arc.

Tip or Center

Normally, the NC code produced will drive the *Tip* of the cutter. When using a ball end mill and the tool path for the ball center is desired, choose *Center*.

Tool Containment

The choice here specifies the way in which you can contain or trim the resulting tool paths.

- None* The tool will cut the entire surface without trying to contain it to any defined spline or patch boundaries. When the surface has been trimmed (it is part of a composite surface), only the visible portion will be machined. To machine the complete surface, or any portion that has been trimmed away, choose the Spline option (described below), and select the boundary curves for the tool to observe. (To machine the whole surface, choose Spline. When prompted to select the boundary curves, choose Done from the menu without chaining any boundary curves.)
- Patch* This allows you to define an orthogonal patch of a surface to be cut (which means: rectangular in parametric space) by picking two locations on or near the surface that determine the corners of an orthogonal region on the surface. The area of the surface that is orthogonal between these points will be cut (parallel to the existing surface patch boundaries). You are prompted to enter two locations after all pop-ups have been answered for the Cut routine.
- Spline* Supports complex tool containment by selecting Surface Intersect Splines, Cutter Intersect Splines, Projected Splines, or Fillet Edge Splines or any combination as cutting boundaries.
- Both* Allows you to contain the tool using a combination of Patch and Spline boundaries.

Comment

Enter the desired comment for the INC record.

Inline

Enter the desired Inline text for the INC record.

Scallop/Increment

<i>Scallop</i>	Choosing <i>Scallop</i> tells SURFCAM to calculate the number of passes according to the scallop height specified.
<i>Increment</i>	Choosing <i>Increment</i> allows you to enter the distance between cuts measured along the surface.

Scallop value

If *Scallop* is chosen for *Step type*, this value specifies the maximum scallop size. If *Increment* is chosen for *Step type*, this value specifies the distance between cutting passes (as measured along the surface).

Surface tolerance

This value is the maximum error allowed in the direction of the cutter. SURFCAM will create as many linear moves as needed to hold the cutting to within this tolerance.

Retrace Type

The choice here specifies the way that the tool will move between successive cutting passes.

- Relative plane* At the end of each cutting pass, the tool will rapid up in Z by the Relative Plane Clearance value (entered in the NC Options menu) from the highest point in the preceding cut (or the start point of the next cut, whichever is higher) to rapid back to the start of the next cut.
- Top Plane* At the end of each cutting pass, the tool will rapid up in Z to the rapid plane, position to the next pass, rapid down to the rapid plunge clearance, then feed down to the start of the pass.
- Bi-directional* The tool will cut alternating passes in opposite directions. No rapid moves will occur between passes when the Maximum feed between, in the NC options menu, is set to a value that is greater than the distance from one pass to the next.
- Feed between* The tool will feed directly from the end of one pass to the start point of the next. It will not use the top plane. This option should be used for a closed surface such as a sphere or cylinder. This type will provide the direction of cut as opposed to the Bi-directional option, which would reverse direction. If this option is used on an open surface the system will create tool motion identical to the Top Plane option.

Highest Node Z

This is not a value for you to enter. The value shown here is the highest node on the surface. SURFCAM displays this automatically so that you can more easily set the rapid plane to avoid crashes.

NOTE: It is possible that the surface actually extends higher than the value displayed above. This is caused when the surface patches bulge above the node points.

Leadout Type

There are four type of leadin moves: 3D, 2D, Undercut 3D, and Undercut 2D. When either 3D is selected the move may be along any axis. When either 2D is selected the move is only along the XY axes. When either undercut option is selected the move is made only when there is an undercut possibility.

NOTE: When a leadout move is desired, it is recommended that the tool vector type be used for 3 axis machining. This option allows no rotary motion for the leadout move.

Finish Depth of Cut

This is the amount of material to remove for the finish pass.

Number of Passes

This is the number of finish passes to perform. The Finish depth of cut amount is used for each of these passes.

Spring Passes

Indicates the number of times the final finish pass is cut. The final finish pass may be repeated to remove material that may be left due to part and/or tool deflection.

Minimum Z Value

When you desire to limit the lowest Z value cut, enter that value here. If there is nothing on this line, there will be no limit imposed in the NC code.

Program Information (Title)

Leadin Move Length

Enter the desired length of the leadin move.

Leadin Move Type

There are four type of leadin moves: 3D, 2D, Undercut 3D, and Undercut 2D. When either 3D is selected the move may be along any axis. When either 2D is selected the move is only along the XY axes. When either undercut option is selected the move is made only when there is an undercut possibility.

Rapid Plunge Clearance

This parameter is used to set the distance above the part to end the rapid move. It is also used as the minimum amount that the rapid plane must be above the part.

Relative plane clearance

This clearance plane is defined as the rapid distance above the start point of the next tool pass, or the distance above the highest Z point on the surface. SURFCAM uses this point for rapid moves when relative plane is selected for the cutting method.

Feed between clearance

This is the relative distance to move straight up when finished with a cutting pass before feeding to the start of the next cutting pass. This value is used when the distance between the end of one pass and the start of the next is less than the Maximum feed between size.

Maximum feed between

When the move is smaller than this value the cutter moves to the Feed between clearance before moving to the start of the next pass. If the move is larger than this value the cutter moves to the Rapid plane. The value entered is selected as either a constant value or a multiplier of the cutter radius.

When Constant is selected the value entered is used for the distance of the Feed Between move. When Cutter Radius is selected the value entered is multiplied by the radius of the cutter selected to determine the Feed Between distance.

Program Information

After accepting the Tool Information values, a Program Information pop-up box is displayed to enter information that relates to the material to be cut, and additional information that affects some of the codes in the resulting NC program.

The first line in the pop-up box allows you to select the material library. Press the spacebar or press the right mouse button and the material library pop-up box containing a list of materials is displayed. This is presented for easy review of the materials and to select the desired material. As the mouse is moved horizontally through the columns in the pop-up, the prompt line shows what type of information is in each column. To scroll through the material library use the up or down arrow keys or the Page Up and Page Down keys. The mouse may also be used to page through the library. Place the arrow above (page up) or below (page down) the materials pop-up box. Press the Left mouse button to page through the listing. Highlight the desired material and press the left mouse button to select the material. The material and tool selected affect the calculations that SURFCAM performs for the spindle speed and feed rates. See APPENDIX B for an explanation of the SURFCAM.MAT and SURFCAM.TOL parameters and information on how to properly edit them.

Program Number

Specifies the program number that the NC machine will use for the program being generated. On some controls, this is the *O* number at the top of the program.

Sequence Number

Specifies the beginning program sequence number of the NC program.

Sequence Increment

Specifies the increment of the program sequence numbers. For instance, setting this to 10 will result in the N numbers displayed below:

N10 G...

N20 G...

N30 G...

...and so on.

Tool Number

Specifies the tool number to be used for the upcoming cutting cycle. In most NC machines this is the *T* number.

Tool Length Offset

This specifies the length offset number to be used with this tool, not a length measurement.

Tool Diameter Offset

Specifies the diameter offset number, not a diameter measurement, to be used with this tool. On most NC machines a D or H number appears in the code to identify which offset number to activate, not the distance to offset the tool.

Spindle Speed

Specifies the spindle speed in clockwise rpm. If counterclockwise rotation is desired, enter a negative number here. SURFCAM calculates the Speeds and Feeds based on the tool and material selected.

Speeds and Feeds

The following parameters are set or calculated in the SURFCAM.TOL and SURFCAM.MAT files and cannot be changed in the Material pop-up.

Calculated Chip Load Displays the total tool load per flute used for automatic spindle speed and feed calculations after multiplying the current tool chip load by the current material factor.

No. of Flutes Displays the total number of flutes of the current tool. Based on the parameters of the current tool library.

Surface Speed Displays the surface feet per minute of the current material. Based on the parameters of the current material library.

Chip Load Factor Displays the material factor that is multiplied with the current tool's chip load. Based on the parameters of the current material library.

The following formula is used to determine the Feeds and Speeds for programmed tool paths.

$$\text{RPM} = (3.82 / \text{Tool Diameter}) * \text{Surface feet per minute}$$

$$\text{IPM} = \text{RPM} * (\text{number of flutes} * \text{chip load} * \text{chip load factor})$$

NOTE: The chip load factor is defined by you in the SURFCAM.MAT file.

Example: For a 2 flute 1/2" end mill using 500 SFM. The chip load is .004 and the chip load factor is .8

$$\begin{aligned} \text{RPM} &= (3.82 / .5) * 500 \\ &= 7.64 * 500 \\ &= 3820 \end{aligned}$$

$$\begin{aligned} \text{IPM} &= 3820 * (2 * .004 * .8) \\ &= 3820 * .0064 \\ &= 24.45 \end{aligned}$$

NOTE: Spindle speeds are rounded to the nearest whole number. Feed rates are rounded to two decimal places.

Metric calculations use the following formula.

$$\text{RPM} = (318.31 / \text{Tool Diameter}) * \text{Surface meters per minute}$$

Feed Rate

Specifies the XY axis feed rate in inches per minute. SURFCAM calculates the Speeds and Feeds based on the tool and material selected.

Speeds and Feeds

The following parameters are set or calculated in the SURFCAM.TOL and SURFCAM.MAT files and can not be changed in the Material pop-up.

Calculated Chip Load Displays the total tool load per flute used for automatic spindle speed and feed calculations after multiplying the current tool chip load by the current material factor.

No. of Flutes Displays the total number of flutes of the current tool. Based on the parameters of the current tool library.

Surface Speed Displays the surface feet per minute of the current material. Based on the parameters of the current material library.

Chip Load Factor Displays the material factor that is multiplied with the current tool's chip load. Based on the parameters of the current material library.

The following formula is used to determine the Feeds and Speeds for programmed tool paths.

$$\text{RPM} = (3.82 / \text{Tool Diameter}) * \text{Surface feet per minute}$$

$$\text{IPM} = \text{RPM} * (\text{number of flutes} * \text{chip load} * \text{chip load factor})$$

NOTE: The chip load factor is defined by you in the SURFCAM.MAT file.

Example: For a 2 flute 1/2" end mill using 500 SFM. The chip load is .004 and the chip load factor is .8

$$\begin{aligned} \text{RPM} &= (3.82 / .5) * 500 \\ &= 7.64 * 500 \\ &= 3820 \end{aligned}$$

$$\begin{aligned} \text{IPM} &= 3820 * (2 * .004 * .8) \\ &= 3820 * .0064 \\ &= 24.45 \end{aligned}$$

NOTE: Spindle speeds are rounded to the nearest whole number. Feed rates are rounded to two decimal places.

Metric calculations use the following formula.

$$\text{RPM} = (318.31 / \text{Tool Diameter}) * \text{Surface meters per minute}$$

Plunge Rate

Specifies the plunge rate in inches per minute. This is the rate at which the tool will feed if it is moving directly down at the beginning of a cutting cycle, or as it moves between passes in feed and bi-directional retrace modes. SURFCAM calculates the Speeds and Feeds based on the tool and material selected.

Speeds and Feeds

The following parameters are set or calculated in the SURFCAM.TOL and SURFCAM.MAT files and can not be changed in the Material pop-up.

Calculated Chip Load Displays the total tool load per flute used for automatic spindle speed and feed calculations after multiplying the current tool chip load by the current material factor.

No. of Flutes Displays the total number of flutes of the current tool. Based on the parameters of the current tool library.

Surface Speed Displays the surface feet per minute of the current material. Based on the parameters of the current material library.

Chip Load Factor Displays the material factor that is multiplied with the current tool's chip load. Based on the parameters of the current material library.

The following formula is used to determine the Feeds and Speeds for programmed tool paths.

$$\text{RPM} = (3.82 / \text{Tool Diameter}) * \text{Surface feet per minute}$$

$$\text{IPM} = \text{RPM} * (\text{number of flutes} * \text{chip load} * \text{chip load factor})$$

NOTE: The chip load factor is defined by you in the SURFCAM.MAT file.

Example: For a 2 flute 1/2" end mill using 500 SFM. The chip load is .004 and the chip load factor is .8

$$\begin{aligned} \text{RPM} &= (3.82 / .5) * 500 \\ &= 7.64 * 500 \\ &= 3820 \end{aligned}$$

$$\begin{aligned} \text{IPM} &= 3820 * (2 * .004 * .8) \\ &= 3820 * .0064 \\ &= 24.45 \end{aligned}$$

NOTE: Spindle speeds are rounded to the nearest whole number. Feed rates are rounded to two decimal places.

Metric calculations use the following formula.

$$\text{RPM} = (318.31 / \text{Tool Diameter}) * \text{Surface meters per minute}$$

Coolant type

Specify Flood, Mist, or Off coolant control before beginning any cutting cycle.

Leadin Move Length

Specifies the length of the leadin move. Generally, a leadin move is used to prevent the tool from positioning directly above (and therefore gouging on the plunge move) the start point of a cutting pass. This parameter sets the distance from the pass start point to position to before plunging to depth.

Leadout move Length

This parameter defines the length of the leadout moves.

Swarf Type

The swarf type may be selected as either Minimum Tilt (Min Tilt) or Ruled. These options direct the shank orientation during the cut across a surface as described below. The tool shank orientation is maintained by rotary axis rotation.

Min Tilt This option is not applicable for 4 axis machining.

NOTE: This option requires the rotary axis to rotate 180 degrees while moving the linear axes. This may cause the tool to gouge the surface. It is recommended that this option not be used for 4 axis rotary operations. The Ruled type should be used for these operations.

Ruled This causes the shank of the tool to be oriented parallel to the flow lines of the surface being cut with no regard to the true Z.

Side Angle

This allows you to specify an angle for shank clearance. The side angle may also be used for an undercut. It also allows you to adjust for tapered tools during swarf cutting. The tool taper angle is input for Side Angle to maintain the tool shaft parallel to the surface. A larger angle can be used when shank clearance is desired for these tools as well.

NOTE: A positive angle value orients the tool shaft away from the surface.

Stock to Leave

Enter the amount of stock that the upcoming cutting cycle will leave for a finish cut.

Depth

Enter the desired depth for the next entity.

How Many Sections

This defines the number of break points.

Break the Original

You may break the original entity or maintain it as a single element.

Points at Breaks

You may create points at the break locations. These points are created whether the element is broken or not.

Chords Between Breaks

When an arc is incremented, you may create chord lines between the break locations.

Distance to Extend Spline

Enter the distance to extend the spline.

Change Magnitude

Enter the desired magnitude for the tangent vector.

Negative Number Reverses the Vector

X Value

Enter the X axis location.

Y Value

Enter the Y axis location.

Z Value

Enter the Z axis location.

Would You Like to Use an INC File

After selecting the surface to rough cut, you are prompted to use an INC file. When a response of Yes is entered, you are prompted for the INC filename. SURFCAM reads and displays the INC file after the filename is entered. You are then prompted for the output INC filename.

Increment

Enter the Auto rough step over amount.

Display Curves in U Direction

Enter the desired display splines in the U direction.

Project Interval

This value defines the surface accuracy of the Auto rough sequence.

Roughing Information (Title)

Depth Step

This sets the Z axis distance for each roughing pass.

Option Parameter Type

These parameters can be toggled between *Constant* and *Cutter Radius*. Use the *Cutter Radius* for a particular value to be calculated automatically by multiplying the number entered in the first column by the cutter radius. For the value to always remain the same, set the 2nd column to *Constant*. The value entered will be the exact inch, or millimeter, value of that parameter and will not vary with step or tool size.

Shelf Plane (Changed 10-12-95. Cf #1002)

Initial Depth

Specifies the Z axis depth for the first pass.

Final Depth

The value entered for this parameter is the Z axis dimension for the final pass. The roughing operation uses this depth for the last cut. When this value is below the actual bottom of the surface being cut, the cutter path will only go as far as the lowest surface cutter tangent point.

Enter Fillet Radius

Enter the desired radius of the fillet.

Layer Name

Enter the desired name for the Layer.

Layer Number

Enter the desired layer number. This number must be between 0 and 255.

Make Current

This parameter allows you to set the new layer as the current active layer. Select Yes to make the new layer active.

Make New Layer

Set Current Layer

Enter the layer number of the desired current active layer.

Menu Item 147

Range of Layers

Beginning Layer Number

Enter the first layer number.

Ending Layer Number

Enter the last layer number .

Layer

Enter the layer number to which entities are to be moved.

Change Data To

Layer Name

Enter the desired text for the layer name.

Change Layer Name

Layer Number

Delete Layer

Point

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Line

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Poly Line

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Arc

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Curve

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Line Mesh

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Surface

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Tool Path

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Ncurve

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Tool Path

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

All Elements

Clicking this button causes an X to be displayed in the box to the right of each element.

If an X is already displayed in each box, clicking on the button will have no result.

When the box to the right of each element contains an X, each element is selectable.

No Elements

Clicking this button causes a the box to the right of each element to become blank.

If each box is already blank, clicking on the button will have no result.

When the box to the right of each element is blank, each element is masked, i.e., no element is selectable.

Menu Item 169

All Colors

Clicking this button causes a bar to be displayed to the left of each color.

If a bar is already displayed to the left of each color, clicking on the button will have no result.

When a bar is displayed to the left of each color, each color is selectable.

You can also click on a color itself to toggle between a bar and a blank for that particular color.

No Colors

Clicking this button causes a blank to be displayed to the left of each color.

If a blank is already displayed to the left of each color, clicking on the button will have no result.

When a blank is displayed to the left of each color, each color is masked, i.e., no color is selectable.

You can also click on a color itself to toggle between a bar and a blank for that particular color.

All Elements

No Elements

All Colors

No Colors

Menu Item 177

Menu Item 178

Menu Item 179

Menu Item 180

Display Curves in V Direction

Enter the desired display splines in the V direction.

Enter Knot Value to Add

Draw the Tool

This option allows you to select the following *Tool* display options.

<i>No</i>	No cutting tool is displayed.
<i>Yes</i>	The cutting tool defined by the drawing named in the SURFCAM.TOL file is displayed.
<i>2d circles</i>	The cutting tool is not displayed. SURFCAM displays 2D circles, the diameter of the tool, for milling, or the diameter of the nose radius of the insert, for turning, representing the cutter path.
<i>With holder</i>	SURFCAM displays the cutting tool and the tool holder defined in the SURFCAM.TOL file.

If *Yes* or *With holder* is selected, the following keys are activated to control the tool display.

<i>Enter</i>	Stops the tool motion. Press Enter again to restart the tool motion.
<i>Space Bar</i>	Single-steps the tool through its moves after using the Enter key to halt the tool path. Pressing the ENTER key again will cause SURFCAM to resume animating the tool.
<i>0-9</i>	Changes the speed of tool path animation. 0 is maximum tool path animation speed, 9 is minimum.

Tool Color

This is the color number of the animated tool display.

Normal Vector Display

Yes will cause SURFCAM to draw the offset vector lines that show the relation of the center of the tool to the contact point on the surface being cut. This also traces curves on the surface itself showing where the tool nose contacts the surface. These vectors and contact path are NOT stored, and will disappear when the screen is redrawn.

Tool Path Display

You may define when the tool path is displayed with the following options.

- Off* The tool path is not displayed. The path is not saved in the graphics area when this option is selected.
- During cutting* The tool path is drawn while the cutter path is created.
- Upon Redraw* The tool path is only displayed when the graphics screen is redrawn using the Display Redraw function.

The Off and Upon Redraw options may result in a significant decrease in actual processing time.

Arrow Color

This parameter affects the way that the surface arrows are drawn. The surface arrow is composed of two parts; the shaft and the arrow itself. The default values for these are set by the SurfaceArrow parameter in the SURFCAM.CFG file.

Arrow Color is the number of the color that will be used to draw the arrows. If zero is selected, the arrow will be drawn in the same color as the surface to which it is attached.

Shaft Size

This parameter affects the way that the surface arrows are drawn. The surface arrow is composed of two parts; the shaft and the arrow itself. The default values for these are set by the SurfaceArrow parameter in the SURFCAM.CFG file.

Shaft size indicates the length of the shaft.

Head Size

This parameter affects the way that the surface arrows are drawn. The surface arrow is composed of two parts; the shaft and the arrow itself. The default values for these are set by the SurfaceArrow parameter in the SURFCAM.CFG file.

Head size is the length of the arrow.

Time Constants

Time Pop-ups are Displayed

This parameter is a delay time value that measures in seconds. This value determines the length of time that pop-up warning messages appear on the screen.

Rotation Wait

This parameter is a delay time value that measures in seconds. When using the mouse to rotate or pan the image, a delay must be introduced to prevent the screen from being redrawn whenever a mouse pulse is detected. If there is no delay, it is difficult to see how the mouse movement is affecting the image.

Tool Display Options

Chaining Tolerance

Elements whose end points are farther away from each other than this distance will not be chained together.

User Tolerance

SURFCAM will regard elements that do not come within this distance of each other to be not touching. As an example, a point will not be created at a selected intersection unless the actual intersection of the two selected elements is within this tolerance.

View Tolerance

This tolerance is used when comparing angular values. This value is in degrees.

Surface Intersect and Fillets

This value is the maximum deviation when constructing surface intersect splines or a fillet surface. The default is 0.00040 inches.

It defines the chordal accuracy of the curves when created.

Tolerance Options

These are Maximum Errors

INC File In

INC File Out

Project Cut Files

Project Cut Control (Title)

Line Interval

Since movements in the input file are projected onto the surface, long linear moves must be broken into short ones so that the surface will be followed accurately. Linear moves in the input file will be broken into moves of this length. Typical values are between .02 and .15 depending on the tool being used and the surface tolerance required.

Curve Tolerance

SURFCAM will break all contour moves along a spline into straight linear movements. When driving the tool around a spline curve, SURFCAM will adjust the length of the cutting moves to make sure that the edge of the contour is machined within this tolerance value.

Spindle Speed Type

This parameter is based on Inch or Metric selection in the Main Menu options. Select either Inches per Revolution (IPR) / Millimeters per Revolution (MMPR) or Constant Spindle Speed (CSS). The selection determines the Post output code for the Spindle Speed.

Feed Rate Type

This parameter is based on Inch or Metric selection in the Main Menu options. Select either Inches per Minute (IPM) / Millimeters per Minute (MMPM) or Inches per Revolution (IPR) / Millimeters per Revolution (MMPR). The selection determines the Post output code for the Feed Rate.

Plunge Rate Type

This parameter is based on Inch or Metric selection in the Main Menu options. Select either Inches per Minute (IPM) / Millimeters per Minute (MMPM) or Inches per Revolution (IPR) / Millimeters per Revolution (MMPR). The selection determines the Post output code for the Plunge Rate.

Minimum Z Value

Add Z Values

Selecting *Yes* causes the Z values from cutting moves in the input file to be added to the projected Z of the surface. For instance, if the incoming move (from the input INC file) has a Z value of -1.5, then the resultant move will be cut 1.5" below the surface; the tool motion display is offset from the surface accordingly. When *No* is selected, the cutting will not go below the surface.

Increment

This is the distance between cutting passes.

Planar Cut Control

SURFCAM may be used to cut a single surface or across multiple surfaces. The planar cut allows you to select the direction of the cut and the distance between the steps. You may select specific surfaces or all surfaces displayed.

In this method of cutting, multiple surfaces are machined by intersecting them with evenly spaced planes. The resulting curves describe the tool motion over the surfaces.

The information that is entered affects the plane of cutting, the distance between the cuts, and the direction the tool will move in the plane. The plane of cutting is always perpendicular to the CView plane and passes through two specified locations.

Surface Tolerance

This value is the maximum error allowed in the direction of the cutter. SURFCAM will create as many linear moves as needed to hold the cutting to within this tolerance.

Cutting Direction

There are two options to define the direction the tool moves along each cut.

- Planar* This option generates tool motion in a straight direction. Two positions must be provided to define the direction of cut. This could be any two locations. The *Location* menu is displayed to define the coordinates just before the tool motion is generated.
- Flow surface* The flow surface option uses an existing surface to define the direction. When this option is used, the system will prompt you to select a surface to use as the flow surface. This surface will not be machined. The arrow on the flow surface defines which direction to use--either the major or the minor. To switch the direction, use the *Edit/Surfaces/Direction* option.

Retrace

The choice here specifies the way that the tool will move between successive cutting passes.

- Top Plane* At the end of each cutting pass, the tool will rapid up in Z to the rapid plane, position to the next pass, rapid down to the rapid plunge clearance, then feed down to the start of the pass.
- Bi-directional* The tool will cut alternating passes in opposite directions. No rapid moves will occur between passes when the Maximum feed between, in the NC options menu, is set to a value that is greater than the distance from one pass to the next.
- Feed between* The tool will feed directly from the end of one pass to the start point of the next. It will not use the top plane. This option should be used for a closed surface such as a sphere or cylinder. This type will provide the direction of cut as opposed to the B-directional option, which would reverse direction. If this option is used on an open surface the system will create tool motion identical to the Top Plane option.

Change Tool Axis Vector

This option allows you to select when the tool axis vector is applied. The tool axis vector is the orientation of the tool shaft after the type of cut and lead angle are applied. You may select any of the following options for applying the tool axis vector.

- | | |
|------------------|---|
| <i>During XY</i> | The tool orientation is applied during the first XY move at the beginning of the program. |
| <i>At Top</i> | The first XY move is completed then the tool orientation is applied. |
| <i>During Z</i> | This option applies the orientation during the first Z axis move to the part. This is not recommended when there is no leadin move. |
| <i>At Bottom</i> | The orientation is applied after the Z axis move is made. The orientation is completed before the leadin move is started. |

Tool Vertical for Rapid

You may select *Yes* or *No* for this parameter. When *Yes* is selected the tool is positioned to vertical before any rapid move is made. When *No* is selected the tool orientation is maintained during the rapid moves.

Menu Item 219

Twist Vectors

Surfaces use Twist vectors to describe the curvature within a surface patch. SURFCAM represents surfaces by drawing their patch boundaries (the U and V splines), and, the Twist vectors do not affect the patch boundaries. SURFCAM will automatically recommend the correct twist vectors for all methods of surface construction. The different types of twist vectors are described below.

- Splined* This is the default for drive curve and 4 curve surfaces.
- Ruled* Is used whenever there are only two cross sections in a Cross section type surface. This results in a true ruled surface. When more than two cross sections are used, you may select Zero, Splined, or Blended twist vectors.
- Zero* Produces a Ferguson patch surface. Zero is usually used only to maintain compatibility with other systems that may not support splined or blended twist vectors.
- Blended* This is the default type for surfaces built from Cross sections.

Basis Type

There are two selections for the Basis Type option. You may select Unit or Chord for this option. This option determines the method of creating the surface. The two types as described below determine the curvature of the surface between cross sections.

Unit This type utilizes equally spaced node points to describe the surface curvature.

Chord This type determines the surface curvature based on proportional node point spacing. This allows the creation of more natural looking surfaces. This is the default method for Cross section surfaces.

Surface Variables

Tilt of Cross Section Planes

Menu Item 224

Offset Distance

Enter the desired distance between the offset surface and the original surface.

Offset Side

Select the desired side for the offset surface. The surface may be created toward the surface arrow or away from the surface arrow.

Offset Information

View Name

Enter the desired text for the view name.

Change View Name

View Name

Enter the desired name for the view created.

Reference Number

Enter the desired view number.

Change the Current View to This New View

Enter *Yes* to switch to the view just created. SURFCAM switches the display into the new view.
Respond with *No* to maintain the current view.

Enter Rotation Angle

Enter the angle to rotate the entities. A positive angle rotates the entities counterclockwise.

Enter Scale Factor

Enter the desired scale factor. A factor of 2 doubles the size of the entities. A factor of 1 maintains the same size of the entities.

Enter New Offset

Enter the desired offset value to be used to offset entities at a Distance.

Enter Number of Copies

Enter the desired number of copies to be created.

Cutter Comp at Top

Supports CNC controllers that require all cutter compensation moves to be made above the part surface or at the rapid plane. The default for this option is *No*. This does not require the tool to be at the rapid plane to activate or de-activate cutter compensation.

Optimize Error

A tolerance for the maximum chordal deviation of the cutter path. The smaller the value, the more accurate the cutter path will be to the part surface.

Lead Angle

After selecting the surface for Cut 5 axis, you are first prompted for a Lead Angle. You may enter any angle from -90 to 90 degrees. The Lead Angle value defines the angle at which the tool passes over the part. The input angle is in relation to the surface normal (measured in the direction of the cut). When the tool shank is perpendicular to the surface at the point of contact, it is normal to the surface.

You may select an angle that allows the tool shank to lead the tip across the surface. This is considered a positive lead angle. This action effectively drags the tool tip over the surface.

A user defined negative angle allows the tool shank to follow the tip across the surface. This produces a pushing effect on the cutter.

WARNING: It is not recommended to use a lag angle with standard end mills and bull nose end mills. This would require the bottom of the end mill to perform the cut, and may not produce the desired results during the cutting process.

Surface Fillet Maximum Evaluation Step

This is the maximum distance that SURFCAM will mathematically travel before generating an evaluation point to be used in the creation of a fillet surface. The default is 0.1000 inches. This step size may be reduced to improve fillet surface generation if the surface intersection to be filleted changes direction often within the evaluation step.

Amount to Remove

Indicate here the amount of stock to be removed both on the sides of the contour and in the Z axis. SURFCAM will use this value to determine how many cutting passes are going to be needed. For a single pass, set this to 0.

NOTE: When the Pocket option is selected, this option is not available for user input. SURFCAM automatically removes everything inside the main boundary paths.

Rough Spacing

This determines how much material the tool will remove on each pass. By default, for rough cutting, the on side value is set to 80% of the tool radius, and the in Z value is set to 15% - 75% of the tool radius, depending on the tool being used. It may also be set to a fixed value for the default.

Finish Passes

SURFCAM automatically incorporates finish passes into the cycle if you enter a non zero number. For a single pass around the contour, set this value to 1 or 0. You can, however, have as many finish passes as desired. Each pass will remove the amount of stock entered in the Finish cut spacing parameter.

Finish Spacing

Specifies the amount of stock to be removed in each finish pass.

Start Z Angle > 90

The Start Z Angle is used to direct the shank orientation at the beginning of the surface cut. The tool is shifted off the axis of rotation centerline. The Start Z Angle determines the direction off centerline to shift. This choice allows you to choose either side of centerline for the shank orientations when the swarf cutting starts.

Choosing No will leave the tip of the tool below the shank (in Z, as depicted on the screen). The tool is oriented to the side of the part that requires the least amount of part rotation.

Select Yes to begin cutting with the tool shank on the opposite side of the rotary axis centerline.

Stock to Leave

This cutting operation will leave the stock, in the amount of the value entered, on the sides and/or bottom of the contour. This means that the depth of the contour as cut will not stop at the exact Z level entered in the Depth reference parameter, but will be adjusted by the amount entered in the Stock to leave parameter. Using a negative amount will cause SURFCAM to overcut the contour on the sides and/or bottom.

Amount to Remove

Indicate here the amount of stock to be removed both on the sides of the contour and in the Z axis. SURFCAM will use this value to determine how many cutting passes are going to be needed. For a single pass, set this to 0.

NOTE: When the Pocket option is selected, this option is not available for user input. SURFCAM automatically removes everything inside the main boundary paths.

Rough Spacing

This determines how much material the tool will remove on each pass. By default, for rough cutting, the on side value is set to 80% of the tool radius, and the in Z value is set to 15% - 75% of the tool radius, depending on the tool being used. It may also be set to a fixed value for the default.

Finish Passes

SURFCAM automatically incorporates finish passes into the cycle if you enter a non zero number. For a single pass around the contour, set this value to 1 or 0. You can, however, have as many finish passes as desired. Each pass will remove the amount of stock entered in the Finish cut spacing parameter below.

Finish Spacing

Specifies the amount of stock to be removed in each finish pass.

Stock to Leave

This cutting operation will leave the stock, in the amount of the value entered, on the sides and bottom of the contour. This means that the depth of the contour as cut will not stop at the exact Z level entered in the Depth reference parameter, but will be adjusted by the amount entered in the Stock to leave parameter. Using a negative amount will cause SURFCAM to overcut the contour on the sides and or bottom.

Main Menu

When the SURFCAM system is executed the *Main* menu is displayed. The *Main* menu allows you to select from the available system functions.

<i>Create</i>	The <i>Create</i> menu allows you to create many types of elements. Each choice on the <i>Create</i> menu matches a different type of element that SURFCAM can create, and each will cause a submenu to be displayed that lists the different ways that each element may be created.
<i>Edit</i>	The <i>Edit</i> menu allows you to edit, or change, the attributes of elements. You may also edit an INC file created during any machining sequence.
<i>Display</i>	The <i>Display</i> menu enables you to manipulate the image. It can be accessed from the <i>Main</i> menu, or by pressing the desired icon .
<i>Transform</i>	The <i>Transform</i> menu allows you to perform a variety of transformation functions on elements in order to move or copy them to another location or view.
<i>Delete</i>	The <i>Delete</i> menu enables you to delete elements using the <i>Delete</i> selection menu. You may also delete views and layers. In addition to deleting, you may undelete (restore) the last element or all previous elements deleted.
<i>Files</i>	The <i>Files</i> menu enables you to load information from disk files into SURFCAM, save work into disk files, display the contents of disk directories, and other tasks that involve information stored in disk files. During these tasks, SURFCAM often requires you to enter the name of a file.
<i>Layers</i>	SURFCAM allows you the ability to organize the drawing by placing entities on different <i>Layers</i> . These layers can be made visible, invisible, selectable, and unselectable. SURFCAM supports the use of 256 layers numbered 0 - 255.
<i>Analyze</i>	The <i>Analyze</i> menu enables you to display physical information about elements, locations and distances on the screen.
<i>Options</i>	The options presented to you from this menu allows access to system parameters and settings. The default can be changed by editing the SURFCAM.CFG file.
<i>Exit</i>	This option exits the current session.

Menu Item 253

Menu Item 254

New Twist Type

You may change a surfaces twist vectors with this option. When the surface is selected a pop-up is displayed for you to select the desired twist vectors for the surface. The surface is recalculated with the new twist vectors.

Blended This is the default blending type for surfaces built from Cross sections and offset surfaces.

Zero This option uses for compatibility with other systems.

Ruled This option is used when there are only two cross sections.

Splined This is the default for drive curve and 4 curve surfaces.

Tool Information

For all of the options on the NC menu that produce tool motion, a tool must be described. So that you can enter the required tooling information, SURFCAM will present a *Tool Information* pop-up box.

The first line in the pop-up box allows you to select the tool library. Click the button and the tool library pop-up box containing a list of the tools in the SURFCAM.TOL file will appear. This is presented so you can easily review the tools and pick the one desired. As the mouse is moved horizontally through the columns in the pop-up, the prompt line shows what type of information is in the columns. To scroll through the tool library, use the up or down arrow keys or the Page Up and Page Down keys. The mouse may also be used to page through the library. Place the arrow above (page up) or below (page down) the tool pop-up box. Press the left mouse button to page through the list.

After selecting a tool from the tool library pop-up, either by moving the cursor to the line that contains the tool and clicking the left mouse button or by entering the reference number from the keyboard, you are returned to the *Tool Information* pop-up.

NOTE: The SURFCAM.TOL file can be modified using the **SURFCAM Configuration Utility**.

Tool Corner Radius

Specifies the tip radius of a tool. In the case of a ball end mill, this is the tool radius; for a bullnose cutter, this is the corner radius.

Tool Total Radius

Specifies the larger radius of a bullnose cutter. If the cutter total radius is greater than the tip radius, SURFCAM uses a bullnose cutter.

Use The Same Locations As Your Last Selection

This pop-up allows you to drill the same locations as previously selected without performing the selections. Enter Yes to use the previous locations. Enter No to select new locations.

Drill Information

The first line allows you to access the tool library, SURFCAM.TOL, by clicking on the button.

Tool Diameter

Unlike the usual *Tool Information* pop-up that appears, the tool diameter is indicated here, instead of the tool radius.

Hole Depth

This is relative to the Z location of the elements being drilled. To drill a through hole, add about 1/3 of the drill diameter to the thickness of the stock. Enter the calculated value in this parameter.

Type of Cycle

Allows you to specify one of the seven different types of canned cycles. A small sub-menu showing all the possible choices can be accessed by clicking the button.

Drill Straight drilling: feeds down to bottom Z then rapids out (G81 on most machines).

Peck Standard peck drilling. The peck increment is described below (G83 on most machines).

Tap Use this choice to access a standard tapping cycle that feeds down, then reverses the spindle at the bottom of the hole and feeds up (G84 on most machines).

NOTE: Tapping requires the feed rate to match the spindle speed. SURFCAM can do this automatically. Set the tapping tools in the SURFCAM.TOL file with 1 flute. Set the chip load to the lead value of the tap. A 1/4-20 tap would have a chip load value of 0.05.

Left tap Use this choice to access a Left-handed tapping cycle that operates exactly like the Tap cycle with reversed spindle direction.

Ream This is similar to the Drill cycle, but this will cause the tool to FEED out of the hole instead of rapiding out and is usually used with a reamer (G85 on most machines).

Bore This is a fine boring cycle that will bore to the bottom of the hole. The spindle will stop and orient. The tool is then moved away from the edge of the hole a small distance. The tool will then rapid out of the hole (G86 on most machines).

Back This is a back-boring cycle that will stop and orient the spindle before the cycle starts. The tool rapids to the bottom of the hole, and feeds toward the wall before starting the spindle. The tool will then feed up to the top of the hole, boring the hole from the bottom up.

Dwell Time

When the cycle type chosen uses a dwell value, enter the desired dwell value on this line.

Rapid Levels

Bounding Curve

A bounding curve is a closed series of elements that is used to contain the tool motion. This curve may be used with both types of cutting direction to limit the area being machined.

Type of Cut

Specifies the two different types of tool motion listed below:

- On Contour* This is used to drive the center line of the tool directly along the path of the chained elements, as would be done to cut a groove or slot. When using a ball end mill the center of the ball is driven along the contour.
- On Top* Using this selection maintains the tool tangent to the contour while the tool tip is above the contour in the Z axis.

NC 3 Axis Menu

SURFCAM allows you to perform a variety of 3 axis machining operations including cut, project, auto rough, planar, and contour cutting. These features may be performed on parametric and NURBS* surfaces.

SURFCAM 3 axis requires either a Surface or, a Spline. The creation of these geometric entities is described in sections concerning Create Spline and Create Surface.

To activate SURFCAM's 3 axis capabilities, select 3 Axis from the NC Mode menu. This allows you to choose the cutting options below.

<i>Cut</i>	This option allows you to perform 3 axis machine cutting.
<i>Project</i>	You may project an existing INC file onto a surface.
<i>Auto rough</i>	This option allows you to create a rough cut of a surface.
<i>Planar</i>	You may cut across single or multiple surfaces with this option.
<i>Contour 3D</i>	This option allows you to make a single 3 axis cut along a chain of elements.
<i>Drill</i>	This option is used for Z axis canned cycles including drilling, reaming, tapping, and boring.
<i>Options</i>	The 3 Axis cutting parameters may be set using this option.
<i>Edit</i>	The tool path may be edited using this option.
<i>Post</i>	This option displays the current Post processor menu.
<i>NC mode</i>	This option is used to select the desired NC menu.

NOTE: When creating the entities to cut, it is important to place the geometry in the proper 3D space.

Part orientation, in 3D space, is important to 3 axis cutting. The physical orientation of the part placement on the machine tool must be considered before creating the part geometry with SURFCAM. This ensures that the NC code and machine tool motion are correlated.

Edit Polyline Menu

This menu allows you to edit a polyline.

Move node Use the Move node option to move one or more nodes by translating.

Add node You may select locations to add to the selected polyline.

Delete node This option allows you to delete selected nodes from a polyline.

Explode The Edit Polylines Explode option is used to create individual line elements that resemble the chosen polyline. It is similar to Create Line Explode option except the polyline is deleted.

Tool Library

This is presented so you can easily review the tools and pick the one desired. As the mouse is moved horizontally through the columns in the pop-up, the prompt line shows what type of information is in the columns. To scroll through the tool library, use the up or down arrow keys or the Page Up and Page Down keys. The mouse may also be used to page through the library. Place the arrow above (page up) or below (page down) the tool pop-up box. Press the left mouse button to page through the list.

After selecting a tool from the tool library pop-up, either by moving the cursor to the line that contains the tool and clicking the left mouse button or by entering the reference number from the keyboard, you are returned to the *Tool (Drill) Information* pop-up.

NOTE: The SURFCAM.TOL file can be modified using the **SURFCAM Configuration Utility**.

Debug level (Testing)

Surfaces Selectable By

This option allows you to choose the following methods of selecting a surface.

Patch edges Allows surfaces to be selected with the mouse anywhere, including the arrow portion.

Arrow only Limits the surface selection so that you may select a surface by picking the arrow only. This option speeds up surface selection for very large DSN files.

NC Edit Menu

Edit NC Menu

This option is used to graphically edit the contents of tool motion, or INC files. An INC file is made up of a series of records that can contain tool movement information such as rapid moves, feed moves, arc moves, etc., and other information that is needed to produce a complete NC program.

The *Edit NC* menu enables you to edit tool motion graphically by moving, inserting, or deleting tool motion on the screen. Since an INC file also contains tooling and machine tool parameters, the *Edit NC* menu also allows you to easily insert, delete and/or alter this information.

There are two groups of menu options used to edit the file. In the first group are choices that provide different ways of selecting a location on the tool path and the corresponding line of code where the editing is to be accomplished. SURFCAM indicates on the prompt line what type of INC record the current location is. When it is movement of some kind, SURFCAM marks the end point of that movement with a small square.

<i>Start</i>	Moves to the first record in the file.
<i>Previous</i>	Moves to the previous record. This is one record closer to the start of the file.
<i>Pick</i>	Allows you to mouse the tool path move to be edited. SURFCAM moves to the location closest to the click point.
<i>Next</i>	Moves to the next record in the file. This is one record closer to the end of the file.
<i>End</i>	Moves to the last record in the file.

In the next group are the choices that you can select to edit the current INC record:

<i>Edit</i>	<p>Allows you to edit, change, the current INC record, the one displayed on the prompt line at the top of the screen.</p> <p>If you edit a movement, Line, Plunge, or Rapid, record, the Location menu will appear, and you may indicate a new location for the current move. (The Relative option on the Location menu is very handy for shifting a location by a specific X, Y, and Z amount.)</p> <p>Editing other types of records may result in other pop-ups appearing on the screen. Any changes made in the pop-ups will be recorded into the INC file when the Done option is selected from the menu.</p>
<i>Insert</i>	This option displays a menu for you to insert moves into the INC file.
<i>Delete</i>	This option allows you to delete moves from the INC file.
<i>Done</i>	Select when all editing is complete.

NC Vert Menu

Edit / NC / Insert Menu

<i>Insert</i>	SURFCAM presents a sub-menu. Selecting one of these options allows you to insert a record at the current editing location in the INC file.
<i>Rapid, Line, Plunge</i>	When you choose one of these, you will be able to indicate a move, of the type chosen, to a location that you specify.
<i>Hole</i>	When hole is selected the system prompts for the hole location. After selecting the hole location, enter the hole depth.
<i>End row</i>	This is a simple record that marks the end of a pass of the tool across the surface or pocket. It requires no other input.
<i>Feed Rate or Plunge rate</i>	Insert one of these records to alter the feed or plunge rate for subsequent movements in the INC file. Choosing either of these will cause SURFCAM to present a small pop-up window that prompts for a new rate.
<i>Tool change</i>	This will allow you to insert a tool change into the INC file. SURFCAM will present both the <i>Tool Information</i> and <i>Program Information</i> pop-up boxes.
<i>Comment</i>	Comments are similar to <i>Inline</i> , except that the comment start and end characters will be added as well. Verify that proper beginning and ending comment characters are defined in the POSTFORM file.
<i>InLine</i>	SURFCAM supports direct insertion of inline machine coding by using these choices. Anything typed in response to the <i>InLine</i> pop-up will appear on its own line in the final NC program exactly as typed, except that an N number may be added at the beginning by the post processor.
<i>NC Mode</i>	Allows you to select NC mode: Lathe, Wire EDM, 2 axis, 3 axis, 4 axis, or 5 axis.

Mat Library #

Enter the number for the desired material type. Click accept or press the Enter key when the desired material number is selected.

NC 2 Axis / Drill Menu

The Select menu is displayed to allow you to choose the points and circles to drill. Choose Backup, or press the backspace key, after selecting all the elements to drill. When you select main, the operation will be aborted. Selecting Backup returns to another menu that contains the following choices:

Get Points Choosing this will display a pop-up to use the same locations as last selection. When you select Yes, SURFCAM will process the same locations into the INC file that were last selected. When you select No, SURFCAM returns to the Select menu to pick elements to drill.

New Tool Allows you to continue building the INC file with additional tools. The select procedure is repeated.

Done Select this option to complete the INC file. The NC 2 axis menu is displayed.

NOTE: Drilling, contouring, and pocket cycles support rotary table indexing by utilizing CView machining. By specifying the correct CView and setting Coords to view, the appropriate indexing move will be generated prior to any tool motion. This requires the POSTFORM.M file to be properly configured to take advantage of Indexing capabilities.

NC 4 Axis Menu

SURFCAM allows you to perform a variety of 4 Axis machining operations including normal to surface, lead angle, lag angle, swarf, contour, and project cutting. These features may be performed on parametric and NURB surfaces.

To activate SURFCAM's 4 Axis capabilities, select 4 Axis from the NC Mode menu. This allows you to choose the cutting options listed below.

<i>Cut</i>	This option allows you to perform 4 axis machine cutting.
<i>Project</i>	You can project an existing INC file onto a surface.
<i>Swarf</i>	This option allows you to cut a surface with the side of the tool.
<i>Contour</i>	You can perform a rotary cut along a chain of elements.
<i>Drill</i>	This option is used for Z axis canned cycles including drilling, reaming, tapping, and boring.
<i>Set Axis</i>	This option allows you to specify the rotary axis. The default rotary axis is set in the SURFCAM.CFG file with the <i>Rot 4 Axis</i> parameter. See the Configuration Utility chapter of the manual for information on how to set this parameter.
<i>Options</i>	The 4 Axis cutting parameters may be set using this option.
<i>Edit</i>	The tool path may be edited using this option.
<i>Post</i>	This option displays the current <i>Post Processor</i> menu.
<i>NC mode</i>	This option is used to select the another NC menu.

SURFCAM 4 Axis requires Surfaces and Splines prior to generating 4 axis simultaneous motion. The creation of these geometric entities is described in sections Create Spline and Create Surface.

Part orientation, in 3D space, and the axis of rotation are important to 4 axis cutting. These two factors affect the Rotary axis zero. It is recommended that the physical orientation of the rotary table on the machine tool be considered before creating the part geometry with SURFCAM. The part geometry and axis of rotation must be created in relation to the physical placement of the rotary table. This ensures that the proper NC code and machine tool motion are created.

NOTE: When performing a 4 axis cut, SURFCAM visually displays the tool moving around the geometry. The tool motion depicted is opposite the actual rotary motion created. Clockwise tool motion, around the axis of rotation, creates positive Rotary axis rotational moves. Counterclockwise tool motion, around the axis of rotation, creates negative Rotary axis rotational moves.

NOTE: The sign of the rotary axis motion may be overridden by the post processor.

SURFCAM 4 Axis creates tool path files along three linear axes and one rotary axis or two linear and one rotary axis. The rotational axis may be located at any orientation. Typical placement of the rotary axis is along the X axis or Y axis.

Operation Turn

The *Turn* operation allows you to machine the part along the length of the material. This is typically accomplished with the cutting motion along the Z axis. This axis designation may vary with specific machine tools. Below is a list of the available tuning operations.

OD Turn

OD Back Turn

ID Back Bore

ID Bore

NOTE: This section uses the X axis for diameters and the Z axis for lengths.

You may select either a rough, finish or both cuts for each Turn operation. When this option is selected you are prompted to select the beginning and ending elements to cut. You are then prompted for the optional material boundary beginning and ending elements. After entering the INC filename the tool information and program information pop-ups are displayed as described previously in this section.

Gouge Checking

Front and back angle gouge checking is automatically applied to all rough and finish tool paths. SURFCAM uses the user-defined angles from the cutting edge of the tool to perform gouge checking. These angles are set in the SURFCAM.TOL file. These angular values may not be overridden from within SURFCAM. To ensure proper gouge checking, the tool library must be updated to users actual tools.

NOTE: When you override any of the insert parameters such as tool nose radius the system will use the last set of angles from the last tool selected from the library.

The Undercut parameter in the Tool pop-up allows you to attempt to cut or ignore undercuts during turning operations. When this parameter is set to No, SURFCAM ignores all undercuts in the chained path. When the parameter is set to Yes, SURFCAM attempts to cut the undercut while performing gouge checking.

Tool Nose Radius

This parameter indicates the radius of the tool nose. You may override the tool nose radius.

Groove Tool Width

Indicates the width of the groove tool.

NOTE: This parameter is not available for input unless a grooving tool is selected from the NC menu. The text is shown in a different color and the input space is not displayed.

Tip or Center

Normally, the NC code produced will drive the *Tip* of the cutter. Choosing *Center* will produce NC code for the center of the toolnose.

Cutter Compensation

This parameter sets the cutter compensation for turning. You may select *None*, *SURFCAM*, *Control*, or *Both*. This also determines the tool nose compensation direction. This parameter is automatically set based on the tool and geometry.

None The tool path is not offset and machine cutter compensation codes are not output.

NOTE: When *Tip* is selected this option may cause an undercut of the part line.

SURFCAM SURFCAM offsets the toolpath for the tool radius and no machine cutter compensation codes are output.

Control The tool path is not offset and machine cutter compensation codes are output.

Both SURFCAM offsets the tool path and the machine cutter compensation codes are output.

Curve Tolerance

This parameter is used to set the tolerance for the tool path to maintain while cutting. This is the value used for chordal deviation on splines.

Rapid Clearance

The distance from the origin, along the retract angle, that the tool may be moved at rapid travel. When this parameter is set below the highest point on the part, it is updated to the highest point on the part.

Stock X

This option allows you to specify the amount of stock to leave on each cut axis. The axis designations described in the SURFCAM.CFG file are displayed. Enter the amount of stock to leave on the part for each axis.

The Operation selected sets the default values for the parameters below. You may override these values.

Turn on Repaint Timer

This option allows you to activate or deactivate the repaint timer.

NC 2 Axis / Contour Menu

This operation may be used to cut on the left or right side, or on the contour of the boundary shape. The boundary shape may be open or closed. Contour differs from pocket in that you must define the amount of stock to be removed. You may perform rough and finish operations separately or in one operation.

When the amount of stock to remove produces collapse areas, the cutter path spirals outward to the boundary shape. As the cutter path for the collapse areas intersect, the tool is retracted to a clearance plane to move to the next collapse area. After all collapse areas are cut to intersection, the pocketing operation continues to spiral outward following the boundary shape.

SURFCAM 2 axis contouring may be used to cut multiple boundary shapes within the same INC file. It may also be used to avoid islands or clamps inside the boundary shape.

Paths This option may be used to select additional boundary shapes for machining multiple contours. It may also be used to avoid cutting shapes that intersect the main boundary shape. To avoid cutting into a clamp that is hanging over the edge of a contour, simply create lines, arcs, or splines that surround the area where the clamp overlaps the main boundary shape. Chain them using this Paths option to avoid cutting that area during the contour cut.

Islands To avoid cutting an island inside the main boundary shape, use this option to select the island shape.

NOTE: The island must be chained in the opposite direction of the contour boundary shape. If not, the island will be cut.

Plunge Use this option to specify plunge locations. SURFCAM will automatically move the tool to the closest user defined plunge point and feed to the cutting depth, then to the beginning point of the cycle's pass. If plunge points are not specified, plunge positions will be automatically calculated by SURFCAM.

Done Select this option after all paths and islands have been chained.

NOTE: Contouring supports rotary table indexing by utilizing CView machining. By specifying the correct CView and setting Coords to view, the appropriate indexing move will be generated prior to any tool motion. This requires the POSTFORM.M file to be properly configured to take advantage of Indexing capabilities.

Geometry At

The following options determine how the total depth of the cutting is measured.

top The geometry that was selected to define the contour or pocket determines the Z axis position of the top of the contour or pocket. The bottom of the contour or pocket is calculated by subtracting the amount to remove from the Z value of the selected geometry.

bottom The geometry that was selected to define the contour or pocket determines the Z axis position of the bottom of the contour or pocket. The top of the contour or pocket is calculated by adding the amount to remove to the Z value of the selected geometry.

Bottom Z Absolute Z axis position of the bottom of the contour or pocket. The top of the contour or pocket is calculated by adding the amount to remove to the bottom Z value entered here.

Top Z Absolute Z axis position of the top of the contour or pocket. The bottom of the contour or pocket is calculated by subtracting the amount to remove from the top Z value entered here.

NOTE: The parameters above let you set the Z depth at which the contour would be finished; however, any non zero value that is entered in the stock to leave parameter in the in Z column (described below) will adjust the lowest Z that this particular operation will cut.

Tip or Center

Normally, the NC code produced will drive the *Tip* of the cutter. When using a ball end mill and the tool path for the ball center is desired, choose *Center*.

Edit / Trim/Break Menu

This option allows you to trim or break elements.

Trim The Trim function on the Edit menu enables you to trim or extend one or two elements. By specifying Trim 1, you can select one element to Trim and then another element or location to which the element is trimmed. By specifying Trim 2, you can select two elements. Each element is trimmed to the intersection, or projected intersection, of the two elements.

The number of items trimmed (1 or 2) can be toggled by selecting the Trim option on the Edit menu when Trim is active.

NOTE: Splines that are part of a composite surface cannot be trimmed, broken or optimized. This helps preserve the integrity of the composite surface.

Break This option is used to break a single element into two parts. The resulting two elements will occupy the same exact space as the original single element.

The Break function on the Edit menu enables you to select either one or two elements to be broken. By specifying Break 1, you can select one element to break and then another element or location at which the break occurs. By specifying Break 2, you can select two elements. Each one will be broken at the intersection, or projected intersection, of the two elements.

The number of items broken (1 or 2) can be toggled by selecting the Break option on the Edit menu when Break is active.

NOTE: It is recommended that the active color be different from that of the elements being broken.

Modal This option allows you to trim multiple elements to one or more elements continuously. The difference between this option and *TRIM 1* is that this option allows multiple elements to be trimmed continuously without having to select the *trim to* element repeatedly.

Divide This option allows you to trim away multiple elements from within other elements.

Menu Item 297

Maximum Feed Between Size

When the tool finishes one pass and must move to the start of the next pass, it can either rapid up and over, or feed across the part at the current feed rate setting. If the distance from the end of one pass to the start of the next is smaller than the value specified here, the tool will feed across without retracting from the work.

NOTE: Too large a value could cause the tool to overcut the part by feeding through a narrow area. To avoid this reduce the maximum feed between size to less than the distance to the start of the next pass. This will cause the tool to pick up to the rapid plane, move over, then plunge down to start the next pass.

Corner Clearance

When the Corner angle cut is in effect, this value determines the tool clearance from the direction change intersection. When this value is zero, the tool remains in contact with the part walls during the directional change.

Side Clearance

This allows you to set the default side clearance value that is used for the infeed and outfeed movements at the start and end of each pass.

Linear Leadin Length

Specifies the length of the linear leadin moves.

Option Parameter Type

Choices here are *Constant*, *Cutter Radius*, and *Step Size*. Use the *Cutter Radius* or *Step Size* for a particular value to be calculated automatically by multiplying the number entered in the first column by the cutter radius or the step size. For the value to always remain the same, set the 2nd column to *Constant*. The value entered will be the exact inch, or millimeter, value of that parameter and will not vary with step or tool size.

Arc Leadin Angle

This option sets the sweep angle for a leadin and leadout move.

Constant

This field cannot be changed.

Arc Leadin Radius

This specifies the size of the leadin radius.

Side Roughing Step

The default distance between roughing passes can be set at this option. This is the distance for the side of the tool to cut into the work on each rough pass. The default overlap is 80% of the tool radius.

Side Finishing Step

This option sets the default distance between finishing passes.

Depth Roughing Step

This is the default vertical Z distance between each roughing level.

Depth Finishing Step

This option sets the default spacing for finishing passes.

Rapid Plane Clearance

This is the default Z level for lateral rapid motions.

Leadin Type

SURFCAM can add linear or arc motion that cause the tool to approach and depart the workpiece with a tangent move. The list of all of the Leadin types can be accessed by clicking the button on the corresponding line of the pop-up.

<i>None</i>	No leadin or out moves are performed.
<i>Line on final</i>	A straight line move is used for the leadin or out move. This is only performed on the finish pass.
<i>Arc on final</i>	An arc is used to move the start point of the finish pass. The arc direction is determined by the tool path direction, so that the move becomes tangent while moving in the same direction as the tool path for the pass.
<i>1) line on all</i>	The straight line leadin or out move is performed for all passes in the tool path.
<i>2) arc on all</i>	The arc move is performed for all passes in the tool path.

When cutting around the outside of a contour, line infeed moves are helpful for causing the tool to plunge far enough away from the finished contour to completely clear the workpiece. This is highly advisable to prevent the tool from plunging directly into the workpiece.

When cutting a finish pass on a contour, using arc infeed moves will reduce the dwell mark usually found at the point the tool first contacts the finished edge of the workpiece.

If None is chosen, the tool will plunge closer to the start point of the contour and infeed in a perpendicular motion.

You are allowed the same options for departing the contour as for approaching the contour in Leadin type. The options are listed above.

NOTE: SURFCAM allows you to control the choice of infeed and outfeed moves on all passes, or just the finish passes.

NOTE: Gouge checking is not performed for leadin or out moves.

Leadout Type

SURFCAM can add linear or arc motion that cause the tool to approach and depart the workpiece with a tangent move. The list of all of the Leadin types can be accessed by clicking the button on the corresponding line of the pop-up.

<i>None</i>	No leadin or out moves are performed.
<i>Line on final</i>	A straight line move is used for the leadin or out move. This is only performed on the finish pass.
<i>Arc on final</i>	An arc is used to move the start point of the finish pass. The arc direction is determined by the tool path direction, so that the move becomes tangent while moving in the same direction as the tool path for the pass.
<i>1) line on all</i>	The straight line leadin or out move is performed for all passes in the tool path.
<i>2) arc on all</i>	The arc move is performed for all passes in the tool path.

When cutting around the outside of a contour, line infeed moves are helpful for causing the tool to plunge far enough away from the finished contour to completely clear the workpiece. This is highly advisable to prevent the tool from plunging directly into the workpiece.

When cutting a finish pass on a contour, using arc infeed moves will reduce the dwell mark usually found at the point the tool first contacts the finished edge of the workpiece.

If None is chosen, the tool will plunge closer to the start point of the contour and infeed in a perpendicular motion.

You are allowed the same options for departing the contour as for approaching the contour in Leadin type. The options are listed above.

NOTE: SURFCAM allows you to control the choice of infeed and outfeed moves on all passes, or just the finish passes.

NOTE: Gouge checking is not performed for leadin or out moves.

Side Clearance Type

You can toggle this choice to indicate where to place the side clearance moves applied:

- *On Islands (only)*
- *On All contours*
- *With C-Comp.*

The *With C-Comp* choice will cause the resulting NC program to add diameter compensation codes to the infeed and outfeed moves, G41, G42 and G40 on most machines.

Side Clearance

Indicates the desired length of the perpendicular infeed and outfeed moves. When diameter compensation is desired at the machine, this value must be larger than the value stored in the corresponding diameter offset register on the machine, otherwise the machine may not be able to correctly cut the contour.

Diameter compensation may be performed using two methods. You may compensate for the entire tool diameter. This is normally used when the part line is programmed. The tool is offset for the tool radius to maintain the proper cut location. You may also program the tool centerline. You may also compensate for undersize tools. This is typically used when programming the tool centerline. The compensation is adjusted for the difference in the tool diameter programmed and the tool diameter being used.

This parameter is used to set the distance above the part to end the rapid move. It is also used as the minimum amount that the rapid plane must be above the part.

NC 3 Axis Menu

SURFCAM allows you to perform a variety of 3 axis machining operations including cut, project, auto rough, planar, and contour cutting. These features may be performed on parametric and NURBS* surfaces.

SURFCAM 3 axis requires either a Surface or, a Spline. The creation of these geometric entities is described in sections concerning Create Spline and Create Surface.

To activate SURFCAM's 3 axis capabilities, select 3 Axis from the NC Mode menu. This allows you to choose the cutting options below.

<i>Cut</i>	This option allows you to perform 3 axis machine cutting.
<i>Project</i>	You may project an existing INC file onto a surface.
<i>Auto rough</i>	This option allows you to create a rough cut of a surface.
<i>Planar</i>	You may cut across single or multiple surfaces with this option.
<i>Contour 3D</i>	This option allows you to make a single 3 axis cut along a chain of elements.
<i>Drill</i>	This option is used for Z axis canned cycles including drilling, reaming, tapping, and boring.
<i>Options</i>	The 3 Axis cutting parameters may be set using this option.
<i>Edit</i>	The tool path may be edited using this option.
<i>Post</i>	This option displays the current Post processor menu.
<i>NC mode</i>	This option is used to select the desired NC menu.

NOTE: When creating the entities to cut, it is important to place the geometry in the proper 3D space.

Part orientation, in 3D space, is important to 3 axis cutting. The physical orientation of the part placement on the machine tool must be considered before creating the part geometry with SURFCAM. This ensures that the NC code and machine tool motion are correlated.

Plunge Clearance

This is the relative point above the Z value of the drill location that the tool begins the feed move.

Initial Rapid

This is the absolute value above the Z location of the elements to be drilled. This is the point that the cycle is initiated.

NOTE: The Rapid plane between hole locations is dependent on the type of cycle desired (G98 type, retract to Initial Rapid, or G99 type, retract to the Plunge clear).

Peck

Increment

When you select *Peck* as the cycle type, use this space to enter the peck increment. This is the distance that the tool will feed down on each peck before moving back up in Z to break the chip.

Bore

Side Clearance

When you select *Bore* as the cycle type, use this space to enter the amount that you want the tool to shift away from the finished hole after the spindle has stopped. This is for clearance to rapid out of the hole.

Drill Information

Corner Angle

This option determines the tool path at each change of direction. When the interior angle of the directional change is larger than this value, the tool path is intersectional. Any directional change with the interior angle greater than 150 degrees is an intersectional tool path. Interior angles less than this value use either an Arc or Line to prevent the intersectional move. Angles smaller than 30 degrees automatically use either Arc or Line moves. The default for this value is 135 degrees.

Arc This option uses an arc between directional changes that are less than the Corner angle value. The tool path uses the Corner clearance amount to determine the location of the arc.

Line This option uses a line between directional changes that are less than the Corner angle value. The tool path uses the Corner clearance amount to determine the line location.

Corner Angle Move

- Arc* This option uses an arc between directional changes that are less than the *Corner angle* value. The tool path uses the *Corner clearance* amount to determine the location of the arc.
- Line* This option uses a line between directional changes that are less than the *Corner angle* value. The tool path uses the *Corner clearance* amount to determine the line location.

Create Menu

This menu allows you to draw the desired entities for the required part geometry. The entities listed below may be selected from the Create menu.

<i>Point</i>	You may create a point in the drawing using the <i>Location</i> menu. This menu is displayed when this option is selected.
<i>Points</i>	This option allows you to create multiple points. When selected the <i>Points</i> menu is displayed.
<i>Line</i>	The <i>Line</i> menu is displayed when this option is selected. You are prompted to select the type of line to create.
<i>Arc</i>	You may select this option to create arcs. The <i>Arc</i> menu is displayed for you to select the arc creation method.
<i>Circle</i>	This option is used to create circles. The <i>Circle</i> menu is displayed for you to select the circle creation method.
<i>Fillet</i>	The <i>Fillet</i> option is used to produce fillet radii between two elements or locations. The <i>Fillet</i> menu is displayed for you to create the fillet radius.
<i>Chamfer</i>	You may create a chamfer between two elements or locations with this option. The <i>Chamfer</i> menu is displayed for you to create the desired Chamfer.
<i>Spline</i>	You may select this option to create parametric or NURB splines. The <i>Spline</i> menu is displayed for you to select the desired spline type.
<i>Surface</i>	This option is used to select the desired surface type to create. The <i>Surface</i> menu is displayed for you to select the desired surface creation method and options.
<i>View</i>	You may use this option to create, change, or list views. The <i>View</i> menu is displayed for you to select the desired option or view creation method.
<i>Text/Dim</i>	This option provides access to a number of features to create text or dimensions. The <i>Text/Dim</i> menu is displayed for you to select the desired text or dimension creation method.
<i>Vector</i>	This option is used to define the offset side when creating a tool path in 2D.

Number of Rotary Axis

This option allows you to select the number of rotary tables used for 5 axis machining. Click the button to view the list of available options. Select 0 when the machining center has no rotary axes.

NC 2 Axis Menu

This menu allows you to perform 2 axis machining functions. The 2 Axis menu options are listed below.

<i>Pocket</i>	This option is used to remove the stock within a specific boundary shape.
<i>Contour</i>	This operation is used to cut to the left, right or on the contour of a boundary shape.
<i>Drill</i>	This option is used for Z axis canned cycles including drilling, tapping, reaming, and boring.
<i>Contour 3D</i>	You may contour a 3D boundary with this option.
<i>Options</i>	The 2 Axis cutting parameters may be set using this option.
<i>Edit</i>	The tool path may be edited using this option.
<i>Post</i>	This option displays the current Post processor menu.
<i>NC mode</i>	This option is used to select the desired NC menu.

Plunge Rate

Edit Curve Menu

Extra Segments

This command divides the NURB spline into a specified number of segments, and creates either a polyline or a parametric spline. Please note that since this method does not contain a tolerance, there can be no guarantee as to the accuracy of this conversion. If the conversion does not appear satisfactory, then increase the number of segments that are used. NURB splines that do not have the required level of continuity to be represented as splines are broken into separate splines. The data is created on the current layer and in the current color. (You may want to keep the current layer free to clean up when converting NURB splines to parametric splines/polylines.)

Enter the desired number of segments to create.

Pocket Menu

The 2 Axis pocket operation is used to remove all stock within the specified boundary shape.

Pocket operations may be accomplished with open or closed boundary shapes. When the boundaries are open, SURFCAM continues to offset the tool path until it collapses.

Gouge checking is accomplished for the tool path with unlimited look-ahead. When a gouge point is detected, SURFCAM creates a collapse area.

SURFCAM begins the pocketing operation at the center most point of the boundary shape and spirals outward. When the boundary path produces collapse point the cut spirals outward from the center most point of each collapse area. As the cutter path for the collapse areas intersect, the tool is retracted to a clearance plane to move to the next collapse area. After all collapse areas are cut to intersection, the pocketing operation continues to spiral outward following the boundary shape.

SURFCAM 2 axis pocketing may be used to cut multiple boundary shapes within the same INC file. It may also be used to avoid islands or clamps inside the boundary shape. After selecting the initial pocket boundary shape, a menu with the following options will be displayed:

Paths This option may be used to select additional boundary shapes for machining multiple pockets. It may also be used to avoid cutting shapes that intersect the main boundary shape. To avoid cutting into a clamp that is hanging over the edge of a pocket, simply create lines, arcs, or splines that surround the area where the clamp overlaps the main boundary shape. Chain them using this Paths option to avoid cutting that area during the pocket cut.

Islands To avoid cutting an island inside the main boundary shape, use this option to select the island shape. NOTE: The island must be chained in the opposite direction of the pocket boundary shape. If not, the island will be cut.

Plunge Use this option to specify plunge locations. SURFCAM will automatically move the tool to the closest user defined plunge point and feed to the cutting depth, then to the beginning point of the cycle's pass. If plunge points are not specified, plunge positions will be automatically calculated by SURFCAM.

Done Select this option after all paths and islands have been chained.

NOTE: Pocketing supports rotary table indexing by utilizing CView machining. By specifying the correct CView and setting Coords to view, the appropriate indexing move will be generated prior to any tool motion. This requires the POSTFORM.M file to be properly configured to take advantage of indexing capabilities.

Tool Library

This is presented so you can easily review the tools and pick the one desired. As the mouse is moved horizontally through the columns in the pop-up, the prompt line shows what type of information is in the columns. To scroll through the tool library, use the up or down arrow keys or the Page Up and Page Down keys. The mouse may also be used to page through the library. Place the arrow above (page up) or below (page down) the tool pop-up box. Press the left mouse button to page through the list.

After selecting a tool from the tool library pop-up, either by moving the cursor to the line that contains the tool and clicking the left mouse button or by entering the reference number from the keyboard, you are returned to the *Tool (Drill) Information* pop-up.

NOTE: The SURFCAM.TOL file can be modified using the **SURFCAM Configuration Utility**.

Tool Library

This is presented so you can easily review the tools and pick the one desired. As the mouse is moved horizontally through the columns in the pop-up, the prompt line shows what type of information is in the columns. To scroll through the tool library, use the up or down arrow keys or the Page Up and Page Down keys. The mouse may also be used to page through the library. Place the arrow above (page up) or below (page down) the tool pop-up box. Press the left mouse button to page through the list.

After selecting a tool from the tool library pop-up, either by moving the cursor to the line that contains the tool and clicking the left mouse button or by entering the reference number from the keyboard, you are returned to the *Tool (Drill) Information* pop-up.

NOTE: The SURFCAM.TOL file can be modified using the **SURFCAM Configuration Utility**.

Tool Library

This is presented so you can easily review the tools and pick the one desired. As the mouse is moved horizontally through the columns in the pop-up, the prompt line shows what type of information is in the columns. To scroll through the tool library, use the up or down arrow keys or the Page Up and Page Down keys. The mouse may also be used to page through the library. Place the arrow above (page up) or below (page down) the tool pop-up box. Press the left mouse button to page through the list.

After selecting a tool from the tool library pop-up, either by moving the cursor to the line that contains the tool and clicking the left mouse button or by entering the reference number from the keyboard, you are returned to the *Tool (Drill) Information* pop-up.

NOTE: The SURFCAM.TOL file can be modified using the **SURFCAM Configuration Utility**.

Tool Library

This is presented so you can easily review the tools and pick the one desired. As the mouse is moved horizontally through the columns in the pop-up, the prompt line shows what type of information is in the columns. To scroll through the tool library, use the up or down arrow keys or the Page Up and Page Down keys. The mouse may also be used to page through the library. Place the arrow above (page up) or below (page down) the tool pop-up box. Press the left mouse button to page through the list.

After selecting a tool from the tool library pop-up, either by moving the cursor to the line that contains the tool and clicking the left mouse button or by entering the reference number from the keyboard, you are returned to the *Tool (Drill) Information* pop-up.

NOTE: The SURFCAM.TOL file can be modified using the **SURFCAM Configuration Utility**.

Copy to Color

You may choose to create the new elements with the current color and layer attributes or the attributes from the original elements being copied.

Copy to Layer

You may choose to create the new elements with the current color and layer attributes or the attributes from the original elements being copied.

Feed rate

This option sets the method for feed on leadin moves from the plunge point. This feed rate may use the *Feed*, *Rapid*, or *Plunge* speeds.

Create / Points Menu

Create / Line Menu

Edit / Surface Menu

Taper Angle

Allows the specification of a constant draft angle as applied to a contour or pocket. The default angle is zero degrees.

NOTE: This parameter is not available with SURFCAM 2 Axis.

NURB Surface

Polygon

Creates a NURB surface with or without displaying the controlling polygonal mesh.

NURB splines are defined by a set of points in space. Unlike splines where the spline passes through the points that define the spline, the points defining the NURB spline (except the first and last) do not usually lie on the spline. Sometimes they are called puppet points, as each controls a piece of the spline and moving one point causes a portion of the spline to move in that direction.

A NURB surface is defined by a rectangular mesh of control points.

Gouge check

Select either *Full* or *Single* to perform 2D gouge checking. This may be answered *Single* when contouring intersecting elements.

Leadin Direction

This parameter selects the leadin move type.

<i>None</i>	No leadin move is made when this option is selected.
<i>Normal</i>	This option directs the tool to leadin to the surface along a normal to tool path at the pass start point. The move is perpendicular to the surface.
<i>Tangent</i>	The leadin move for this option is tangent to the surface in the direction of the cut.
<i>Tangent Angle</i>	The leadin move for this option is tangent to the surface, plus a defined angle, in the direction of the cut.
<i>Fixed Angle</i>	The leadin move for this option is at a fixed angle to the surface in the direction of the cut.
<i>Tool vector</i>	The leadin move is parallel to the tool vector at the point of the move.

Leadout Direction

This parameter selects the leadout move type.

<i>None</i>	No leadout move is made when this option is selected.
<i>Normal</i>	This option directs the tool to leadout to the surface along a normal to tool path at the pass start point. The move is perpendicular to the surface.
<i>Tangent</i>	The leadout move for this option is tangent to the surface in the direction of the cut.
<i>Tangent Angle</i>	The leadout move for this option is tangent to the surface, plus a defined angle, in the direction of the cut.
<i>Fixed Angle</i>	The leadout move for this option is at a fixed angle to the surface in the direction of the cut.
<i>Tool vector</i>	The leadout move is parallel to the tool vector at the point of the move.

Degree

If the NURB spline uses Uniform knots, then the maximum degree is 20 and the minimum degree is 1. The minimum number of control points in a Uniform NURB spline is the degree of the NURB spline plus one. If the NURB spline uses Bezier knots, then the degree is automatically the number of points in the controlling polygon minus one, and this value is ignored. Since the maximum degree is 20, the largest Bezier spline that can be created will contain 21 control points.

This specifies the complexity of the equations defining the NURB spline. The higher the degree, the number of points that can be interpolated by a single segment increases. Also, with a higher degree the spline tends to stay away from the control polygon (considered smoother). Higher degree splines have problems of different kinds (numerical instability and oscillations). In general many geometric modeling applications limit the degree of the splines to be 3 although degrees of 5 through 8 are not uncommon. SURFCAM limits the degree of a NURB spline to 20.

A NURB surface has two degrees; one along the U direction and the other along the V direction.

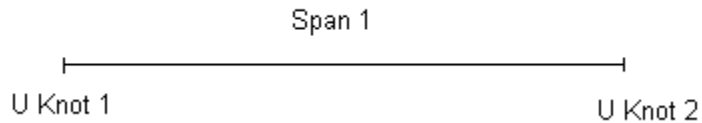
The degree of a NURB spline must be at least one less than the number of control points. In the case of a NURB surface, the degree in each direction must be at least one less than the number of control points in that direction. A NURB spline of degree 1 can be used to represent lines and polylines. SURFCAM splines can be accurately represented by NURB splines of degree 3.

Knots

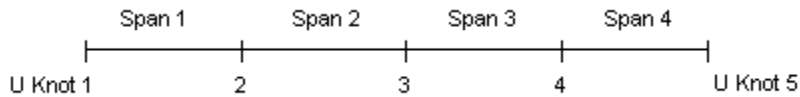
Nurb splines are composed of distinct *spans*. Each span is defined by a single mathematical equation. Knots are the transition points between spans.

Knots are of three types: Bezier, Uniform, and Non-uniform. Non-uniform knots will usually give the smoothest results. Use the others only if you have a specific need.

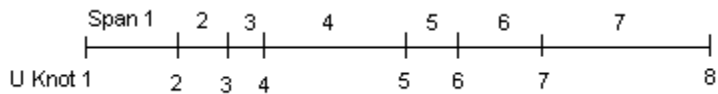
Bezier There is only one span (two knots).



Uniform All spans are of equal length. You can have an unlimited number of spans.



Non-uniform Spans are different lengths. The length is calculated from the control points so as to keep the speed of the curve as uniform as possible. You can have an unlimited number of spans.



NURB Curve

Polygon

Will create a NURB spline with or without displaying the controlling polygon. Select *Yes or No* to display or not display the polygons.

NURB splines are defined by a set of points in space. Unlike splines where the spline passes through the points that define the spline, the points defining the NURB spline (except the first and last) do not usually lie on the spline. Sometimes they are called puppet points, as each controls a piece of the spline and moving one point causes a portion of the spline to move in that direction.

A NURB surface is defined by a rectangular mesh of control points.

Alternate Tool Path Colors

Yes will cause the tool path to be drawn with alternating colors. Every other move will alternate between the primary tool path color (as shown in the status menu), or the secondary color, which is the next higher number color in the color selection pop-up.

Draw Rapid Moves

You may select Yes to display the rapid moves. When this is set to No, the rapid moves will not be visible.

Surface Arrow Options

NC Edit / NC Insert Menu

<i>Insert</i>	SURFCAM presents a sub menu; selecting one of these options allows you to insert one of these types of INC records at the current editing location in the INC file. Following is a brief description of the types of records that you may insert:
<i>Rapid, Line, Plunge</i>	When you choose one of these, you will be able to indicate a move, of the type chosen, to a location that is specified from the Location menu.
<i>Hole</i>	When hole is selected the system prompts for the hole location. After selecting the hole location, enter the hole depth.
<i>End row</i>	This is a simple record that marks the end of a pass of the tool across the surface or pocket. It requires no other input.
<i>Feed Rate or Plunge rate</i>	Insert one of these records to alter the feed or plunge rate for subsequent movements in the INC file. Choosing either of these will cause SURFCAM to present a small pop-up window that prompts for a new rate.
<i>Tool change</i>	This will allow you to insert a tool change into the INC file; SURFCAM will present both the Tool Information and Program Information pop-up windows.
<i>Comment</i>	Comments are similar to Inline, except that the comment start and end characters will be added as well. Verify that proper beginning and ending comment characters are defined in the POSTFORM file.
<i>InLine</i>	SURFCAM supports direct insertion of inline machine coding by using these choices in the insert mode of NC Edit. Anything typed in response to the InLine pop-up will appear on its own line in the final NC program exactly as typed, except that an N number may be added at the beginning by the post processor.
<i>NC Mode</i>	Allows you to select NC mode: Lathe, Wire EDM, 2 axis, 3 axis, 4 axis, or 5 axis.

Create / Spline Menu

A spline is a single element that joins a series of points or locations.

<i>Points</i>	This option allows you to create a natural cubic spline by indicating the locations of the node points. When SURFCAM is used to create a spline in this manner, the resulting spline is called a natural spline which means that the curvature is zero at the ends of the spline.
<i>Elements</i>	Using this option, you can create a spline from a series of chained lines, arcs, and even other splines.
<i>Optimized</i>	This option creates an optimized spline by duplicating an existing spline, then removing the unneeded nodes from it. The new spline created will only use the nodes that are needed to hold a tolerance, specified by you, to the original spline, reducing the amount of data and time that is required to represent curves and surfaces.
<i>Ellipse</i>	An elliptical spline may be created using this option.
<i>Spri hlx</i>	This option allows you to create a spiral spline, helical spline, or helical spiral spline.
<i>Surf. flow</i>	When a surface is created SURFCAM creates surface flow splines. These create the boundaries of the surface. This option allows you to create additional surface flow splines in either direction along the surface.
<i>X-section</i>	The X-section (cross-section) spline is created by slicing surfaces (single, multiple, or composite) at defined intervals relative to the CView. The plane of the sections is perpendicular to the CView. When the CView is set to zero, the current view is used.
<i>Surf. Int</i>	This option allows you to create a spline along the intersection of two surfaces.
<i>Cutter Int</i>	A Cutter Int spline is offset from the intersection of two surfaces such that it defines the center of a cutter.
<i>Project</i>	This option allows you to project a spline onto a surface. The spline must be located in space relative to the location of the surface. The projections is accomplished either normal to the surface or through the construction view.
<i>Blend</i>	This option is used to create a smooth NURBS spline between two existing curves.

Create / Surface Menu

In SURFCAM, a surface is a mathematical representation of a part's outer or inner shape. Surfaces have a positive and a negative side to them, typically indicated by a surface arrow. The direction of the surface arrow indicates the side, and the cutter path direction of the surface to be machined. A surface is displayed using U and V splines. The area bounded by the U and V splines is a surface patch, and the splines are the patch boundaries.

<i>Points</i>	A surface created directly from point data or a linemesh into a surface.
<i>Cross sect</i>	A surface created by selecting a series of cross sections in 3D space that describe the shape of the desired surface.
<i>Drive Curve</i>	A surface created by sweeping one or more 2D or 3D cross sections along a curve. Supports an optional twist-directing curve for additional control.
<i>Offset</i>	A surface created by offsetting normal to an existing surface.
<i>Fillet</i>	A surface that is a constant or variable radius fillet between two existing surfaces.
<i>Composite</i>	Trims and links together existing surfaces into a single composite surface.
<i>Primitives</i>	This option provides a set of different basic surface features such as a cylinder, plane, sphere, etc.
<i>Revolution</i>	A surface created by rotating a profile around an axis or vector.
<i>Extrude</i>	A surface created by extending a profile along a defined direction.
<i>Blend</i>	Creates a smooth, tangent surface between 2 - 4 curves or surfaces.
<i>Trimplane</i>	A flat surface that is automatically trimmed to the edge of a surface or curve.
<i>Options</i>	A pop-up window containing surface creation tolerances and options.

Thread Information

Edit / Spline Menu

SURFCAM allows you to change the attributes of splines including changing the locations of the nodes, the tangent vector directions and magnitudes.

- | | |
|-------------------|--|
| <i>Node/Tvect</i> | Choosing Node/Tvect from the menu will result in another menu that gives control over the node and tangent vector information for splines. |
| <i>Optimize</i> | This option creates an optimized spline, and is similar to the Optimized choice on the Create Spline menu. The original spline is replaced by the optimized spline. Multiple spline may be optimized at the same time. |
| <i>Untrim</i> | This option returns a spline that has been trimmed to its original untrimmed state. |
| <i>Polygon</i> | This option is used to create a polygon shape for a spline. |
| <i>Explode</i> | This option is used to convert NURB splines to either parametric splines or polylines. |

Point

Select *Yes* or *No* to create points projected onto the spline or surface selected.

Normal

Select *Yes* or *No* to project lines normal to the surface or spline at the location selected.

U Tangent

Select *Yes* or *No* to project lines tangent to the U direction of the spline or surface.

V Tangent

Select *Yes* or *No* to project line tangent to the V direction of the surface.

U Curvature

Select *Yes* or *No* to project lines along the U curvature of the spline or surface.

V Curvature

Select *Yes* or *No* to project lines along the V curvature of the surface.

U 1st Derivative

Select *Yes* or *No* to project lines along the 1st derivative of the U direction on the spline or surface.

V 1st Derivative

Select *Yes* or *No* to project lines along the 1st derivative of the V direction on the surface.

U 2nd Derivative

Select *Yes* or *No* to project lines along the 2nd derivative of the U direction on the spline or surface.

V 2nd Derivative

Select *Yes* or *No* to project lines along the 1st derivative of the V direction on the surface.

U Cross V

Select *Yes* or *No* to project lines representing the cross product along the U and V direction of the surface.

Convert To

Select the desired entity type to convert the NURB spline to. You may select *Spline* or *Polyline*. *Spline* converts the NURB spline to a parametric spline.

Create / Arc Menu

Direction

Select Normal or CView for the direction to project the points.

Path & Island Depths

This option can be set to either Constant, Multiple, or Roughing 3D. Choose constant to have all chained islands and paths avoided at all cutting levels. When the islands, or other paths to avoid, are at different heights, choose multiple. When Multiple is selected, the islands must be drawn at their actual height, Z position, on the part. The pocketing and contouring tool motion will avoid them once the cutting level comes below or at the same Z height as the path or island.

The Roughing 3D option may be used in 2 axis cutting to perform roughing of 3D surfaces.

Align

You may choose toward or away from a point or a vector.

Align

You may select a *Point* or *Vector* to align the surfaces . To use a point, you may not need to create a reference point if there is already some location (element endpoint, center, etc.). To use a vector, you are prompted to indicate two locations. Again, you may or may not need to create reference geometry, depending on the elements available in the drawing. Usually a line is good to use as a reference vector.

NOTE: The location of the surface corner with the arrow is used to determine which side the arrow should point towards. For complicated geometry, you may need to do the alignment in more than one step.

Align Options

Radius At This Point

Enter the desired fillet radius at the point selected.

Start

You may select the blend type for the ends of the surface. This affects the change of the radius at the ends of the curve. SURFCAM displays a pop-up for you to select the desired start and end blend type for the surface.

Blended Blended attempts to keep a constant radius at the end of the fillet and is the recommended choice for closed reference curves. These are curves that start and end at the same location. This helps match the two ends of the fillet surface.

NOTE: SURFCAM automatically determines Blend or Linear default parameters.

Linear Linear specifies a constant rate of change in radius at the ends of the reference curve and generally yields better results in fillet surfaces that are not closed.

End

You may select the blend type for the ends of the surface. This affects the change of the radius at the ends of the curve. SURFCAM displays a pop-up for you to select the desired start and end blend type for the surface.

Blended Blended attempts to keep a constant radius at the end of the fillet and is the recommended choice for closed reference curves. These are curves that start and end at the same location. This helps match the two ends of the fillet surface.

NOTE: SURFCAM automatically determines Blend or Linear default parameters.

Linear Linear specifies a constant rate of change in radius at the ends of the reference curve and generally yields better results in fillet surfaces that are not closed.

Limit Rotary Motion

This parameter is used to set 3 or 4 axis motion. When 4 is selected, three linear and one rotary axes are used to cut the surface. When 3 is selected, two linear and one rotary axes is used dependent upon the axis of rotation. When the axis of rotation is on the X axis, no Y axis motion occurs. When the axis of rotation is along the Y axis, no X axis motion occurs. When the axis of rotation is not along any standard axis, all 4 axis motion may occur.

3 Axis When 3 axis is selected the operation is performed with the tool tip pointing through the axis of rotation. The side angle entry, in the Tool Information pop-up, has no effect on the tool angle. The tool is always pointing through the axis of rotation.

4 Axis During the 4 axis simultaneous motion, the tool orientation is often normal to the surface.

NC / NC Mode Menu

The *NC* option allows you to access machining functions that produce the instructions needed to machine a part. Functions are available for producing instructions for the following types of machines: lathe, wire EDM, 2 axis, 3 axis, 4 axis, and 5 axis. In SURFCAM these instructions are saved in files called Intermediate Numerical Control or INC files.

When the *NC* option is selected from the *Main* menu, SURFCAM displays the current *NC* menu which could have any of the following titles: *NC Lathe*, *NC EDM*, *NC 2 Axis*, *NC 3 Axis*, *NC 4 Axis*, or *NC 5 Axis*. Each of these *NC* menus has an option called *NC mode* which is used to switch to a different *NC* menu. The default *NC Mode* can be changed by using the Configuration Utility.

In the current SURFCAM session, you can change the *NC mode* by selecting the mode from the *NC Mode Menu*.

Surface Parametric Tolerance

This is the tolerance within which the surface will deviate from the cross sections. It is used only for the *Cross Section*, *Drive Curve*, and *Blend* types of surface generation. The default value is 0.00100 inches.

Delete / Delete Layers Menu

The *Delete layers* option allows you to delete unwanted layers that you created.

Deleting a layer deletes all of the elements on that layer along with the layer. If the deleted layer is not visible, SURFCAM displays a message telling you so. Deleted layers can be undeleted by using the Undelete Last or Undelete All functions provided you have not created a new layer with the same number.

A box is displayed on the screen showing the different layers. A layer or view that has been selected with the mouse will show the letter D in the first column to indicate that it will be deleted when Done is selected. Picking any line in the box toggles the D in the first column.

- | | |
|-----------------|--|
| <i>Delete</i> | This option allows you to select a range of layers. A pop-up is displayed for you to enter the beginning and ending layer numbers to delete. After accepting these values, the letter D is displayed in the first column for the layer box for the layer numbers entered. This is an inclusive list. This option is not used to delete non consecutive layers. |
| <i>Undelete</i> | This option is used to undelete a range of layers. The pop-up for the beginning and ending layer numbers is displayed. The letter D is removed from the layer numbers entered. This option is not used for non consecutive layers. |
| <i>Swap D</i> | When this option is selected the letter D for the layers marked for deletion are removed and displayed for the layers that are not marked for deletion. The letter D toggles each time this option is selected. |
| <i>Done</i> | When the desired layers are marked correctly select this option to delete the marked layers. These layers may be undeleted using the Undelete Last or Undelete All options on the Delete menu. |

Range of Views

Beginning View Number

Enter the first view number to delete or undelete.

Ending View Number

Enter the last view number to delete or undelete.

Delete / Delete Views Menu

This option allows you to delete unwanted views that you created. The eight system views should not be deleted. When this option is selected the menu below is displayed.

- Delete* This option allows you to select a range of views. A pop-up is displayed for you to enter the beginning and ending view numbers to delete. After accepting these values, the letter D is displayed in the first column for the view box for the view numbers entered. This is an inclusive list. This option is not used to delete non consecutive views.
- Undelete* This option is used to undelete a range of views. The pop-up for the beginning and ending view numbers is displayed. The letter D is removed from the view numbers entered. This option is not used for non consecutive views.
- Swap D* When this option is selected the letter D for the views marked for deletion are removed and displayed for the views that are not marked for deletion. The letter D toggles each time this option is selected.
- Done* When the desired views are marked correctly select this option to delete the marked views. These views may be undeleted using the Undelete Last or Undelete All options on the Delete menu.

Variable Fillet Options

NC Edit / Delete NC Menu

<i>Delete</i>	When <i>Delete</i> is selected from the <i>Edit NC</i> menu, SURFCAM displays the <i>Delete NC</i> menu. This allows you the option of deleting the current INC record by choosing <i>Current</i> or choosing from several methods of deleting a series of INC records. If any of these options is chosen, all records are deleted from the present move to the item picked from the menu. The menu choices are described below.
<i>Current</i>	Deletes the current record or move only.
<i>Start</i>	Deletes all NC data from the beginning of the INC file to the current position
<i>Previous</i>	Deletes the current and the previous moves.
<i>Pick</i>	Picking an item from the display, by using the mouse, deletes the NC data from the current location up to the selected move.
<i>Next</i>	Deletes the current and the next moves.
<i>End</i>	Deletes all NC data from the current position to the end of the INC file.

Tool Information (Title)

Tool Library

This is presented so you can easily review the tools and pick the one desired. As the mouse is moved horizontally through the columns in the pop-up, the prompt line shows what type of information is in the columns. To scroll through the tool library, use the up or down arrow keys or the Page Up and Page Down keys. The mouse may also be used to page through the library. Place the arrow above (page up) or below (page down) the tool pop-up box. Press the left mouse button to page through the list.

After selecting a tool from the tool library pop-up, either by moving the cursor to the line that contains the tool and clicking the left mouse button or by entering the reference number from the keyboard, you are returned to the *Tool (Drill) Information* pop-up.

NOTE: The SURFCAM.TOL file can be modified using the **SURFCAM Configuration Utility**.

Tool Corner Radius

Specifies the tip radius of a tool. In the case of a ball end mill, this is the tool radius; for a bullnose cutter, this is the corner radius.

Tool Total Radius

Specifies the larger radius of a bullnose cutter. If the cutter total radius is greater than the tip radius, SURFCAM uses a bullnose cutter.

Tip or Center

Normally, the NC code produced will drive the *Tip* of the cutter. When using a ball end mill and the tool path for the ball center is desired, choose *Center*.

Side Angle

This allows you to specify an angle for shank clearance. The side angle may also be used for an undercut. It also allows you to adjust for tapered tools during swarf cutting. The tool taper angle is input for *Side Angle* to maintain the tool shaft parallel to the surface. A larger angle can be used when shank clearance is desired for these tools as well.

NOTE: A positive angle value orients the tool shaft away from the surface.

Curve Tolerance

SURFCAM will break all contour moves along a spline into straight linear movements. When driving the tool around a spline curve, SURFCAM will adjust the length of the cutting moves to make sure that the edge of the contour is machined within this tolerance value.

Minimum Z Value

This parameter sets the most negative Z axis move allowed. The tool path will not move below this level. The tool path maintains this Z level along the contour until the end of the contour or a higher Z level is intersected.

Surface Section (Title)

Increment

Enter the desired distance between the cross section splines.

Rough Depth of Cut

The amount of material to remove for each roughing pass. This amount does not effect the Finish operation.

Follow

Sets the roughing pattern. When set to *Yes*, the roughing pattern follows the contour of the chained elements between roughing passes. When set to *No*, the roughing pattern steps in on the appropriate cutting angle for each pass.

Angle Length (Title)

Retract

Enter the angle for the retract moves. The angles are determined from the *CView* of the geometry.

Leadin Angle

Sets the angle for all leadin moves. The leadin move precedes the starting cut position. These angles are determined by the current *CView*.

Leadin Length

This parameter sets the length of the leadin move.

Leadout Angle

Sets the angle for all leadout moves. The leadout move follows the end of the cut. These angles are determined by the current CView.

Leadout Length

This parameter sets the length of the leadout move.

Minimum Z Value

When you want to limit the lowest Z value cut, enter that value here. If there is nothing on this line, there will be no limit imposed in the NC code.

Lead IN OUT

This parameter is the distance the cutter feeds into the surface when a leadin move is specified.

Leadin Move Length

Enter the desired Leadin move length.

Leadout Move Length

Enter the desired Leadout move length.

Leadin Direction

This parameter selects the leadin move type.

<i>None</i>	No leadin move is made when this option is selected.
<i>Normal</i>	This option directs the tool to leadin to the surface along a normal to tool path at the pass start point. The move is perpendicular to the surface.
<i>Tangent</i>	The leadin move for this option is tangent to the surface in the direction of the cut.
<i>Tangent Angle</i>	The leadin move for this option is tangent to the surface, plus a defined angle, in the direction of the cut.
<i>Fixed Angle</i>	The leadin move for this option is at a fixed angle to the surface in the direction of the cut.
<i>Tool vector</i>	The leadin move is parallel to the tool vector at the point of the move.

Leadout Direction

This parameter selects the leadout move type.

<i>None</i>	No leadout move is made when this option is selected.
<i>Normal</i>	This option directs the tool to leadout to the surface along a normal to tool path at the pass start point. The move is perpendicular to the surface.
<i>Tangent</i>	The leadout move for this option is tangent to the surface in the direction of the cut.
<i>Tangent Angle</i>	The leadout move for this option is tangent to the surface, plus a defined angle, in the direction of the cut.
<i>Fixed Angle</i>	The leadout move for this option is at a fixed angle to the surface in the direction of the cut.
<i>Tool vector</i>	The leadout move is parallel to the tool vector at the point of the move.

Surface Generation Tolerances

Drive Curve Options

Cross Section Orientation

You may select either 2D or 3D orientation for the cross sections used for Drive Curve surfaces.

Directing Curve Selection

You may create a Drive curve surface using a directing curve. This is another curve that directs the rotation of the cross sections about the drive curve and tilt along the drive curve. The tilt is the attachment angle of the cross section to the drive curve. A surface with normal tilt attaches the cross sections perpendicular to the drive curve.

The directing curve is used to manipulate the surface rotation and tilt when using 2D cross section geometry. When using 3D geometry the orientation of the geometry affects the surface. The 3D cross sections are not rotated or tilted with the directing curve; however, the surface between the cross sections may be affected.

The Directing curves can each be either a single element such as a line, Arc, or Spline, or a smooth chain of any combination of these.

The default setting in the Create Surface Options for the directing curve is No.

The directing curve rotates the cross sections around the drive curve placing the Y axis of the cross section parallel to a line connecting the drive curve and directing curve.

When the tilt is normal the cross section is attached perpendicular to the drive curve. When the tilt is directed, the cross section X axis is parallel to a line connecting the drive and directing curves.

Yes Select this option to direct the cross sections.

No Select this option when the cross sections are not to be directed.

Surface of Revolution (Title)

Start Angle

Enter the desired start angle of the surface. The cross section location is zero degrees and the surface is created counterclockwise.

End Angle

Enter the desired end angle of the surface.

Feed rate

This option sets the method for feed on leadout moves from the plunge point. This feed rate may use the *Feed*, *Rapid*, or *Plunge* speeds.

Increment

Enter the incremental distance between splines.

Direction

Select the direction along the surface to create the splines. Major is the direction the surface arrow is pointing. Minor is opposite the surface arrow direction.

Isometric Curve

Thread Pitch

Enter the distance from one thread to the next. This may be entered as Thread per inch, Inches per thread or Millimeters per thread. Inches per thread is calculated as $1 / \text{number of threads per inch}$.

Thread Pitch Type

Select either Threads per inch (TPI) or Inches per thread (IPT).

Depth of Thread

The distance from the Major radius of the thread to Minor radius. Do not use the thread diameter.

First Cut Depth

This is the initial material removal amount per side.

Min Cut Depth

Indicates the minimum amount of material to remove for any pass.

Infeed Angle

Enter the desired angle for the infeed move.

This parameter is used to set the distance above the part to end the rapid move. It is also used as the minimum amount that the rapid plane must be above the part.

Corner Angle

Determines the tool path at each change of direction. When the interior angle of the directional change is larger than this value, the tool path is intersectional. Any directional change with the interior angle greater than 150 degrees is an intersectional tool path. Interior angles less than this value use either an Arc or Line to prevent the intersectional move. Angles smaller than 30 degrees automatically use either Arc or Line moves. The default for this value is 90 degrees.

Arc Uses an arc between directional changes that are less than the Corner angle value. The tool path uses the Corner clearance amount to determine the location of the arc.

Line Utilizes a line between directional changes that are less than the Corner angle value. The tool path uses the Corner clearance amount to determine the line location.

Corner Angle Move

- Arc* Uses an arc between directional changes that are less than the *Corner angle* value. The tool path uses the *Corner clearance* amount to determine the location of the arc.
- Line* Utilizes a line between directional changes that are less than the *Corner angle* value. The tool path uses the *Corner clearance* amount to determine the line location.

Corner Clearance

When the Corner angle cut is in effect, this value determines the tool clearance from the direction change intersection. When this value is zero, the tool remains in contact with the part walls during the directional change.

Leadin speed

This option sets the method for feed on leadin moves from the plunge point. This feed rate may use the Feed, Rapid, or Plunge speeds.

Leadout speed

This option sets the method for feed on leadout moves from the plunge point. This feed rate may use the Feed, Rapid, or Plunge speeds.

Lathe Drill Menu

New Tool Allows you to continue building the INC file with additional tools. The select procedure is repeated.

Done Select this option to complete the INC file. The NC Lathe menu is displayed.

NOTE: Drilling, contouring, and pocket cycles support rotary table indexing by utilizing *CView* machining. By specifying the correct *CView* and setting *Coords* to view, the appropriate indexing move will be generated prior to any tool motion. This requires the POSTFORM.M file to be properly configured to take advantage of indexing capabilities.

First Trim Value

Enter the desired value to trim from the first entity.

Second Trim Value

Enter the desired value to trim from the second entity.

Chamfer Menu

The chamfer option is used to create a chamfer between two lines or locations.

Chamfers may be created between two elements, two locations or an element and location. When elements are selected SURFCAM creates the chamfer based on the intersection point of the two elements. This method prompts you for the distance from this intersection point to create the chamfer. These points are referred to as the first and second trim distance.

*Change *chm** When this option is selected a pop-up is displayed for you to enter the desired first and second trim distances.

Trim This option allows you to trim 0, 1, or 2 of the elements used for the chamfer. The options are described below.

0 This option creates the chamfer without trimming any elements.

1 The first element selected is trimmed to the chamfer. The other element is not trimmed.

2 Both elements are trimmed to the chamfer.

Other This option is displayed after the chamfer is created. You may select this option to display all chamfer possibilities between the two elements.

Undo You may Undo the chamfer created with this option. This option is only available after the creation of the chamfer. When another menu selection or element is selected this option is not active or displayed. You may either click this option .

Stock Z

This option allows you to specify the amount of stock to leave on each cut axis. The axis designations described in the SURFCAM.CFG file are displayed. Enter the amount of stock to leave on the part for each axis.

The Operation selected sets the default values for the parameters below. You may override these values.

Cut Angle

This angle determines the direction of tool motion. An angle of 180 degrees causes the tool to cut from right to left.

Groove Side Step

The amount of material to remove for each roughing pass. The tool steps over this amount for each pass.

Finish Depth of Cut

This is the amount of material to remove for the finish pass.

Number of Passes

This is the number of finish passes to perform. The Finish depth of cut amount is used for each of these passes.

Spring Passes

Indicates the number of times the final finish pass is cut. The final finish pass may be repeated to remove material that may be left due to part deflection.

Operation Facing

The Face operation allows you to machine the part along the diameter of the material. This is typically accomplished with the cutting motion along the X axis. This axis designation may vary with specific machine tools. Below is a list of the available operation for the Face option.

OD Face

OD Back Face

ID Back Face

ID Face

NOTE: This section uses the X axis for diameters and the Z axis for lengths.

You may select either a rough, finish, or both cuts for each Face operation. When this option is selected you are prompted to select the beginning and ending elements to cut. You are then prompted for the optional material boundary beginning and ending elements. After entering the INC filename the tool information and program information pop-ups are displayed as described previously in this section.

Gouge Checking

Front and back angle gouge checking is automatically applied to all rough and finish tool paths. SURFCAM uses the user-defined angles from the cutting edge of the tool to perform gouge checking. These angles are set in the SURFCAM.TOL file. These angular values may not be overridden from within SURFCAM. To ensure proper gouge checking, the tool library must be updated to users actual tools.

NOTE: If you override any of the insert parameters such as tool nose radius the system will use the last set of angles from the last tool selected from the library.

Operation Grooving

Specifies the type of operation being performed. The operation type determines the tool orientation required. SURFCAM uses the tool orientation to ensure the proper tangent points are used. The list below indicates the operations available.

OD Groove

OD Back Groove

ID Back Groove

ID Groove

Face Groove

Back Face Groove

SURFCAM allows you to select elements to cut with the grooving operation. These elements may be either OD or ID elements.

The Groove operation allows you to rough and finish, with any groove cutting angle, along the diameter of the material. This is typically accomplished with the cutting motion perpendicular the spindle centerline. This axis designation may vary with specific machine tools.

NOTE: Rough Groove Cutting is the only option available.

Rough and/or Finish

You may select either a *Rough cut*, *Finish cut* or both with this option. Options that are not selectable are displayed in gray text. The gray options change with the selection of *Rough*, *Finish*, or *Both*. For *Turning*, *Facing* the following options are gray.

Rough

Groove Tool Width

Finish Depth of Cut

Number of Passes

Spring Passes

Finish

Groove Tool Width

Cut Angle

Rough Depth of Cut

Both

Groove Tool Width

Offset Side

Rough and/or Finish

Rough cut is the only option available for *Grooving*. Options that are not selectable are displayed in gray text. The following options are gray.

Groove Tool Width

Undercut

Finish Depth of Cut

Number of Passes

Spring Passes

The Groove Tool Width is gray until a Groove tool is selected from the Tool Library.

Tool Nose Radius

This parameter indicates the radius of the tool nose. You may override the tool nose radius.

Undercut

You may select *Yes or No* to turn undercut geometry. Select **Yes** to turn geometry that has an undercut. Select **No** to disregard undercut geometry.

Project Curve Information

Maximum Error

This entry sets the maximum deviation from the original spline shape.

Direction

Select either *CView* or *Normal*. *CView* projects the spline through the current CView setting. *Normal* projects the spline in the Z minus direction.

Extend Length

Enter the amount to extend the surface.

Dump

UV Output

XYZ Output

Points

Connect Points

Connect Type

Create / Points Menu

Create / NSurface / Menu

Create / Ncurve Menu

Radial Pitch

This parameter describes the amount of change in the radius for every 360 degrees when Spiral is selected.

Helical Pitch

This is the amount of height change in the spline for every 360 degrees when Helix is selected.

Helical Length

This is the overall height of the spline when Helix or Both is used.

Axial Direction

This is the direction to apply the Helical Length. This direction is based on the current construction view. You may select Towards or Away.

Away This is in the negative Z axis of the current construction view.

Towards This is in the positive Z axis in the current construction view.

Rotation Direction

This parameter defines the direction of the spline. You may select CW or CCW.

Ellipse Information (Title)

Spiral Helix Information (Title)

Rectangular Array Information (Title)

X Y

Points along X axis

Enter the desired number of points along the X axis.

Points along Y axis

Enter the desired number of points along the Y axis.

Distance along X axis

Enter the desired distance between the points along the X axis.

Distance along Y axis

Enter the desired distance between points along the Y axis.

Circular Array Information (Title)

Number of points

Enter the number of points in the circular pattern.

Radius

Enter the desired radius for the circular pattern.

Total Angle

This is the angle between the first and last point. A positive value creates the points in the counterclockwise direction. A negative value creates the points in a clockwise direction.

Start Angle

This is the angle from the horizontal to the first point in the circular array. A positive value produces a counterclockwise angle. A negative value produces a clockwise angle.

Angle

Enter the desired angle from 0 to 360 for the rows.

No. of Pens

This parameter defines the number of pens that the desired plotter uses. The default is 1.

Paper Size

You may select any standard size from the list below. Pressing the right mouse button displays the available sizes.

The paper sizes are defined by the letters below. The letter sizes A through E below are in inches.

A	8.5 by 11
B	11 by 17
C	17 by 22
D	22 by 34
E	34 by 44

The letter sizes a6 through a0 below are in millimeters.

a6	105 by 148
a5	148 by 210
a4	210 by 297
a3	297 by 420
a2	420 by 594
a1	584 by 841
a0	841 by 1189

Plotter Options

Plotter Tolerance

This is the tolerance used by the plotter to maintain the drawing sizes. The tolerances are in metric for millimeter sizes.

Circle Menu

The Circle function on the Create menu displays a submenu of different ways that you can create circles.

- | | |
|-------------------|--|
| <i>3 Points</i> | This option allows you to create a circle by specifying three points on the screen. These three points are selected using the Location menu. Each desired point may be selected with a different method from the Location menu. |
| <i>Center/Dia</i> | When using this method of circle creation, SURFCAM displays a pop-up for you to enter the diameter,. |
| <i>Center/Rad</i> | This option works exactly the same as the Center/Dia option except that SURFCAM prompts you for the Radius instead of the diameter. |
| <i>2 pnts rad</i> | This option allows you to create a circle by specifying two end points. The points are selected using the Location menu. Each desired point may be selected with a different method from the Location menu. The points selected define the radius of circle. The circle is created in the counterclockwise direction from the first point to the second point. |
| <i>2 pnts dia</i> | This option allows you to create a circle by specifying two end points. The points are selected using the Location menu. Each desired point may be selected with a different method from the Location menu. The points selected define the diameter of circle. The circle is created in the counterclockwise direction from the first point to the second point. |
| <i>Tangent 2</i> | You may create a circle tangent to two elements. When this option is selected the Circle menu is displayed for you to select the circle radius and trim options. |
| <i>Tangent 3</i> | You may create a circle tangent to three elements. When this option is selected you are prompted to select the three elements that are to be tangent to the circle. The elements selected are restricted to points, lines, arcs and circles. |
| <i>Offset</i> | This option allows you to offset a circle by a distance, to a location, or tangent to another entity. |

Ellipse Menu

An elliptical spline may be created using this option. When selected a menu is displayed.

- Axis* This selection displays the Location menu for you to select the center of the ellipse and points at the end of the major and minor axes. The point for the minor axis is projected to the true minor axis.
- Center* This option allows you to create an ellipse at an angle around a center point. Enter the major and minor diameters.

Major Diameter

Enter the length of the major, or long, axis of the ellipse in this parameter.

Minor Diameter

This parameter is the length of the minor, or short, axis for the ellipse.

Angle

This parameter allows you to create the ellipse at the specified angle.

Start Radius

This is the radius of the spline at the start point location.

Start Angle

This is the angle to the start point of the spline.

End Radius

This value is the radius of the spline at the endpoint. This parameter is not used when Both is selected for the type.

This parameter is used to set the distance above the part to end the rapid move. It is also used as the minimum amount that the rapid plane must be above the part.

Scale

Enter the desired scale factor for the display screen. The drawing is displayed at the scale factor entered. The drawing scale is not changed.

Max Cut Angle

Enter the maximum angle from the horizontal at which end cuts are to be made.

Surface Display Options

Operation Thread

The Thread operation may be used to program inside or outside canned cycle threads. This operation performs the rough and finish cuts for the desired thread.

Operation Specifies the type of operation being performed. The operation type determines the tool orientation required. SURFCAM uses the tool orientation to ensure the proper tangent points are used. The list below indicates the operations available.

OD Thread

OD Back Thread

ID Back Thread

ID Thread

Chamfer Out Threads

Enter the number of threads from the last thread to lift the tool out of the part.

Spring Passes

This indicates the number of times the final finish pass is re-cut. The final finish pass may be repeated to remove material that may be left due to part and or tool deflection.

Leadin Length

Enter the distance for the approach to the first thread. This value may be entered as threads, inches or millimeters.

Leadin Type

This value may be entered as threads, inches or millimeters.

Leadout Length

Enter the distance to overrun the last thread to insure a complete thread. This value may be entered as threads, inches or millimeters.

Leadout Type

This value may be entered as threads, inches or millimeters.

Menu Item 526

Menu Item 527

Menu Item 528

Leadin Type

There are four type of leadin moves: 3D, 2D, Undercut 3D, and Undercut 2D. When either 3D is selected the move may be along any axis. When either 2D is selected the move is only along the XY axes. When either undercut option is selected the move is made only when there is an undercut possibility.

Leadout Type

There are four type of leadout moves: 3D, 2D, Undercut 3D, and Undercut 2D. When either 3D is selected the move may be along any axis. When either 2D is selected the move is only along the XY axes. When either undercut option is selected the move is made only when there is an undercut possibility.

Feed rate

This option sets the method for feed on leadin moves from the plunge point. This feed rate may use the Feed, Rapid, or Plunge speeds.

Feed rate

This option sets the method for feed on leadout moves from the plunge point. This feed rate may use the Feed, Rapid, or Plunge speeds.

Lead On Feed Between

When this option is selected as Yes, the leadin move is performed at each feed between move. When No is selected the leadin and leadout moves are only at the beginning and end of the surface cut.

Leadout Type

There are four type of leadout moves: 3D, 2D, Undercut 3D, and Undercut 2D. When either 3D is selected the move may be along any axis. When either 2D is selected the move is only along the XY axes. When either undercut option is selected the move is made only when there is an undercut possibility.

NOTE: When a leadout move is desired, it is recommended that the tool vector type be used for 3 axis machining. This option allows no rotary motion for the leadout move.

Leadin Angle On Tangent Plane

Enter the desired angle to rotate the leadin move along the tangent plane around the normal vector. This parameter is only used when the leadin move is Tangent Angle. Positive angles are counterclockwise.

Leadout Angle On Tangent Plane

Enter the desired angle to rotate the leadout move along the tangent plane around the normal vector. This parameter is only used when the leadin move is Tangent Angle. Positive angles are counterclockwise.

Leadin Angle From Tangent Plane

Enter the desire angle to rotate the leadin move from the Tangent plane. This parameter is only used when the leadin move is Tangent Angle. Positive angles are counterclockwise.

Leadout Angle From Tangent Plane

Enter the desire angle to rotate the leadout move from the Tangent plane. This parameter is only used when the leadin move is Tangent Angle. Positive angles are counterclockwise.

Tangent 2 Circle

This menu allows you to set the radius of the tangent circle and the trim elements.

Change rad Change the circle radius.

Trim Indicate the number of elements neighboring the new circle that are to be trimmed. You may select 0, 1, or 2. This option toggles between the choices when it is clicked.

 0 This option creates the circle without trimming any of the elements.

 1 The first element selected is trimmed to the circle. The other element is not trimmed.

 2 Both elements are trimmed to the circle.

Location Use this option if you want to create a circle tangent to an element and passing through a given point. The *Location* menu is displayed allowing you to select the point.

The following options are displayed ONLY after the circle has been created.

Other Display other tangent possibilities.

Reverse Reverse the trimming of the elements. This option is only displayed when either *Trim 1* or *Trim 2* is selected.

Undo This allows you to return the elements to the original configuration and remove the created arc. All elements are untrimmed to the original size.

Menu Item 540

Create New One

Enter Password String Obtained From Surfware, Inc.

Enter the entire password received from Surfware . It should be entered exactly as typed on the FAX. When the password has more than one line of characters, it must be entered on two lines. Be sure that the computers clock is set correctly before entering the password. Passwords must be entered in order . When they are entered out of sequence the message *Incorrect Password* is displayed.

Password entry line 1

Enter the password received from Surfware. This entry may require more than one line. Passwords received from Surfware **MUST** be entered in the order that they are created. The Reference Code is provided for technical support reference. Record the Reference Code number for technical support assistance.

Password entry line 2

Enter the balance of the password received from Surfware. Passwords received from Surfware **MUST** be entered in the order that they are created. The Reference Code is provided for technical support reference. Record the Reference Code number for technical support assistance.

Cut

This angle determines the direction of tool motion. An angle of 180 degrees causes the tool to cut from right to left. This parameter is for roughing only.

Tool Animation Wait

This parameter is measured in seconds. This is the amount of time between rotations of the tool animation.

Shade Menu

This menu allows you to shade the desired surfaces.

Select Allows you to select the desired surfaces to shade.

Shade The shading is performed when this option is selected.

Undercut After you have chosen the surfaces, select the *Undercut* option and SURFCAM will begin to render the surfaces. The difference between this option and the *Shade* option is that *Undercut* shades in different colors representing machinable and non-machinable areas of surfaces. The side evaluated is defined by the surface arrow. This is done in reference to 3 axis machining capabilities.

Options This option allows you to set the desired shading parameters.

Shade Options

Sort Type

This parameter allows you to select Full or Quick shading.

Quick This option is used for shading of non-intersecting surfaces. Use this option for single surfaces or simple surface models.

Full This option is used for more accurate shading of intersecting surfaces or complicated surface models. Although, the time required is longer than *Quick* you will receive the best results with this option.

Shade Color

This parameter allows you to select Single or Multiple for the shading colors.

Single This option overrides the surface color and uses a single color with the maximum number of possible shades.

Multiple This option uses multiple colors for the shading. The colors are determined by the surface colors.

Single Color

This is the color used when Single is selected.

Ambient Light

This parameter defines the strength of the ambient light or brightness of the background light. Choose a number between 0 and 1.

Material Finish

This parameter defines the type of material finish desired. Choose a number between 0 and 1.

Light Source Location X

Enter the X axis location for the light source.

Light Source Location Y

Enter the Y axis location for the light source.

Light Source Location Z

Enter the Z axis location for the light source.

Surface Tolerance

Enter the desired tolerance to use to maintain the surface shape.

Sampling Curves

Enter the desired number of sampling curves to create the shading.

Menu Item 559

Gouge Check

Select either *Full* or *Single* to perform 2D gouge checking. This may be answered *Single* when contouring intersecting elements.

Plunge Rate

This option sets the speed for the plunge moves. This feed rate may use the Feed rate, Rapid, or Plunge speeds.

Feed rate

This option sets the method for feed on leadout moves from the plunge point. This feed rate may use the *Feed*, *Rapid*, or *Plunge* speeds.

Plunge speed

This option sets the speed for the plunge moves. This feed rate may use the Feed, Rapid, or Plunge speeds.

Feed Between Rate

This option sets the speed for the feed between moves. This feed rate may use the Feed rate, Rapid, or Plunge speeds.

Feed Between Rate

This option sets the speed for the feed between moves. This feed rate may use the Feed rate, Rapid, or Plunge speeds.

Menu Item 566

Menu Item 567

Comment

Enter the desired comment for the tool. The post processor places this comment in the program for the Comment variable.

Work Offset

Enter the work offset number for the operation. This value is used by the post processor to define the desired fixture offset register.

Spline Tolerance

This parameter specifies the tolerance, that the cross section splines will be to the original surfaces .

Comment

Enter the desired comment for the tool. The post processor places this comment in the program for the Comment variable.

Comment

Enter the desired comment for the tool. The post processor places this comment in the program for the Comment variable.

Depth Location

Select either *Yes* or *No* to determine the construction view depth with the location menu

Depth Value

Enter the desired construction view depth.

Stock To Leave

Enter the desired amount of material to leave as stock for a subsequent machining operation. A negative amount of stock may be specified to undercut the shape.

Menu Item 576

Polygon

Select Yes if you want the NURBS surface display polygon to be drawn along with the surface.

Menu Item 581

Polygon

Select Yes if you want the NURBS curve display polygon to be drawn along with the curve.

Shade Type

This parameter allows you to select Quick or Smooth shading.

Quick This option is a polygon based.

Smooth This option is a pixel based shading method and is recommended for optimal image quality.

Menu Item 585

Menu Item 586

Menu Item 587

Menu Item 588

Menu Item 589

Menu Item 590

Menu Item 591

Menu Item 592

Menu Item 593

Turret Number

Enter the desired turret for the current lathe machining operation.

Menu Item 595

Menu Item 596

Direction

This parameter allows you to select either clockwise or counter clockwise direction for construction of the element.

Menu Item 598

Type

With this option you can create the blended surface or spline using a tolerance when exact tangency is not possible.

<= Tolerance With this option you can define the angular tolerance for blending the surface. A tolerance is required when there is not enough distance between the two surfaces in which curvature can be created. A common circumstance occurs when the edges of the surfaces touch.

Exact This option uses the tangency of the surfaces selected.

Ruled The ruled type creates a linear (flat) surface between the chosen elements. This is only available when two surfaces are chosen.

NOTE: SURFCAM automatically recommends the best *Type*.

Angle Tolerance

This option is only available when the *Type* option is set to \leq *Tolerance*. This parameter specifies the angular tolerance within which a blended surface will be tangent to the original surfaces when the \leq *Tolerance* blend type is selected.

1st Edge Fullness

Enter the desired fullness factor for the first surface selected. The fullness factors from the surfaces edge determine the final blended surface shape.

2nd Edge Fullness

Enter the desired fullness factor for the second surface selected. The fullness factors from the surfaces edge determine the final blended surface shape.

Menu Item 603

Check Points

Enter the number of points to analyze the surface edges.

Angle Tolerance

This parameter specifies the angular tolerance within which the surface edges will be tangent when the \leq *Tolerance* type is selected.

Menu Item 606

Major Direction

Enter the number of points in the major surface direction to analyze the surface distances.

Minor Direction

Enter the number of points in the minor surface direction to analyze the surface distances.

Tolerance

This parameter specifies the tolerance, that the surfaces will be tangent, when the greater than tolerance type is selected.

Menu Item 610

Create Points

This parameter allows you to select None, Maximum, Minimum or > Tolerance surface distance analysis.

None This option displays the maximum and minimum distances of the surfaces selected.

Maximum This option creates points on the surfaces selected at the maximum distance check point.

Minimum This option creates points on the surfaces selected at the first minimum distance check point.

> Tolerance This option creates points on the surfaces selected at the check points that are greater than the specified tolerance.

Create Normals

This parameter allows you to select *None*, *Maximum*, *Minimum* or *> Tolerance* surface edge analysis.

None This option displays the maximum and minimum angles of the surfaces selected.

Maximum This option creates lines on the surfaces selected at the maximum angle check point.

Minimum This option creates lines on the surfaces selected at the first minimum angle check point.

> Tolerance This option creates lines on the surfaces selected at the check points that are greater than the specified angle tolerance.

Element Type

This parameter allows you to select Points, Lines, Polylines, Linemesh, Splines, Ncurves, Ncurves w/ctrl, Nsurface, Nsurface w/ctrl.

<i>Points</i>	This option creates points from the digitized data.
<i>Lines</i>	This option creates lines from the digitized data
<i>Polylines</i>	This option creates polylines from the digitized data.
<i>Linemesh</i>	This option creates a linemesh from the digitized data.
<i>Splines</i>	This option creates splines from the digitized data.
<i>Ncurves</i>	This option creates Ncurves from the digitized data.
<i>Ncurves w/ctrl</i>	This option creates Ncurves using the digitized data as control points.
<i>Nsurface</i>	This option creates a Nsurface from the digitized data.
<i>Nsurface w/ctrl</i>	This option creates a Nsurface using the digitized data as control points.

NOTE: The Nsurface w/ctrl and Ncurve w/ctrl create a shape with only the endpoints exactly matching the digitized data. The remaining points influence the shape based on the degree specified.

Menu Item 614

Menu Item 615

Menu Item 616

1st End Fullness

Enter the desired fullness factor for the first spline end selected. The fullness factors from the splines end determine the final blended spline shape.

2nd End Fullness

Enter the desired fullness factor for the second spline end selected. The fullness factors from the splines end determine the final blended spline shape.

Menu Item 619

Tangent

Select either *Yes* or *No* to blend the edges of two surfaces or splines being joined.

Menu Item 621

Edge Tangency

Select the type of edge tangency for the two joined elements. Select Average Both to use the average tangency between both surfaces or splines. Select First Element to create the join tangent to the first element only.

Menu Item 623

Edge/Corners do not meet.

Select either *Yes* or *No* to continue the surface or spline edit.

Type

With this option you can create the blended surface or spline using a tolerance when exact tangency is not possible.

<= Tolerance With this option you can define the angular tolerance for blending the surface. A tolerance is required when there is not enough distance between the two surfaces in which curvature can be created. A common circumstance occurs when the edges of the surfaces touch.

Exact This option uses the tangency of the surfaces selected.

Ruled The ruled type creates a linear (flat) surface between the chosen elements. This is only available when two surfaces are chosen.

NOTE: SURFCAM automatically recommends the best *Type*.

Angle Tolerance

Enter the angle within which the joining of the surfaces/splines can deviate from exactly tangent.

Menu Item 627

Menu Item 628

Trim/Break Direction

Select either the Major (along the arrow) or Minor (away from the arrow) direction along the surface to break or trim.

Project Direction

Select the direction of the projection on the surface or spline. This may be either normal to the surface or in the construction view.

Menu Item 631

Option

Select either *Extend* or *Trim* to extend or trim the surface or spline. The original surface or spline is replaced by the trimmed or extended element.

Length

Enter the desired distance to extend or trim the surface or spline.

Type

Select the type of trim or extension to perform. You may select any of the following:

- | | |
|-------------------|---|
| <i>Tangent</i> | Extends tangent to the element (continuous in first derivative). |
| <i>Curvature</i> | Extends along the curvature of the element (continuous in first and second derivatives). |
| <i>Continuity</i> | Extend continuous to the element (continuous in first through third derivatives). |
| <i>Full</i> | Extend completely continuous to the element (continuous in all derivatives). |
| <i>Parameter</i> | This option is available only if the curve or surface was previously trimmed. It extends along the original curve definition. |

Type

Select the type of trim or extension to perform. You may select one of the following:

- | | |
|-------------------|---|
| <i>Tangent</i> | Extends tangent to the element (continuous in first derivative). |
| <i>Curvature</i> | Extends along the curvature of the element (continuous in first and second derivatives). |
| <i>Continuity</i> | Extend continuous to the element (continuous in first through third derivatives). |
| <i>Full</i> | Extend completely continuous to the element (continuous in all derivatives). |
| <i>Parameter</i> | This option is available only if the curve or surface was previously trimmed. It extends along the original curve definition. |

Menu Item 636

Menu Item 637

Create

Select either Lines, Points, Both or None to create elements when analyzing surface points.

Direction

When analyzing surface points, this option allows you to create the elements in either Normal to the surface or in the Cview.

Tolerance

Enter the tolerance to the surface to use when creating elements for analyzing surface points.

Menu Item 641

In Tolerance

Select the color to use to create elements that are within the tolerance entered for analyzing surface points.

Above

Select the color to use to create elements that are above the tolerance entered for analyzing surface points.

Below

Select the color to use to create elements that are below the tolerance entered for analyzing surface points.

Offset

Enter the desired offset distance to create the elements when analyzing surface points.

Menu Item 646

Menu Item 647

Align Edges

When editing tangents between two surfaces or splines you may align the edges of the elements. This moves the edges of the second element to align with the first element selected. Select either *Yes* or *No* to align the edges of the elements.

Type

Select one of the following plunge types:

Plunge Straight down plunge move.

Ramp Angle down plunge move.

Helical Helix down plunge move.

Pitch

This field is used when the plunge type is either Ramp or Helical. If type is Ramp then this field specifies the distance the tool will move down the Z axis in one ramp move. If type is Helical then this field specifies the distance the tool will move down the Z axis in one circular revolution of the helix.

Length

Specifies the length of the Ramp move in the XY plane.

Position

Defines where to place the ramp or helix move relative to the toolpath. The options are:

- Tangent* The plunge move (ramp or helix) will end at the toolpath point, this mode is usually used when contouring.
- Centered* The plunge move (ramp or helix) will be centered about the toolpath point, this mode is usually used when pocketing.

Feed Rate

Specifies the feed rate of the plunge move.

Angle

Specifies the angle of the plunge move. If plunge type is *Ramp*, this is the angle of the ramp move in the XY plane. If plunge type is *helical* this is the angle of a line from the toolpath point to the center of the helix.

Menu Item 657

Radius

This field specifies the radius of the helix when plunge type is helical.

Increment Value

Specifies the distance between tool passes when flow cutting.

Menu Item 660

Menu Item 661

Length

Specifies the length of the linear leadout moves.

Arc Leadout Angle

This option sets the sweep angle for a leadout move.

Constant

This field cannot be changed.

Arc Leadout Radius

This specifies the size of the leadout radius.

Leadin Type

SURFCAM can add linear or arc motion that cause the tool to approach and depart the workpiece with a tangent move. The list of all of the Leadin types can be accessed by hitting the right mouse button on the corresponding line of the pop-up.

<i>None</i>	No leadin or out moves are performed.
<i>Line on final</i>	A straight line move is used for the leadin or out move. This is only performed on the finish pass.
<i>Arc on final</i>	An arc is used to move the start point of the finish pass. The arc radius is four times the previous pass amount. The arc direction is determined by the tool path direction, so that the move becomes tangent while moving in the same direction as the tool path for the pass.
<i>1) line on all</i>	The straight line leadin or out move is performed for all passes in the tool path.
<i>2) arc on all</i>	The arc move is performed for all passes in the tool path.

When cutting around the outside of a contour, line infeed moves are helpful for causing the tool to plunge far enough away from the finished contour to completely clear the workpiece. This is highly advisable to prevent the tool from plunging directly into the workpiece.

When cutting a finish pass on a contour, using arc infeed moves will reduce the dwell mark usually found at the point the tool first contacts the finished edge of the workpiece.

If None is chosen, the tool will plunge closer to the start point of the contour and infeed in a perpendicular motion.

You are allowed the same options for departing the contour as for approaching the contour in Leadin type. The options are listed above.

NOTE: SURFCAM allows you to control the choice of infeed and outfeed moves on all passes, or just the finish passes.

NOTE: Gouge checking is not performed for leadin or out moves.

Menu Item 669

Menu Item 670

Menu Item 671

Leadout Type

SURFCAM can add linear or arc motion that cause the tool to approach and depart the workpiece with a tangent move. The list of all of the Leadin types can be accessed by hitting the right mouse button on the corresponding line of the pop-up.

<i>None</i>	No leadin or out moves are performed.
<i>Line on final</i>	A straight line move is used for the leadin or out move. This is only performed on the finish pass.
<i>Arc on final</i>	An arc is used to move the start point of the finish pass. The arc radius is four times the previous pass amount. The arc direction is determined by the tool path direction, so that the move becomes tangent while moving in the same direction as the tool path for the pass.
<i>1) line on all</i>	The straight line leadin or out move is performed for all passes in the tool path.
<i>2) arc on all</i>	The arc move is performed for all passes in the tool path.

When cutting around the outside of a contour, line infeed moves are helpful for causing the tool to plunge far enough away from the finished contour to completely clear the workpiece. This is highly advisable to prevent the tool from plunging directly into the workpiece.

When cutting a finish pass on a contour, using arc infeed moves will reduce the dwell mark usually found at the point the tool first contacts the finished edge of the workpiece.

If None is chosen, the tool will plunge closer to the start point of the contour and infeed in a perpendicular motion.

You are allowed the same options for departing the contour as for approaching the contour in Leadin type. The options are listed above.

NOTE: SURFCAM allows you to control the choice of infeed and outfeed moves on all passes, or just the finish passes.

NOTE: Gouge checking is not performed for leadin or out moves.

Menu Item 673

Menu Item 674

Menu Item 675

Menu Item 676

Menu Item 677

Angle modifier

This field specifies how the angle is used. The options are:

- | | |
|-----------------|--|
| <i>Relative</i> | The angle is measured relative to the toolpath. |
| <i>Absolute</i> | The angle is measured in the cut view. |
| <i>Auto</i> | The angle is automatically determined based on your cutting cycle. |

Clearance

This field applies to contouring only. It specifies the clearance amount between the finished contour and the plunge move.

Type

Select one of the following plunge types:

Plunge Straight down plunge move.

Ramp Angle down plunge move.

Helical Helix down plunge move.

Angle Type

Specifies the angle of the plunge move. If plunge type is *Ramp*, this is the angle of the ramp move in the XY plane. If plunge type is *Helical*, this is the angle of a line from the toolpath point to the center of the helix.

Length

Specifies the length of the Ramp move in the XY plane.

Radius

This field specifies the radius of the helix when plunge type is helical.

Pitch

This field is used when the plunge type is either Ramp or Helical. If type is Ramp then this field specifies the distance the tool will move down the Z axis in one ramp move. If type is Helical then this field specifies the distance the tool will move down the Z axis in one circular revolution of the helix.

Position

Defines where to place the ramp or helix move relative to the toolpath. The options are:

- Tangent* The plunge move (ramp or helix) will end at the toolpath point, this mode is usually used when contouring.
- Centered* The plunge move (ramp or helix) will be centered about the toolpath point, this mode is usually used when pocketing.

Feed Rate

Specifies the feed rate of the plunge move.

Clearance

This field applies to contouring only. It specifies the clearance amount between the finished contour and the plunge move.

Angle

Plunge Type

Select the desired type of plunge move. You may select Plunge, Ramp, or Helical.

Plunge Produces a straight line move to depth.

Ramp Creates a linear X, Y and Z axis move to depth.

Helical Creates a circular X, Y and Z axis move to depth.

Angle Type

Specifies the angle of the plunge move. If plunge type is *Ramp*, this is the angle of the ramp move in the XY plane. If plunge type is *helical* this is the angle of a line from the toolpath point to the center of the helix.

You may select either Relative or Absolute for the angle of the Ramp or Helical plunge moves. Relative is in relation to the leadin moves.

Angle Type

You may select either Relative or Absolute for the angle of the Ramp or Helical plunge moves. Relative is in relation to the leadin moves.

Length

Enter the desired length of the plunge moves. the default is .1. This length may be either constant or relative to the cutter radius. Cutter radius is the default.

Pitch

Enter the desired pitch for the helical moves. This pitch may be either constant or relative to the cutter radius. Cutter radius is the default.

Position

The plunge move may be either tangent or centered on the plunge location.

Feed Rate

Select the desired speed for the plunge moves. You may select plunge, rapid or feed.

Radius

This is the radius of the helix.

Menu Item 702

Menu Item 703

Major Direction

Enter the number of locations to analyze along the major axis of the surface or spline.

Minor Direction

Enter the number of locations to analyze along the minor axis of the surface or spline.

Create Points

Select either None, Minimum, Maximum, or < radius. Points are created based on the selected option. When < radius is selected the default radius is 0.1.

Radius

This option is only used when Create Points is selected as < radius. Enter the desired value to use for this parameter. The default is .1.

Menu Item 708

Menu Item 709

Menu Item 710

Menu Item 711

Menu Item 712

Menu Item 713

Menu Item 716

Rapid Plunge Clearance

This parameter is used to set the distance above the part to end the rapid move. It is also used as the minimum amount that the rapid plane must be above the part.

Plunge side

When contouring with a helical or ramp plunge this field specifies the side of the contour to plunge on.

Menu Item 721

Menu Item 722

Call SURFWARE

Files / STL Write Menu

This option allows you to save all active elements, or selected elements, into a STL file in any directory.

Menu Item 725

Part Name

STL files have an internal field that specifies the part name. This is not the name of the file.

STL file type

Specify output file type. Binary files are much smaller than ASCII files. Check with the system that will be receiving the STL file to see if it supports binary format.

Precision

Specifies the number of digits of accuracy the STL file will contain.

Surface tolerance

Because STL files can contain only polygons, SURFCAM must break surfaces up into polygons. This field specifies the tolerance to use in creating the polygons.

Sampling Curves

Enter the desired number of sampling curves used in surface polygon.

Menu Item 731

Menu Item 732

Surface output type

Because DXF does not support the non-uniform surfaces that SURFCAM uses the surface must be converted to another type. The allowed options are linemesh or polyface.

Surface tolerance

This field specifies the tolerance to use in creating the linemesh or polyfaces.

Sampling Curves

Enter the desired number of sampling curves used in surface polygonization.

DDE Menu

This menu is active when digitizing.

Line leadin Angle

This option sets the angle for a leadin move.

Line leadin Angle

This option sets the angle for a leadin move.

Menu Item 741

Enter Text

The four lines under *Enter Text* are there for you to type the annotation. To the right are *PgUp/PgDn* buttons. Each time you click on one of these SURFCAM will scroll up/down four lines. The rows being currently displayed are indicated at the top of the pop-up box next to *Line Number*.

After you accept the annotation text, the *Location* menu will be displayed and the prompt line will direct you to select the start point for the leader. This is where the point of the arrow will be placed. Once the point is chosen, you can click on the left mouse button and the leader and text will be displayed.

Rather than using the *Location* menu, you can move the mouse and thereby move the display of the leader. The prompt line directs you to click the left mouse button (LM) when you are satisfied with the location of the display. The prompt line also indicates that if you click the right mouse button (RM), you can insert intermediate points in the line of the leader.

Create

Arc This option creates an arc in the current construction view.

Spline This option creates a 3D spline between the selected elements with the curvature defined in the current construction view.

Both Both will create an arc and a spline

Menu Item 744

Menu Item 745

Under Color

This field specifies the color the surface will be shaded where there would be an undercut.

Over Color

This field specifies the color the surface will be shaded where there would not be an undercut.

Vector Menu

The *Vector* menu is displayed whenever a location and a direction are required in constructing a surface. *Revolution and Extrude*, from the *Create/Surface* menu, are two examples of surfaces that are constructed using a vector.

- Keyboard* If you select *Keyboard*, SURFCAM will display a pop-up box in which you can select the X, Y or Z axis as the vector or define your own axis.
- Tool Path* This option is available in the *Create/Vector* menu. It allows you to create a vector by chaining a profile. It requires selection of a beginning element and an ending element to identify a profile. The vector will always point to the left of the chained direction.
- 2 Points* With this option you can select two points. The first point is the starting point. The second defines the direction.
- Element* With this option you can select a single geometric entity for the vector. If a line is chosen the nearest endpoint becomes the starting location and the opposite end defines the direction. If you choose an arc or a circle the nearest endpoint becomes the starting location and the direction is the tangent at the endpoint.
- Point/Ang* If you choose this option, the *Location* menu will be displayed and you will be prompted to select the start point for the vector. A pop-up box will be displayed for you to enter an angle. The vertex of this angle is the vector start point you selected. The initial side of the angle is parallel to and in the same direction as the positive part of the horizontal axis of the current cview. The terminal side of the angle will be rotated in a clockwise (negative angle value) or counterclockwise (positive angle value) direction from the initial side and be on the plane of the current cview. This terminal side is the vector you are creating.
- Perp/Elem* This option defines a vector by using a selected endpoint of an entity such as a curve or another vector as the vector end point. The vector direction is defined by a perpendicular to the plane of the entity. You will first be prompted to select the end point of a curve or vector. Then you will be prompted to select a side of the curve. Clicking on one side or the other of the curve or vector will determine the direction of the vector you are creating.

Text

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Vector

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Edit Text Menu

The *Text/Dim* option allows you to make changes in the text and dimensions contained in your drawing. When you select *Text/Dim* for the *Edit* menu the *Edit Text* menu is displayed. Most of these options produce pop-ups that were used in the original creation of the text and/or dimensions you will be editing.

- Text* When you select *Text* from the *Edit Text* menu, you are prompted to select the text or dimension to change. Make your selection by clicking on any part of the dimension or text, i.e., arrows, extension lines, dimension lines, or text. For a detailed discussion of this option refer to the **Text** section of the **Text/Dimensioning** chapter of the manual.
- Font* This option allows you to change the font of an individual dimension or text. When you select *Font* from the menu, SURFCAM prompts you to select the text or dimension for which to change fonts. After you make your selection, the SURFCAM font pop-up is displayed allowing you to change the font type.
- Explode* This option allows you to change text into a geometric entity that can be machined. When you select *Explode*, you are prompted to select the text to explode. Click on an item of text and it will be changed to machinable geometry. It cannot be changed back to text.
- Style* The *Style* option allows you to create new styles for your text and dimensions that will apply to your entire drawing. Refer to the section on **Style** in the **Text/Dimensioning** chapter of the manual.
- Position* The *Position* option allows you to change the position of an existing dimension. After selecting *Position* from the menu, you are prompted to select the dimension to edit. After clicking on a dimension the rubber-banding mode discussed in the section on **Placing the Dimension** in the manual is activated. This allows you to reposition the dimension.
- StyleNum* This option is the same as the *Change Stl* option in the *Create/Dimension* menu. It allows you to change the style for the entire drawing.
- Properties* The *Properties* option allows you to change the style properties for an individual dimension. The individual dimension will have a style different from the rest of the drawing. The style is created in the *Dimension Style* pop-up which is discussed in the **Style** Section of the **Text/Dimensioning** chapter.

Spindle direction

CW for clock-wise, CCW for counter clock-wise.

Spindle direction

CW for clock-wise, CCW for counter clock-wise.

Spindle direction

CW for clock-wise, CCW for counter clock-wise.

Menu Item 755

Menu Item 756

Cone Start Angle

Identifies the beginning of the revolution that creates the surface.

Cone End Angle

Identifies the end of the revolution that creates the surface.

Cone Major Radius

The radius at the base of the cone.

Cone Minor Radius

The radius at the top of the cone.

Torus start angle in major direction

Start angle around major axis.

Torus end angle in major direction

End angle around major axis.

Torus start angle in minor direction

Start angle around center of torus cross section.

Torus end angle in minor direction

End angle around center of torus cross section.

Type

Single	This option allows you to create curves at a specific location anywhere on the surface. The direction of the splines is defined in the Direction option.
Increment Length	This option allows you to create curves a specified distance apart along the surface. The direction the splines follow is defined in the Direction option.
No. of splines	This option allows you to define a number of curves to be created evenly spaced along the surface. The direction the splines follow is defined in the Direction option.
Edges	This option allows you to define a number curves to be created along the edges or the surface. The edges on which the splines are created are based upon the Direction option.
Knot lines	This option allows you to create curves at the natural knots of the surface. The edges on which the splines are created are determined by the Direction.

Number of splines

No. of splines When the Type is set to *No. of splines*, this field sets the number of curves to create.

Projection Direction

Prj Direction There are two options to define the direction in which the element is to be projected.

Cuboid length

The length of the cuboid along the X axis.

Cuboid width

The length of the cuboid along the Y axis.

Cuboid height

The height of the cuboid along the Z axis.

Menu Item 771

Constant between similar cross sections

This option affects how the surface blends through similar cross sections. If set to *Yes*, the system will maintain the shape of the surface between similar cross sections before changing shape to a different section. If you select *No*, SURFCAM will allow the surface to change its shape as it blends through all cross sections

Create / Surface / Primitives Menu

Surface *Primitives* are simple geometric definitions that are common in many designs. In SURFCAM there are seven different primitive surface features; cylinder, cone, sphere, torus, wedge, cuboid, and plane. With these features you can immediately construct surfaces that would require extra steps and time using other surface construction methods.

Cylinder You can create a full or partial cylinder.

Cone You can create a full or partial cone.

Sphere You can create a full or partial sphere.

Torus You can create a full or partial torus.

Wedge You can create a wedge.

Cuboid You can create a cuboid.

Plane You can create a plane.

In five of these *Primitives* options (*Cylinder*, *Cone*, *Sphere*, *Torus*, and *Wedge*) surfaces are created whose constructions involve using a surface of revolution. If the revolution is a full 360 degrees a full surface will be created. If the revolution is less than 360 degrees, a partial surface is created. In each of these five options the degree of revolution is controlled by values for a *Start angle* and an *End angle*. If the *Start angle* is 0 and the *End angle* is 360, you create a full surface of revolution. If the *Start angle* is -30 and the *End angle* is 330, you also create a full surface of revolution. If, however, the *Start angle* is 30 and the *End angle* is 180, you create a 150 degree partial surface of revolution. *Start* and *End* angle values are entered in pop-up boxes that accompany each of these options. The default values for *Start* and *End* angles in those options will produce full surfaces.

Cone Height

Distance from the top to the bottom along the cone axis..

Menu Item 775

Menu Item 776

Menu Item 777

Menu Item 778

Cylinder Height

The distance from top to bottom along the cylinder axis.

Cylinder Radius

The radius from the axis.

Cylinder Start Angle

Identifies the beginning of the revolution that creates the surface.

Cylinder End Angle

Identifies the end of the revolution that creates the surface.

Menu Item 783

Menu Item 784

Sphere Radius

This is the radius of the sphere.

Sphere start angle in Major Direction

Start angle around the defined Z or major axis.

Sphere end angle in Major Direction

End angle around the defined Z or major axis.

Sphere start angle in Minor Direction

Start angle around the defined X or minor axis.

Sphere end angle in Minor Direction

End angle around the defined X or minor axis.

Menu Item 790

Menu Item 791

Plane length

The length of the plane along the X axis.

Plane breadth

The length of the plane along the Y axis.

Torus major Radius

Outside radius from the defined Z or major axis.

Torus minor Radius

Radius of the torus cross section.

Primitive Orientation

There are four options to define the axis of a primitive.

- Cview X, Y, Z* The first three options are based on the current construction view and local origin. For example, if the *Cview* is set to 1 and the *Orientation* is set to *Cview Z*, the major axis of the primitive will be parallel to the *World Z* axis.
- Specify* With this option you can define the axis through a specified origin and vector. SURFCAM will prompt you to select major and minor axes. The vector is normal to the major and minor axis positions.

Primitive Orientation

There are four options to define the major axis of a primitive.

Cview X, Y, and Z The first three options are based on the current construction view and local origin. If the *Cview* is set to 1 and the *Orientation* is set to *Cview Z*, the major axis of the sphere will be parallel to the World Z axis.

Specify With this option you can define the axis through a specified origin and vector. SURFCAM will prompt you to select major and minor axes. The vector is normal or perpendicular to the major and minor axis positions.

Menu Item 798

Wedge height

The height of the wedge

Wedge radius

This is the radius of the wedge

Wedge start angle

Start angle around the defined Z, or Major axis.

Wedge end angle

End angle around the defined Z, or Major axis.

Plane Orientation

There are four options to define the orientation of the plane.

Cview X, Y, and Z The first three options are based on the current Cview and local origin. If the Cview is set to 1 and the Orientation is set to Cview XY, the plane will be parallel to the XY plane.

Specify You can define the Major(X) and Minor(Y) axes.

Angle

This is the angle of the primitive around the Major, or Z axis.

Block Spacing

Block Spacing of text is available by selecting *Create* from the *Main* menu, then *Text/Dim*, then *Text* .

This option affects the spacing between characters. When set to *No* the system uses the default spacing for the chosen font. When set to *Yes*, the spacing parameter is activated. The value you enter for spacing will be the distance between the characters of text.

With block spacing each character exists within a block that cannot overlap another block.

Spacing

This option is available only if you have chosen block spacing. Enter the amount of space you want between characters.

Menu Item 807

Create / Dimension Menu

With SURFCAM you are able to display both text and dimensions.

<i>Dimension</i>	A single entity consisting of text, dimension lines, and extension lines. The text for linear measurements consists of numbers only. Arc radius measurements are preceded by the letter <i>R</i> . Arc diameter measurements are preceded by the symbol \varnothing .
<i>Text</i>	You can annotate your drawing with textual material. One option, called <i>Leader</i> , is text connected to a line pointing to a specific entity in your drawing. The other option, called <i>Text</i> , allows you to create text used for notes in the drawing.

Depending on the dimension option you choose, SURFCAM will display the *Location* menu or prompt you to select an entity.

SURFCAM treats dimensions like other geometric types, i.e., it places them on the plane of the current construction view.

When creating a dimension, SURFCAM recognizes when the text is centered between the extension lines. As you move the mouse to locate the dimension on the drawing, the text color will change when the text becomes centered between the extension lines.

<i>Linear</i>	measures the horizontal or the vertical distance between points.
<i>Align</i>	measures the straight-line distance between two points.
<i>Radius</i>	measures the radius of any circle or arc.
<i>Diameter</i>	measures the diameter of any circle or arc.
<i>Angular</i>	measures the angle between two lines or the angle of an arc.
<i>Leader</i>	creates a line with an annotation.
<i>Text</i>	creates text for notes in a drawing or to be exploded for machining.
<i>Style</i>	All the characteristics of a dimension format. Used to make changes in the <i>Default Style</i> or to create and name new styles. Use <i>Change stl</i> , not <i>Style</i> , to change from one style to another.
<i>Change stl</i>	SURFCAM displays a pop-up box listing the available styles.
<i>MouseText / StyleText</i>	Toggle switch. With <i>MouseText</i> you can move the mouse and click on the location where you want the dimension displayed. With <i>StyleText</i> you depend on the current style to place the dimension.

Menu Item 809

Menu Item 810

Menu Item 811

Annotation--Dimension units

Closest Round Off This value defines the number of places to the right of the decimal point for rounding purposes. The measured distance will always be a multiple of this value.

Center Mark

Mark Length This value affects the size of the mark drawn in the center of the arc.

Menu Item 814

Menu Item 815

Enter Style Name

This field is used to enter a new style name when you are creating a new style or changing the name of an existing one.

If you are creating a new style, enter the new style name and move to the *Enter Style Number* box to assign a number to the new style.

Menu Item 817

Menu Item 818

Menu Item 819

Menu Item 820

Dimension Lines

Dimension Lines Dimension lines are the lines that point to extension lines (lines extending from points on the drawing) indicating which distance is being measured. The one pointing at the extension line of the point selected first is considered the first dimension line. Dimension lines also point to arcs and angles being measured.

Suppress Dimension Lines

You may suppress the display of dimension lines. Selecting *Yes* next to *1st* suppresses the first dimension line. Selecting *Yes* next to *2nd* suppresses the second dimension line. Selecting *No* next to either causes the corresponding dimension line to be displayed

Dimension Lines

Dimension Lines Dimension lines are the lines that point to extension lines (lines extending from points on the drawing) indicating which distance is being measured. The one pointing at the extension line of the point selected first is considered the first dimension line. Dimension lines also point to arcs and angles being measured.

Suppress Dimension Lines

You may suppress the display of dimension lines. Selecting *Yes* next to *1st* suppresses the first dimension line. Selecting *Yes* next to *2nd* suppresses the second dimension line. Selecting *No* next to either causes the corresponding dimension line to be displayed

Extension Lines

Extension Lines The extension lines are lines extending from points on the drawing indicating the distance being measured. The one extended from the point selected first is considered the first extension line.

Suppress Extension Lines You may suppress the display of extension lines. Selecting *Yes* next to *1st* suppresses the first extension line. Selecting *Yes* next to *2nd* suppresses the second extension line. Selecting *No* next to either causes the corresponding extension line to be displayed.

Extension Lines

Extension Lines The extension lines are lines extending from points on the drawing indicating the distance being measured. The one extended from the point selected first is considered the first extension line.

Suppress Extension Lines You may suppress the display of extension lines. Selecting *Yes* next to *1st* suppresses the first extension line. Selecting *Yes* next to *2nd* suppresses the second extension line. Selecting *No* next to either causes the corresponding extension line to be displayed.

Menu Item 825

Dimension Lines

Color

This value is the number for the color of the dimension lines. The default color is set to *Inherit* which makes the dimension lines the same color as the entity being measured. Clicking on the color button displays a color chart allowing you to set the color value to a different number.

Extension Lines

Color

This is the value for the extension line color. The default color is set to *Inherit* which makes the extension lines the same color as the entity being measured. Clicking on the color button displays a color chart allowing you to set the color value to a different number.

Dimension Text

Text Color

Select the color for the dimension text. Select a color from the palette. The value for that color is then displayed in the *Text Color* field. Or select *Inherit* for the text to be the color of the element you choose to dimension.

Menu Item 829

Annotation--Dimension Tolerance

Upper and Lower Tolerances

With these options you are able to define tolerance values above(+) or below(-).

Annotation--Dimension Tolerance

Upper and Lower Tolerances

With these options you are able to define tolerance values above(+) or below(-).

Annotation--Dimension Tolerance

Tolerance Precision

This value defines the number of places to the right of the decimal point.

Annotation--Dimension Tolerance

Tolerances If you do not wish to allow any tolerance, select *None*. To display the same tolerance value above and below, select *Symmetrical*. To allow for a different tolerance above and below, select *Deviation*.

Annotation--Dimension Tolerance

Vertical Justification

There are three parameters for this option: Top, Middle, or Bottom. These position the tolerances relative to the dimension text.

Dimension Lines

Force Line Inside Selecting **Yes** for this option will cause a continuous line to be displayed between the extension lines. This option is intended for use when the dimension text is placed outside of the extension line.

Menu Item 836

Arrow Heads

Draw Arrow Head Selecting *Yes* will cause the display of an arrow head. If you select *No* there will be no arrow heads displayed.

Menu Item 838

Arrow Heads

Arrow Height

The value entered here defines the length of the arrow head measured along the dimension line.

Arrow Heads

Arrow Width

The value entered here defines the width of the arrow head.

Menu Item 841

Extension Lines

Origin gap

The origin gap is the distance between the extension lines and the corresponding coordinates.

Extension Lines

Extension The extension is the distance the extension line extends beyond the dimension line.

Menu Item 844

Menu Item 845

Annotation--Dimension units

Angle format You are able to measure angles in decimal values or in degrees and minutes.

Menu Item 847

Menu Item 848

Dimension Text

Dimension Line Gap

The dimension line gap is the distance between the text and the dimension line.

Dimension Text

Vertical Gap

This option is available only if you have selected either *Above dimension line*, or *Away from defining point*. The value you enter defines the distance between the text and dimension line.

Dimension Text

Text Position with respect to

The parameters that follow this heading affect the placement of the text relative to the dimension and extension lines.

Extension Line

For this option you may select *Chosen by user* or *Outside extension lines*. The first is the default which provides you with the option of placing the dimension display in any location you choose. Move the mouse until the dimension is in the desired location and then click on the left mouse button. Selecting *Outside extension lines* will place the text outside one of the extension lines.

Dimension Text

Text Position with respect to

The parameters that follow this heading affect the placement of the text relative to the dimension and extension lines.

Dimension Line You may select from *Center between dimension lines*, *Above dimension lines*, or *Away from defining point*.

NOTE: You may modify this placement by moving the mouse until the text is at your desired location. Then click on the left mouse button.

The *Away from defining point* option places the text on the opposite side of the dimension line from the two points concerned.

Dimension Text

Text alignment The default option for *Text alignment* is *Horizontal*. This will display the text horizontally regardless of the dimension line direction. The other option, *Align with dimension line*, displays the text along the dimension line.

Dimension Text

Text Font

To select a font, click on the *Text Font* button. A pop-up will be displayed for you to select a font.

Style

The style may be chosen by typing its number or selecting it with the mouse.

Keep outer contours

Select *Yes* or *No*

Pocket nesting depth

Select *All enclosed* or *Choose*. If you select *Choose*, the *Depth* parameter will be available for your input.

Depth

Enter the number.

Annotation--Dimension units

Dimension Precision

This value defines the number of places to the right of the decimal point in linear measurements.

Dimension

Click the button to toggle between an X and a blank.

When the box contains an X, the element is selectable.

When the box is blank, the element is masked and may not be selected.

Menu Item 861

Dimension Lines

Lead length

This value is the horizontal length of a dimension line. If the entire dimension line is horizontal, this is the distance from the dimension text to the extension line. If there is an elbow in the dimension line, this is the length of only the horizontal part of the dimension line.

Annotation--Dimension units

Angle Precision This value defines the number of places to the right of the decimal point in angular measurements.

Scale

Scale affects the size of the dimension display. Changing the value of *Dimension scale* changes the display size, e.g., a *Dimension scale* of two will double the size of the display.

Menu Item 865

Menu Item 866

Menu Item 867

Create / Surface / Points Menu

This menu is used to create a surface through either a set of points or a line mesh.

Points If you select the *Points* option, the Points pop-up box will be displayed. After selecting parameters in this pop-up you will be prompted to select points that will be used to create the surface.

Line Mesh If you select the *Line Mesh* option, the *Line Mesh* pop-up box will be displayed. After selecting and entering parameters in this pop-up you will be prompted to select a line mesh.

NOTE: A line mesh may be read into SURFCAM in one of three ways. An IGES file created by another CAD/CAM product can be read in using the IGES2DSN utility described in the SURFCAM Utilities chapter of the SURFCAM manual. A DXF file can be read in using the *DXF* option in the main Files menu. Files containing digitized data can be read in using the *Digitized* option in the main Files menu.

Points Option

ControlPts

Specifies that the points are used to develop the surface using the specified knots and degrees defined later.

Interpolate

Specifies that the points are used to develop a 3rd degree surface. If this option is chosen, the system will gray the *Knot* and *Degree* variables since they do not apply.

Menu Item 870

Length

This value defines the length of extrusion along the defined axis.

Menu Item 872

Line Mesh Option

- ControlPts* Specifies that the points are used to develop the surface using the specified knots and degrees defined later in the pop-up.
- Interpolate* Specifies that the points are used to develop a 3rd degree surface. If this option is chosen, the system will gray the *Knot* and *Degree* variables since they do not apply.

Menu Item 874

Menu Item 875

Rows

This defines the number of rows, or sections, that will be used to define a surface.

Columns

This defines the number of points in each row. There must be an equal number of columns/points in each row.

Menu Item 878

Keyboard Axis Option

With the *Axis* option you can select the X, Y, or Z *Axis* to define the vector or select *Define* to define your own vector. If you choose either the X, the Y, or the Z *Axis*, the origin (0,0,0) is the end point of the vector and the positive X, Y or Z direction is the vector direction. If you choose *Define*, the end point of the vector will also be the origin but the direction of the vector will be determined by the X, Y, and Z values that you enter.

Synchronization

When creating a surface from cross sections that contain the same number of elements or nodes, SURFCAM prompts you to synchronize the elements. Synchronizing causes each cross section element to be matched with the other cross section elements.

The pop-up will only be displayed when it is applicable. For example, if the section consisted of two separate arcs the system would not display the *Synchronize* pop-up due to the fact that there is only one element per section. The pop-up will not be displayed if sections traveling both directions are chained.

The *Synchronize* pop-up gives you the following options:

- | | |
|-----------------|--|
| <i>None</i> | The system will not match any of the entities from one section to another. |
| <i>Nodes</i> | When the system recognizes splines, it will match each node together. If any single spline in a chain has a different number of nodes, this option will not be available. If the sections contain both splines and elements the system will provide an option to synchronize the elements only, the nodes only, or none. |
| <i>Elements</i> | The system matches each element to the corresponding element in each cross section. If the sections contain both splines and elements the system will provide an option whether to synchronize the elements or not. |

Menu Item 881

Synchronize

When creating a surface from cross sections that contain the same number of elements or nodes, SURFCAM prompts you to synchronize the elements. Synchronizing causes each cross section element to be matched with the other cross section elements.

The pop-up is displayed only when it is applicable. For example, if the section consisted of two separate arcs the system would not display the *Synchronize* pop-up due to the fact that there is only one element per section. The pop-up will not be displayed if sections traveling both directions are chained.

The *Synchronize* pop-up gives you the following options:

- | | |
|-----------------|--|
| <i>None</i> | The system will not match any of the entities from one section to another. |
| <i>Nodes</i> | When the system recognizes splines, it will match each node together. If any single spline in a chain has a different number of nodes, this option will not be available. If the sections contain both splines and elements the system will provide an option to synchronize the elements only, the nodes only, or none. |
| <i>Elements</i> | The system matches each element to the corresponding element in each cross section. If the sections contain both splines and elements the system will provide an option whether to synchronize the elements or not. |

Menu Item 883

Text Font

Select *True Type* or *Stroke* fonts. Then select your font from the list displayed and click on OK.

Angle

The number of degrees that you enter for this value will determine the angle at which the text will be displayed on the drawing--measured counterclockwise from horizontal.

Menu Item 886

Annotation--Dimension Tolerance

Text Height Factor This value is the size of the tolerance text relative to the size of the dimension text.
A scale factor of .5 would make the tolerance texts half the size of the dimension text.

Menu Item 888

Menu Item 889

Menu Item 890

Center Mark

Draw Center Mark The center mark indicates the center of the arc being measured.

None

Selecting *None* will result in no mark being drawn.

Mark

If you select *Mark*, a point will be drawn at the center of the arc.

Line

If you select *Line*, a line will be drawn in each quadrant extending from the center.

Menu Item 892

Menu Item 893

Menu Item 894

Enter Text

The four lines under *Enter Text* are there for you to type the annotation. To the right are *PgUp/PgDn* buttons. Each time you click on one of these SURFCAM will scroll up/down four lines. The rows being currently displayed are indicated at the top of the pop-up box next to *Line Number*.

After you accept the annotation text, the *Location* menu will be displayed and the prompt line will direct you to select the start point for the leader. This is where the point of the arrow will be placed. Once the point is chosen, you can click on the left mouse button and the leader and text will be displayed.

Rather than using the *Location* menu, you can move the mouse and thereby move the display of the leader. The prompt line directs you to click the left mouse button (LM) when you are satisfied with the location of the display. The prompt line also indicates that if you click the right mouse button (RM), you can insert intermediate points in the line of the leader.

Enter Text

The four lines under *Enter Text* are there for you to type the annotation. To the right are *PgUp/PgDn* buttons. Each time you click on one of these SURFCAM will scroll up/down four lines. The rows being currently displayed are indicated at the top of the pop-up box next to *Line Number*.

After you accept the annotation text, the *Location* menu will be displayed and the prompt line will direct you to select the start point for the leader. This is where the point of the arrow will be placed. Once the point is chosen, you can click on the left mouse button and the leader and text will be displayed.

Rather than using the *Location* menu, you can move the mouse and thereby move the display of the leader. The prompt line directs you to click the left mouse button (LM) when you are satisfied with the location of the display. The prompt line also indicates that if you click the right mouse button (RM), you can insert intermediate points in the line of the leader.

Enter Text

The four lines under *Enter Text* are there for you to type the annotation. To the right are *PgUp/PgDn* buttons. Each time you click on one of these SURFCAM will scroll up/down four lines. The rows being currently displayed are indicated at the top of the pop-up box next to *Line Number*.

After you accept the annotation text, the *Location* menu will be displayed and the prompt line will direct you to select the start point for the leader. This is where the point of the arrow will be placed. Once the point is chosen, you can click on the left mouse button and the leader and text will be displayed.

Rather than using the *Location* menu, you can move the mouse and thereby move the display of the leader. The prompt line directs you to click the left mouse button (LM) when you are satisfied with the location of the display. The prompt line also indicates that if you click the right mouse button (RM), you can insert intermediate points in the line of the leader.

Text

Page Up Button

Clicking on the *Page Up* button will display the four lines of text that precede the lines currently displayed (unless the current lines are the first four). The numbers of the lines being displayed are listed at the top of the pop-up box.

Text

Page Down Button Clicking on the *Page Down* button will display the four lines of text that follow the lines currently displayed. The numbers of the lines being displayed are listed at the top of the pop-up box.

Menu Item 900

Menu Item 901

Menu Item 902

Variable Radius

If this option is set to *Yes*, the system will gray out the *Enter fillet radius* option since it is not applicable.

Set this option to *Yes* only if you have the two surfaces and their required reference curve is already created.

Select *Accept* and SURFCAM will prompt you to select the first and second surfaces and the reference curve.

After making these selections, you will be prompted to select a reference point on the reference curve or select *Done*.

When you select a reference point or *Done*, a radius pop-up will be displayed prompting you to enter the fillet radius to be used at that reference point.

After you enter a value and select *Accept*, this step is repeated. When the last radius is selected, select *Done* to continue.

Menu Item 904

Line Type

Lines The Lines option will allow you to create a set of separate line segments connected to each other. The Location menu is displayed and you are prompted to select the starting point and the ending point.

Polyline The Polyline option allows you to create a set of connected line segments that form a single entity. The point where neighboring segments meet is called a node.

A polyline is created in the same manner as a line string. The *Location* menu is also displayed when the *Polyline* option is selected.

Menu Item 906

Display text/dimensions

To display text/dimensions or not, choose *Options* from the *Main* menu and *Display* from the *Options* menu. You will be able to set the *Display Text/Dimensions* parameter.

This option allows you to set dimensions to be viewed at all times or only in certain views.

Always This is the default and will force the dimensions to be visible regardless of the current view.

In view only This will display dimensions only when the current view matches the construction view the dimension were created within.

NC EDM Menu

SURFCAM is used to perform 2 axis and 4 axis operations for different EDM applications.

To program a 2 axis machine, it is necessary to create only a single tool path. Create the tool path by selecting *Contour* from the *NC EDM* menu and chaining the desired profile. After the profile has been identified, define the tool and program parameters and the system will generate the INC file containing the tool path information.

To program a 4 axis machine it is necessary to creating tool paths for the top and the bottom profile using the *Contour* option. There are a few rules to follow when creating these tool paths.

Both profiles **MUST** have an equal number of elements.

Curve tolerance value **SHOULD** be set to 0.

Both profiles **SHOULD** be contoured at Z0

Wire compensation and offset side **MUST** be programmed:

The INC filename for the UV contour **MUST** include an X:

Splines are not supported:

After the profiles have been identified, define the tool and program parameters and the system will generate the INC file containing the tool path information.

Contour When *Contour* is selected from the *NC EDM* menu, SURFCAM will display the chaining menu.

Options Choosing *Options* from the *NC EDM* menu results in a pop-up box with 2 axis cutting parameters. Many items that affect the *Contour* option can be set from this pop-up.

NOTE: For three of these options, the rightmost column can be toggled between *Constant* and *Cutter Radius*. Use the *Cutter Radius* for a particular value to be calculated automatically by multiplying the number entered in the first column by the cutter radius. For the value to always remain the same, set the 2nd column to *Constant*. The value entered will be the exact inch, or millimeter, value of that parameter and will not vary with step or tool size.

Edit This option displays an Open dialog box allowing you to load an INC file for editing. It then displays the Edit NC menu allowing you to edit the INC file.

Post This option displays the Post menu allowing you to perform post processing operations.

NC Mode This option displays the NC Mode menu allowing you to switch to another NC Mode by selecting from a list of available modes.

Wire Radius

Specifies the radius of the wire.

Offset Side

<i>Center</i>	This is used to drive the center line of the wire directly along the path of the chained elements.
<i>Right</i>	SURFCAM drives the tool to the right side of the contour based on the chaining direction.
<i>Left</i>	SURFCAM drives the tool to the left side of the contour based on the chaining direction.

Curve Tolerance

When a toolpath follows a spline, SURFCAM will break the moves into linear segments. This value you enter for *Curve tolerance* defines the chordal deviation.

Menu Item 913

Menu Item 914

Side Clearance Type

You can toggle this choice to indicate where to place the side clearance moves applied: *None*, *On All (Contours)*, or *With C-Comp*.

The *With C-Comp* choice will cause the resulting NC program to add diameter compensation codes to the infeed and outfeed moves: G41, G42 and G40 on most machines.

Side Clearance

Indicate the desired length of the perpendicular infeed and outfeed moves. When diameter compensation is desired at the machine, this value must be larger than the value stored in the corresponding diameter offset register on the machine, otherwise the machine may not be able to correctly cut the contour.

Diameter compensation may be performed using two methods. You may compensate for the entire tool diameter. This is normally used when the part line is programmed. The tool is offset for the tool radius to maintain the proper cut location. You may also program the tool centerline. You may also compensate for undersize tools. This is typically used when programming the tool centerline. The compensation is adjusted for the difference in the tool diameter programmed and the tool diameter being used.

Finish Passes

SURFCAM automatically incorporates finish passes into the cycle if you enter a non zero number. For a single pass around the contour, set this value to 1 or 0. You can, however, have as many finish passes as desired. Each pass will remove the amount of stock entered in the Finish cut spacing parameter below.

Finish Spacing

Specifies the amount of stock to be removed in each finish pass.

Stock to leave

This cutting operation will leave the stock, in the amount of the value entered, on the sides of the contour.

Initial Power

This value is used by the machine for the initial burn into the material.

Power Setting

This value is used by the machine for the length of the contour.

Feed Rate

Specifies the XY axis feed rate in inches per minute.

Taper Angle

This is the angle of the wire as it moves along the contour.

Menu Item 924

Comment - Line 1

You can place a comment into the tool change output during post processing as a message to the operator.

Comment - Line 2

You can place a comment into the tool change output during post processing as a message to the operator.

Menu Item 927

Planar

The system will chain any element, any direction. This may not lend itself well when there is excess geometry. If set to *Yes* the system will only chain elements that are in the same plane as the current construction view. If set to *No* the system will chain any direction.

Tangent

This option, when set to Yes, identifies an angular threshold to limit the chaining to elements that do not exceed this tolerance. It is not commonly used in chaining 2 axis profiles. It is commonly used when chaining elements for surface construction.

Angle Tolerance

This value defines the maximum gap or overlap to jump. If the geometry is not trimmed properly the system may not chain a complete profile. Increasing this value will allow the system to make up for these gaps.

Planar

When this option is set to *No*, the system will chain in any direction. When set to *Yes*, the system maintains the chaining direction along the plane defined by the first entity chosen.

Tangent

With this option you can constrain the chaining of two elements within an angular tolerance. If tangent is set to *No*, SURFCAM will chain two elements that come together at any angle. If set to *Yes*, SURFCAM will chain only elements that meet at an angle less than you specify in the *Angle tolerance* parameter.

Angular tolerance

Angle tolerance This is the angular tolerance threshold before the system will stop chaining.

Height

This is the height of the text.

Dimension Text

Text Height This is the height of the text.

Dimension Text

Block spacing

This option affects the spacing between characters. When set to *No*, the system uses the default spacing for the chosen font. When set to *Yes*, the spacing parameter is activated. The value you enter for spacing will be the distance between the characters of text.

With block spacing each character exists within a block that cannot overlap another block.

Dimension Text

Block spacing

This option affects the spacing between characters. When set to *No*, the system uses the default spacing for the chosen font. When set to *Yes*, the spacing parameter is activated. The value you enter for spacing will be the distance between the characters of text.

With block spacing each character exists within a block that cannot overlap another block.

Cross Section Menu

- Sections Use this option to select cross sections that are parallel to each other. This option may not be used if any of the sections intersect one another.
- Grid This is used when the cross sections travel in both directions. Use this when the sections form a four-sided patch, or number of patches. It is not necessary to break each section that intersects another.
- Auto The *Auto* function provides automatic selection of the cross sections used to develop the surface. When *Auto* is chosen the system displays the selection pop-up to identify the sections.
- Manual The *Manual* function provides individual selection of each cross section used to develop the surface. When *Manual* is chosen the system displays the *Single/Chain* menu to allow identification of the cross sections.
- Options If you choose *Options* the system will display a further pop-up box for your input.

Create Rationals

SURFCAM uses this option when creating splines and surfaces.

- Yes** This will force EXACT representation of the elements used for construction. It will increase the size of the database and is recommended when perfect accuracy is a must.
- No** This will provide an approximate representation of the elements used for construction. It will decrease the size of the database and is recommended when general tolerances are required.

The surface will be built within the *User Tolerance* of the exact theoretical surface. (Enter the *User Tolerance* value in pop-up that appears when you select *Tolerances* from the *Options* menu.) The default is .00005 inches or .001 mm.

Menu Item 940

Constant between similar cross sections

When creating a surface through the cross sections, the system changes the shape to smoothly transition between the sections. If two sections are identical, a Yes response will cause the system to maintain the shape between the two identical sections.

Menu Item 942

Planar

This option locks the chaining plane to the current construction view. Any intersecting geometry that does not lie within this view is disregarded.

Tangent

This option affects how the system chains cross sections. Since SURFCAM can automatically create a surface out of cross sections traveling in different directions, the system is faced with circumstances-- primarily in corners--in which a *U* section ends and a *V* section begins. If some ambiguity exists in this regard, choose *Yes*. You will then have the opportunity to define the *Angle tolerance*. SURFCAM relies on this angular tolerance to evaluate situations when sections change direction abruptly.

This option affects how the system chains cross sections. Since SURFCAM can automatically create a surface out of cross sections traveling in different directions, the system is faced with circumstances-- primarily in corners--in which a *U* section ends and a *V* sections begins. If some ambiguity exists in this regard, choose *Yes*. You will then have the opportunity to define the *Angle tolerance*. SURFCAM relies on this angular tolerance to evaluate situations when sections change direction abruptly.

Angle tolerance

SURFCAM relies on angular tolerance to evaluate situations in which sections change direction abruptly.

Maximum gap between intersecting splines

When creating a surface through sections that cross each other, the system requires a tolerance to develop a surface. This value defines the maximum distance between intersecting sections. If the distance is larger than the tolerance then the system will error.

Vertical surf check

With this option you can accelerate the tool path generation by excluding edge protection on vertical surfaces. When importing designs from solids based systems, this feature can be set to *NO* since all surfaces should be closed. If, however, a design leaves a vertical surface open without a top it should be set to *YES*. If it is questionable then set the option to *YES*.

Trim Plane Menu

Access this menu by selecting *Create/Surface/Trim Plane*.

The *TrimPlane* option creates a composite planar surface at the edge of another surface or within a closed curve.

Cap The *Cap* option will create a planar surface at the edge of another surface or along a curve. When you select *Cap*, you will be prompted to select a curve or a surface edge to cap. Select a curve such as a circle, an arc, or a spline, or select the edge of a surface. SURFCAM will create a plane surface along the edge of the surface or along the curve.

Chain When you select the *Chain* option, SURFCAM will display another menu with two options: *Chain* and *Single*. (Notice that this is a separate Chain option.)

With the *Single* option you can create a trim plane along a single curve similar to the *Cap* option.

With the *Chain* option you can chain a series of separate curves, creating a single curve, and automatically create a trim plane along its edge.

Auto When you select *Auto*, SURFCAM displays the *Select* menu and you are prompted to *Select contour(s) to be chained, then Done*.

Menu Item 949

Menu Item 950

Menu Item 951

Menu Item 952

Menu Item 953

Menu Item 954

Menu Item 955

Menu Item 956

Menu Item 957

4/5 Axis Cuts

If you have already selected 4 Axis or 5 Axis for the *NC Mode*, you can access this screen by selecting *NC/Cut*, then select a surface.

There are three types of 4/5 Axis cuts that can be selected: *Lead/Lag*, *Point*, and *Curve*.

Lead/Lag Enter a value for the lead angle. The *Lead Angle*--the angle at which the tool passes over the part--is measured from the normal to the surface to the shank of the tool. It has a positive value if it is in the direction of the cut.

You may select an angle that allows the tool shank to *lead* the tip across the surface. This is considered a positive lead angle. This action effectively drags the tool tip over the surface.

Selection of a negative angle allows the tool shank to *follow* the tip across the surface. This produces a pushing effect on the cutter.

NOTE: It is not recommended that you use a lag angle with standard end mills. This would require the bottom of the end mill to perform the cut, and may not produce the desired results during the cutting process.

NOTE: It is recommended, for all machines with rotary axis limits, that all 4 axis cuts be made *Bi-directional*. This allows the rotary axis to unwind at each pass.

After entering the desired lead angle the *Save As* pop-up is displayed prompting you to give a name to the INC file SURFCAM is about to create for you. After entering the name and clicking *OK* in the *Save As* pop-up, the *Tool Information* and *Program Information* pop-ups are displayed.

Point Selecting this option forces the center of the shank to pass through a defined point. This will control the tool vector. It is only available in the cut cycle. It is commonly used when there is an undercutting application where the tool must cut between surfaces or underneath another surface. When the *Point* option is chosen, the system will display the *Location* menu and prompt you to select a tool axis point. Select the point and the system will display the *Save As* pop-up prompting you to enter a file name. After entering the name and clicking *OK*, the *Tool Information* and *Program Information* pop-ups are displayed. After the tool and program parameters are defined, the system will begin to create a toolpath by cutting the surface while the shank continually passes through the point.

Curve The *Curve* option is similar to the point option. This method provides greater control over where the tool vector passes. It is commonly used for more complex undercutting applications where the tool must cut between surfaces or underneath another surface. When this option is chosen, the system will display the *Chain* menu and prompt you to select the beginning element of the tool axis curve. Then you are prompted to select the ending element of the tool axis curve. After you select the ending element, the *Save As* pop-up is displayed prompting you to enter a file name. After entering the name and clicking *OK*, the *Tool Information* and *Program Information* pop-ups are displayed. After the tool and program parameters are defined the system will begin to create a toolpath by cutting the surface while the shank passes through the curve.

Planar

This option locks the chaining plane to the current construction view. Any intersecting geometry that does not lie within this view is disregarded.

Tangent

This option affects how the system chains cross sections. Since SURFCAM can automatically create a surface out of cross sections traveling in different directions, the system is faced with circumstances-- primarily in corners--in which a *U* section ends and a *V* section begins. If some ambiguity exists in this regard, choose *Yes*. You will then have the opportunity to define the *Angle tolerance*. SURFCAM relies on this angular tolerance to evaluate situations when sections change direction abruptly.

Angle tolerance

SURFCAM relies on angular tolerance to evaluate situations when sections change direction abruptly.

Menu Item 962

3 Axis Cut Parameters

If 3 axis has been selected as the NC mode, access this screen by selecting *NC/Cut*. Then select a surface and give a file name for your INC file. The *Gouge check* parameter will be available on the pop-up that is displayed.

<i>Gouge check</i>	These options are for use when there are circumstances where the tool has a curvature larger than the surface or there are adjacent surfaces that would be gouged.
<i>None</i>	This option does not check for gouging.
<i>Single Surface</i>	This option will check the curvature of the tool and compensate for any undercutting conditions.
<i>Multi Surface</i>	With this option, SURFCAM will check adjacent surfaces. SURFCAM will prompt for a surface to be machined, then the surfaces to avoid.

Write normal

This option has a *Yes/No* toggle. It is used to output the I, J, K vector normal to the surface at the contact point of the tool. This is used by some machines for 3D cutter compensation. If set to *Yes*, SURFCAM will output the vector. If set to *No*, it will not.

Write normal

This option has a *Yes/No* toggle. It is used to output the I, J, K vector normal to the surface at the contact point of the tool. This is used by some machines for 3D cutter compensation. If set to *Yes*, SURFCAM will output the vector. If set to *No*, it will not.

Write normal

This option has a *Yes/No* toggle. It is used to output the I, J, K vector normal to the surface at the contact point of the tool. This is used by some machines for 3D cutter compensation. If set to *Yes*, SURFCAM will output the vector. If set to *No*, it will not.

Menu Item 967

Delete / Styles / Del Styles Menu

If you select the *Styles* option from the *Delete* menu, a *Del Style* menu will be displayed along with a pop-up listing the styles available. A style that has been selected with the mouse will show the letter D in the first column of the pop-up to indicate that it will be deleted when *Done* is selected. Clicking on any line in the box toggles the D in the first column.

Deleting a style deletes all of the parts of that style along with the style. Deleted styles can be undeleted by using the *Undelete Last* or *Undelete All* functions provided you have not created a new style with the same number.

Delete This option allows you to select a range of styles. A pop-up is displayed for you to enter the beginning and ending style numbers to delete. After accepting these values, the letter D is displayed in the first column of the style box for the style numbers entered. This is an inclusive list. This option is not used to delete non-consecutive styles.

Undelete This option is used to undelete a range of styles. The pop-up for the beginning and ending style numbers is displayed. The letter D is removed from the style numbers entered. This option is not used for non-consecutive style.

Swap D When this option is selected the letter D for the styles marked for deletion are removed and displayed for the styles that are not marked for deletion. The letter D toggles each time this option is selected.

Done When the desired styles are marked correctly select this option to delete the marked styles. These styles may be undeleted using the *Undelete Last* or *Undelete All* options on the *Delete* menu.

Menu Item 969

Menu Item 970

Menu Item 971

Chain Options

Angle Tolerance This value defines the maximum gap or overlap to jump. If the geometry is not trimmed properly the system may not chain a complete profile. Increasing this value will allow the system to make up for these gaps.

Chain Options

Tangent

With this option you can constrain the chaining of two elements within an angular tolerance. If *tangent* is set to *No*, SURFCAM will chain two elements that come together at any angle. If set to *Yes*, SURFCAM will chain only elements that meet at an angle less than you specify in the *Angle tolerance* parameter.

Menu Item 974

Enter Style Number

Type in the new style number for the new style name entered in the *Enter Style Name* box and select *Accept*.

Side step mode

Selecting *Side step mode* provides two options to control how the tool will move between each cut. This option applies only to the *Cut* feature.

Follow edge This option keeps the tool in contact with the material as it moves to the next cut.

Direct This option moves the tool in X, Y then Z between moves if the next cut is lower in Z or reverses the order if the next move is higher.

Side step mode

Selecting *Side step mode* provides two options to control how the tool will move between each cut. This option applies only to the *Cut* feature.

Follow edge This option keeps the tool in contact with the material as it moves to the next cut.

Direct This option moves the tool in X, Y then Z between moves if the next cut is lower in Z or reverses the order if the next move is higher.

Side step mode

Selecting *Side step mode* provides two options to control how the tool will move between each cut. This option applies only to the *Cut* feature.

Follow edge This option keeps the tool in contact with the material as it moves to the next cut.

Direct This option moves the tool in X, Y then Z between moves if the next cut is lower in Z or reverses the order if the next move is higher.

Spark Gap

This value defines the distance to offset the tool path for spark compensation. The system does include this offset in the tool path, rather it is used by the control.

Wire Feed

If the machine has the ability to automatically feed the wire through the material, set this to Yes.

Surface Tolerance

This value is the maximum error allowed in the direction of the cutter. SURFCAM will create as many linear moves as needed to hold the cutting to within this tolerance.

Offset Side

Specifies the two types of tool motion listed below:

- Left* Causes a clockwise cutting tool to climb cut the contour with standard clockwise cutting tools. SURFCAM drives the tool along to the left side of the contour.
- Right* Exactly opposite to the above choice, SURFCAM drives the tool along to the right side of the contour. This results in a conventional cutting process if using standard clockwise cutting tools.

NOTE: You can machine either side of the contour. The side that gets machined depends on the relationship between the chaining direction and the type of cut parameter. SURFCAM always machines contours in the direction that they are chained, and that in conjunction with the type of cut parameter determines which side of the contour is machined. If you choose *Left* to climb cut a pocket, the pocket must be chained in a counter-clockwise direction, and any islands inside the pocket must be chained in a clockwise direction. Conversely, if you choose *Right* to conventional cut a pocket, the pocket must be chained in a clockwise direction, and islands inside the pocket must be chained in a counterclockwise direction. Chaining in the wrong direction will cause the tool to cut on the wrong side of the contour or islands.

Menu Item 983

Change Color of Last Surface

In a multi-surface cutting operation, if a surface cannot be processed because of some problem, this option allows you to change its color so you can identify it and fix the problem.

Color

Enter the color number for the last surface.

Wire EDM--Options

Max Feed Between Size

When the wire finishes one pass and must move to the start of the next pass, it can either rapid or feed across the part at the current feed rate setting. If the distance from the end of one pass to the start of the next is smaller than the value specified here, the wire will feed across.

NOTE: Too large a value could cause the tool to overcut the part by feeding through a narrow area. To avoid this, reduce the *Max feed between size* to less than the distance to the start of the next pass.

Wire EDM--Options

Corner Clearance When the *Corner angle* cut is in effect, this value determines the wire clearance from the direction change intersection. When this value is zero, the wire remains in contact with the part walls during the directional change.

Wire EDM--Options

Side Clearance This allows you to set the default *Side clearance* value that is used for the infeed and outfeed movements at the start and end of each pass.

Wire EDM--Options

Corner angle

This option determines the tool path at each change of direction. When the interior angle of the directional change is larger than this value, the tool path is intersectional. Any directional change with the interior angle greater than 150 degrees is an intersectional tool path. Interior angles less than this value use either an *Arc* or *Line* to prevent the intersectional move. Angles smaller than 30 degrees automatically use either *Arc* or *Line* moves. The default for this value is 135 degrees.

Wire EDM--Options

- Arc* This option uses an arc between directional changes that are less than the *Corner angle* value. The tool path uses the *Corner clearance* amount to determine the location of the arc.
- Line* This option uses a line between directional changes that are less than the *Corner angle* value. The tool path uses the *Corner clearance* amount to determine the line location.

Cut Waterfall Ends

This parameter controls how planar cutting is done in relation to edge protection.

- Yes** Selecting *Yes* causes SURFCAM to cut the surfaces and to extend the cut along or through the edge protection. This results in what is often called the *waterfall* effect or *waterfalling* as the tool path cascades over the surface edge.
- No** Selecting *No* causes SURFCAM to cut only the actual surfaces and not cut beyond the edges into the edge protection regions. There will be no *waterfalling* of the tool paths at the surface edges.
- Cutter Diameter** This is similar to the *Yes* option but SURFCAM will cut all the edge protection region except a distance equal to the cutter diameter.
- Cutter Radius** This is similar to the *Cutter Diameter* option but SURFCAM will cut all the edge protection region except a distance equal to the cutter radius.

Limit Angle at Ends

This parameter controls the cutting at the start and end of each planar cut. The parts of a planar cut between the two ends are always cut.

- No* Selecting *No* causes SURFCAM to cut the entire planar cut, regardless of the angle at the ends.
- Yes* Selecting *Yes* causes SURFCAM to not cut areas steeper than the *Max Cut Angle* at the start and end of each planar cut.

Menu Item 999

Cap

When extruding geometry to create a surface, this option allows you to automatically create cap surfaces at the positions indicated.

Start A cap will be generated at the original geometry position.

End A cap will be generated at the position to which you extrude the geometry.

Both A cap will be generated at both the *Start* and the *End*.

None No cap will be created.

Depth First

This option applies to 2D Pocketing and Contouring with multiple depth cuts.

Yes Will cut down the depth of each pocket before going to the next pocket.

No Will cut all pockets at each depth before going to the next depth.

Shelf plane

This is the Z value that is used when the projection does not fall on a surface.

Curve tolerance

SURFCAM will break all contour moves along a spline into straight linear movements. When driving the tool around a splined curve, SURFCAM will adjust the length of the cutting moves to make sure that the edge of the contour is machined to this tolerance value.

Rapid plane

This specifies an incremental height relative to the top of the tool path where the tool stops rapid motion and begins to plunge into the material.

Stock to leave

Stock to leave This option is active when the *Type of cut* option is set to *On Top*. It specifies the amount of material left on the surface after the tool path is generated.

Use Centerline

Use Centerline If this option is set to *No*, the *Hole depth* parameter controls the depth of the drilled hole. If it is set to *Yes*, the *Hole depth* option is disabled and a line, which you must have already drawn, controls the depth of the drilled hole. The *Create/Line* menu can be used to draw the line. The starting point of the line must be a separate point or the center of a circle. SURFCAM uses the starting point of this line to locate the center of the drilled hole. The line must be parallel to the Z axis.

Mask circle

Mask circle

This option allows you to define drilling locations by selecting circles with a specified diameter. Circles with a different diameter and points can thus not be used to select drilling locations.

Circle diameter

Circle diameter The circle diameter used when the *Mask circle* option is set to Yes.

Menu Item 1009

Menu Item 1010

Menu Item 1011

Menu Item 1012

Menu Item 1013

Depth to

This option controls which location on the drill is taken to the depth defined by *Hole depth*.

Center This will move the drill deeper to carry the full diameter to the defined depth. To do this the *Included angle* field must be defined.

Tip This, the default, will take the tip of the drill to the defined depth.

Included Angle

Included Angle The *Included Angle* is the Tool Tip Angle. The default value for a standard twist drill is 118. The default value for each drill in the tool library file, SURFCAM.TOL, can be changed using the *Tool File/Modify Tool* option of the **Configuration Utility**.

Clearance

Clearance

This field allows you to define an extra distance for the overall depth of the hole. This can make up for a drill that has rounded edges and can break the edge to reduce deburring.

Check Surfaces

Check surfaces If you select *Yes*, the *Check clearance* parameter appears so that you can enter a value. This defines the amount of stock to leave on the surfaces chosen to avoid. This is useful when tool deflection is in question and provides added flexibility when using the *Planar* feature.

Check clearance

Check clearance This defines the amount of stock to leave on the surfaces chosen to avoid. This is useful when tool deflection is in question and provides added flexibility when using the Planar feature.

Check clearance

Check clearance When using *Multi-surface* as the option for *Gouge check*, the *Check clearance* field becomes available to enter a value to define the amount of stock to leave on the surfaces chosen to avoid.

Finish cut on

This option defines when the pocketing or contouring function will take the finish pass. The default is to take it at the final depth.

Final depth only This takes the finish pass at the final depth.

All depths This takes a finish pass at each depth.

NURB Curve Option

- Interpolate* This option will create a NURB curve that passes through the points. This is done by creating new control points.
- ControlPts* This option will create a NURB curve that uses the chosen points as control points. When this option is chosen, the *Knots* and *Degree* parameters become available.

Points

This option allows you to create a natural cubic spline by indicating the locations of the node points. When SURFCAM is used to create a spline in this manner, the resulting spline is called a natural spline which means that the curvature is zero at the ends of the spline.

Menu Item 1023

Menu Item 1024

Menu Item 1025

