

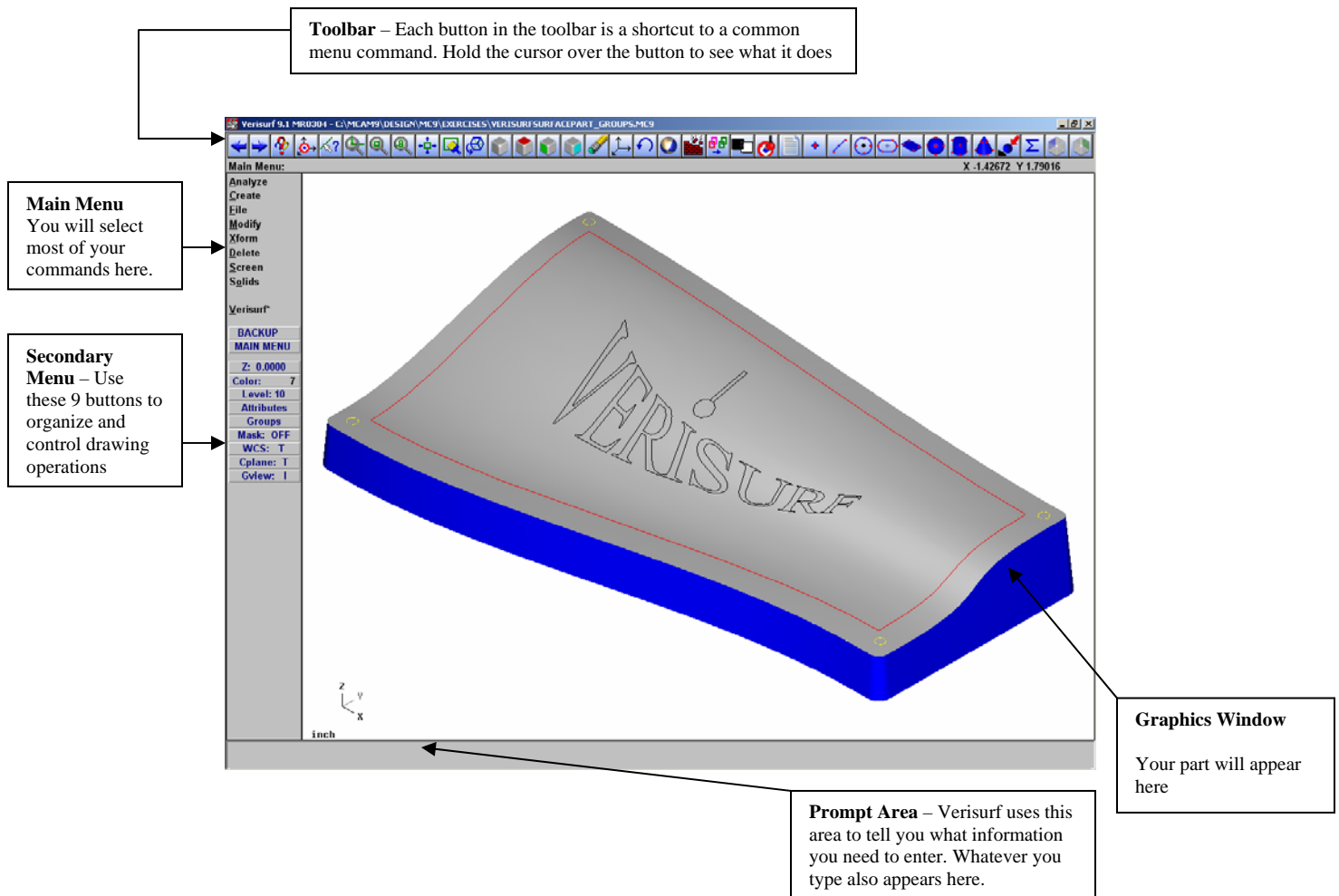


Verisurf Cad Design Basics REV A1

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Learning the Verisurf Interface

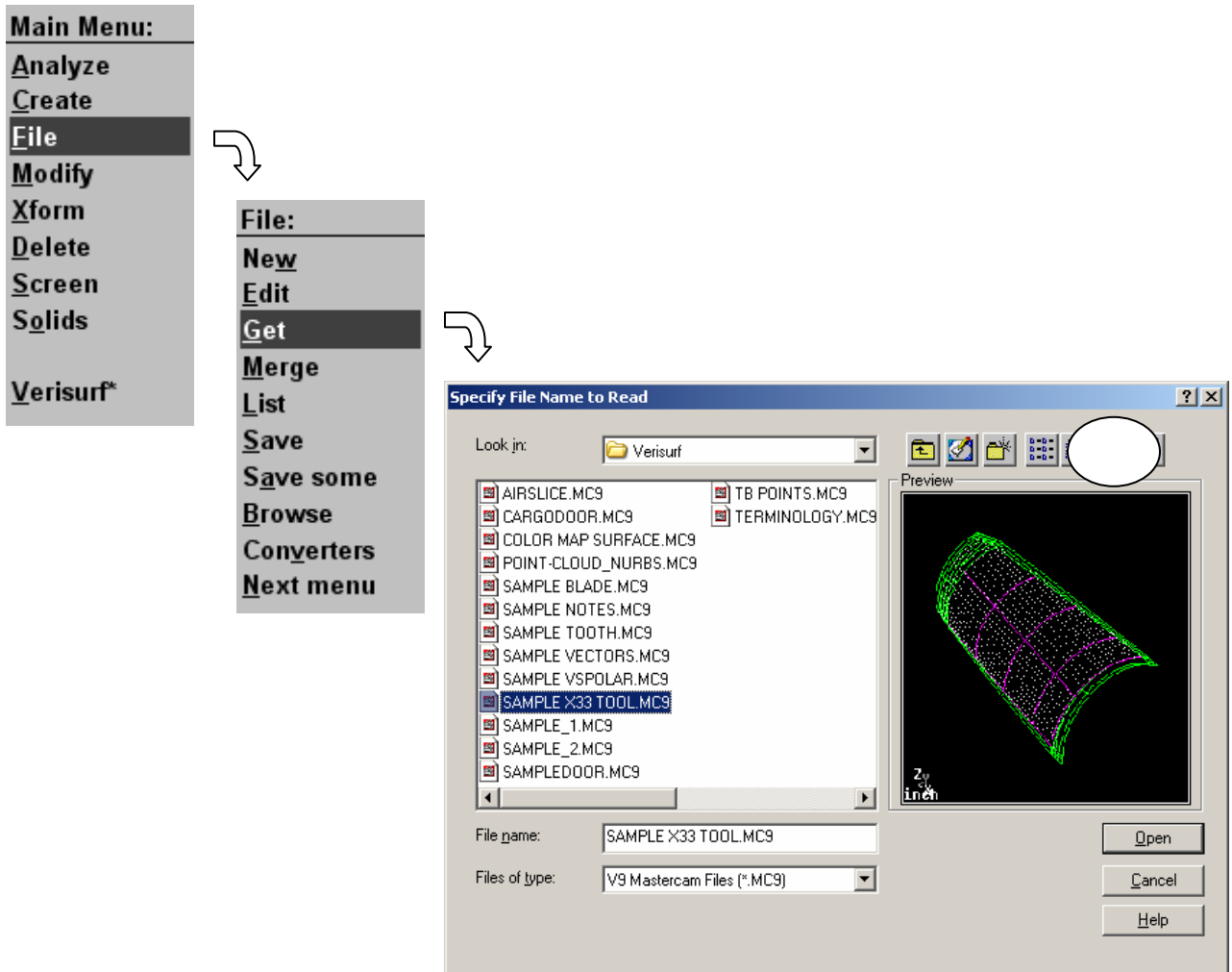


Learning about the Hasp

Verisurf Software uses two types of licensing: single-user licensing and network licensing. Most likely you are using single-user licensing. If you are, you need to have a special piece of hardware called a HASP (sometimes known as a dongle or SIM) attached to your parallel or USB port. If you are using network-licensing then a NetHASP must be installed on a computer on your network.

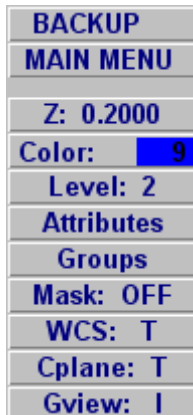
Opening up a part file

Go to Main Menu.



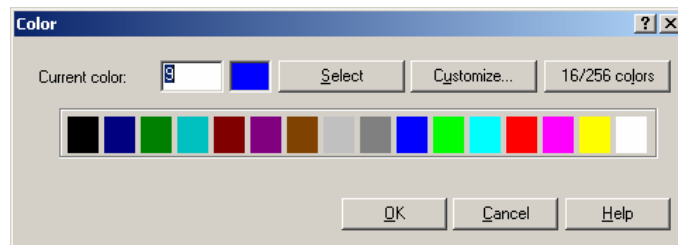
Choose one of the parts to bring it into your graphics window. In this example the part file SAMPLE X33 TOOL.MC9 file is chosen. Note that the screen shows a preview of the part. This is due to the button circled being pressed. It is the preview button.

Secondary menu



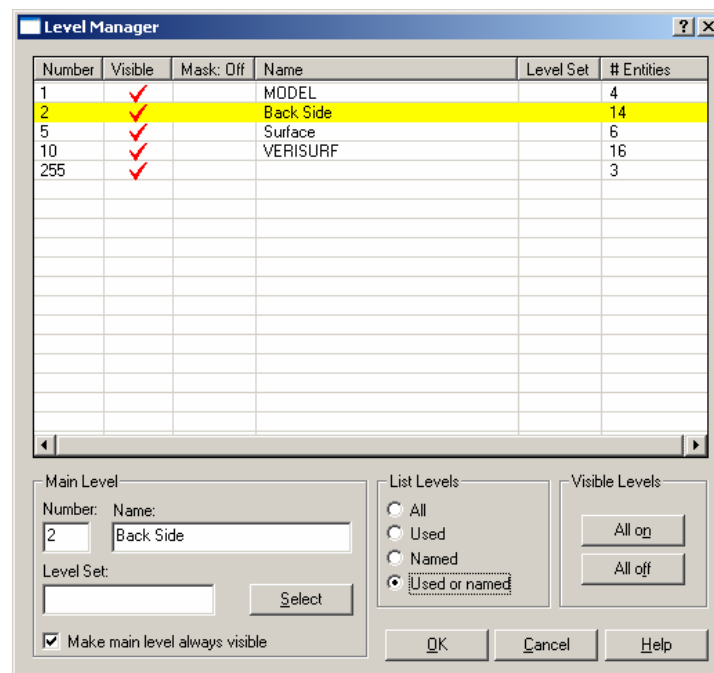
The secondary menu is used to quickly adjust a number of parameters. The primary ones used are Color, Level, Gview, Z and Attributes. They are used to edit and draw entities.

Color – Used to change the active color of newly created entities or to change the color of current entities.



Level – Levels are a primary organizational tool in the cad system. A drawing file can contain separate levels for wireframe, surfaces, drafting and entities. By organizing your drawing into levels, you can more easily control which areas of the drawing are visible at any time and which parts are selectable so that you don't inadvertently make changes to areas of the drawing you don't want to change.

You are always drawing on the main level. You can set any one of 255 levels to be the main level, and you can name any level. You can set the main level using the Level button on the Secondary Menu, and set a level mask by using the Mask button. Verisurf also allows copying or moving of geometry from level to level, lets you hide levels from view, provides level naming utilities, and allows you to organize several levels into sets.

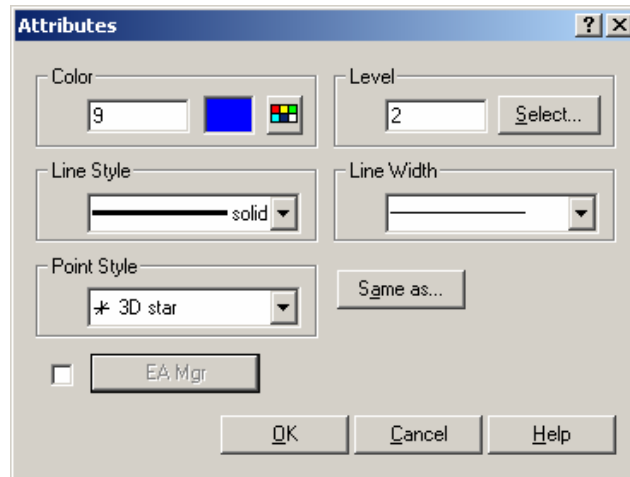


Gview - In Verisurf you can view parts from any vantage point. The Gview function in the Secondary Menu provides several predefined views and allows you to freely rotate your vantage point around the geometry. Gview does not move or otherwise translate the geometry. Rather, Gview moves your vantage point. However, it appears the geometry is rotating when you change the Gview, especially when using Gview, Dynamic.

Z level - The Z button in the Secondary Menu sets the current construction depth (Z depth). The Z depth corresponds to a point on the Z axis of the current construction plane (Cplane) with respect to the system origin (X0,Y0,Z0).

Attributes – Change one or more attributes of the entities you will create or edit as follows:

- Color - Type the number of the color that you want to assign, or choose the Color palette button to select a color from the Color dialog box.
- Level - Type the number of the level that you want to assign, or choose Select to choose a level using the Level dialog box.
- Line Style - Choose a line pattern from the drop-down list.
- Line Width - Choose a line thickness from the drop-down list.
- Point Style - Choose a point type from the drop-down list.
- Same as - In the graphics window, select the entity whose attributes you want to assign.



Creation basics for Verisurf

The Create menu enables the operator to draw a number of different types of entities. The entities we will cover are Point, Line, Arc, Fillet, Spline and Drafting. Here is the Create menu which can be found from the main menu.

From the main menu > Create we have access to quite a few items. After choosing any of the entity types you will be prompted to select the method to create the entity. This selection will then take you to a point entry prompt area.

Let's look at Point.

Main Menu:
Analyze
Create
File
Modify
Xform
Delete
Screen
Solids
Verisurf®

Create:
Point
Line
Arc
Fillet
Spline
Curve
Surface
Rectangle
Drafting
Next menu

Point

Choosing Point will bring up this menu.

Point:
Position
Along ent
Node pts
Cpts NURBS
Dynamic
Length
Slice
Srf project
Perp/dist
Next menu

As you can see there are a number of ways to create a point. In the previous exercise we used Position to create a point. We manually typed in the position, however, the position can be derived in a number of ways by using the Point Entry menu. Many of the following selection methods will use Autocursor. AutoCursor is a point entry feature that is available whenever Verisurf displays the

Point Entry menu and prompts you to enter a point. AutoCursor eliminates menu steps by detecting and snapping to points as you move the cursor over geometry on the screen.

Point Entry:
Origin
Center
Endpoint
Intersec
Midpoint
Point
Last
Relative
Quadrant
Sketch

- Origin – Will create a point at X0, Y0. The Z will be at wherever the Z level is set.
- Center – Will create a point at the center of a circle.
- Endpoint – Will create a point at the endpoint of a line, spline or arc.
- Intersec – Will create a point at the intersection of two entities.
- Midpoint – Will create a point at the midpoint of a line.
- Point – Will create a point at the X,Y location of another point.
- Last - From the Point Entry menu, choose Last. Verisurf highlights the point previously entered.
- Relative – Will create a point a relative distance from another point.
- Quadrant – Will create a point at one of the quadrants of a circle. 0, 90, 180, 270 degrees.
- Sketch – Will create a point at any cursor location.

Line

Choosing Line will bring up this menu.

Here again there are a number of ways to create a line.

- Horizontal – Create a line that is from left to right, usually the X axis.
- Vertical – Create a line that is up and down, usually the Y axis.
- Endpoints – Create a line between two known points or two entered points.
- Multi – Create multiple lines with the first lines 2nd point becoming the second lines 1st point.
- Polar – Create a line from a selected point along a direction.
- Tangent – Create a line tangent to another entity, usually an arc.
- Perpendclr – Create a line perpendicular to another entity.
- Parallel – Create a line parallel to another entity
- Bisect – Create a line that is midway between, or bisecting, two lines.

Line:
Horizontal
Vertical
Endpoints
Multi
Polar
Tangent
Perpendclr
Parallel
Bisect
Closest

Arc

Choosing Arc will bring up this menu. Here again there are numerous ways to create an arc, or circle.

- Polar - Polar Arc gives you options for creating arcs using a variety of polar coordinates methods.
- Endpoints – Create circle with 2 points and a radius. 2 arcs will result and you choose the one to keep.
- 3 points – Create an arc through 3 selected points.
- Tangent – Create an arc that can be tangent to one or many entities.
- Circ 2 pts – Create a circle through 2 points that define the diameter.
- Circ 3 pts – Create a circle through 3 points that define the diameter.
- Circ pt+rad - Create a circle using a center point and a predefined radius.
- Circ pt+dia - Create a circle using a center point and a predefined diameter.
- Circ pt+edg - Create a circle using a center point and a cursor or point defined radius.

Arc:
Polar
Endpoints
3 points
Tangent
Circ 2 pts
Circ 3 pts
Circ pt+rad
Circ pt+dia
Circ pt+edg

Fillet

The Fillet menu gives you options for filleting curves. When you fillet curves, you create an arc of a defined radius tangent to the curves. By default, Verisurf Software trims the curves to the fillet, although you have the option to turn off trimming.

New in Version 9.1 is the addition of the clearance fillet. A clearance fillet creates an arc trimmed to two line, but the arc is not tangent to the lines.

You can access the Fillet menu by choosing Main Menu, Modify, Fillet or by choosing Main Menu, Create, Fillet. Click on the menu and topics below for more information.

Fillet: Select Curves or:	
<u>R</u> adius	
<u>A</u> ngle<180	S
<u>T</u> rim	Y
<u>C</u> hain	
<u>CW</u> /CCW:	A
<u>C</u> learance	Y

Radius

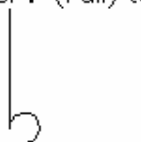
Determines the resulting radius of the fillet.

Angle<180

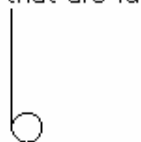
Determines the sweep of the fillet arc. This option toggles between S, L, and F. Choose **S** (Smaller) to create fillets that are smaller than 180 degrees, **L** (Larger) to create fillets that are larger than 180 degrees, or **F** (Full) to create fillets that are full circles (360 degrees).



fillet arc < 180 degrees



fillet arc > 180 degrees



fillet arc = 360 degrees

Note: curves are trimmed to fillet

Trim

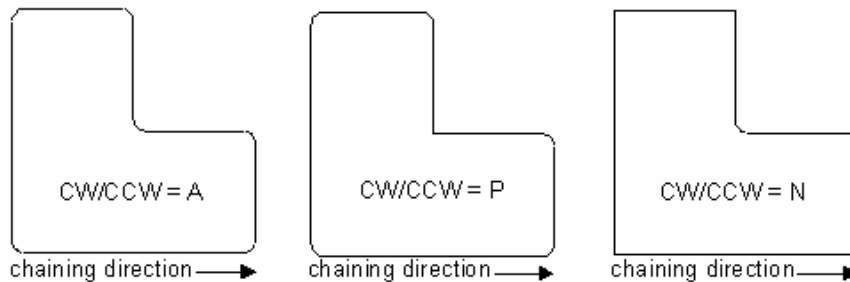
Determines whether Verisurf trims the selected curves to the fillet. This is a Yes/No toggle. Choose Y to activate trimming, N to deactivate trimming.

Chain

Displays the chaining methods menu where you can select chains of curves to create fillets at sharp corners along the chains.

CW/CCW

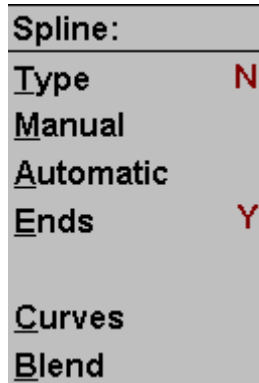
Determines which corners are used to create fillets along the chains of curves. This option toggles between **A**, **P**, and **N**. Choose **A** to create fillets along all corners, **P** to create fillets at positive, or counterclockwise (CCW) corners (relative to the chaining direction), or **N** to create fillets at negative, or clockwise (CW) corners (relative to the chaining direction).

**Notes:**

- ◆ This option applies only when [Filletting corners along chains of curves](#).
- ◆ The ability to fillet only inside or outside corners of a chained boundary is particularly useful during geometry creation for wire EDM parts (punches and dies). These parts often require different radius values on inside versus outside corners in order to provide adequate corner clearance.

Spline

The Spline menu gives you options for defining the method used to create the spline as well as parameters that further define the resulting geometry. You can access the Spline menu by choosing Main Menu, Create, Spline. Click on the menu and topics below for more information.



Type

Sets the type of spline to create. P for parametric, N for Nurbs.

Manual

Creates a spline that passes through points that you enter in the graphics window. The spline is created through the points in the order that they were picked

Automatic

Creates a spline from a string of existing points that lie in a defined pattern

Ends

Determines how Verisurf calculates the tangency of a spline at its endpoints when you create the spline using manual or automatic point entry. Toggle this option to Y to define the tangency of each endpoint or N to accept the default tangency, which results in the minimal spline length.

Curves

Create splines based on the geometry of existing entities. Verisurf creates a separate spline for each chain that you select.

Blend

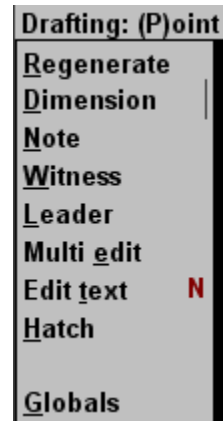
Create a spline tangent to two selected curves.

Drafting

The Drafting menu gives you options for creating and editing drafting entities: dimensions, labels, notes, witness lines, leader lines, and hatch entities. Accessing the Drafting menu also puts you in SmartMode, where you can select, create, and edit a variety of drafting entities. In addition to creating or editing specific drafting entities, you can also set or edit drafting parameters that affect drafting entities you select or create. You can access the Drafting menu by choosing Main Menu, Create, Drafting.

Regenerate

When you create associated drafting entities, changing the size or position of geometry changes the status of associated drafting entities from "clean" to "dirty." A status of "dirty" means that the position and/or value of the associated entity no longer matches its geometry. Dirty drafting entities are highlighted in red. Regenerating the drafting entity updates its position and/or value so that they once again match the geometry. Only associated drafting entities can be regenerated. However, you can validate both associated and non-associated drafting entities.



Dimension

Verisurf provides two menus for creating drafting dimensions: the Drafting menu (SmartMode) and the Dimension menu.

Drafting menu (SmartMode). The Drafting menu, when displayed, puts you into SmartMode, an integrated drafting environment where you can create all dimensions except baseline, chained, or ordinate. You can also create other entities such as notes, labels, leaders, and hatch entities from the Drafting menu. To access the Drafting menu, choose Main Menu, Create, Drafting.

When to use. Because of its flexibility in allowing you to create or edit many different kinds of drafting entities, you will usually find it more convenient and efficient (fewer mouse clicks) to create dimensions (except baseline, chained, or ordinate) in SmartMode than from the Dimension menu—especially when you are creating or editing different types of dimensions in succession. You must use SmartMode to create perpendicular dimensions, which cannot be created from the Dimension menu.

Dimension menu. The Dimension menu gives you options for creating specific types of dimensions. You can use the Dimension menu to create all dimensions except perpendicular. To access the Dimension menu, choose Main Menu, Create, Drafting, Dimension. Then choose the option (Horizontal, Vertical, etc.) that corresponds to the dimension you want to create.

When to use. If you are creating multiple instances of a particular dimension type (horizontal, vertical, etc.), you may find it more efficient to choose that option from the

Dimension menu than to create the dimension in SmartMode. You must use the Dimension menu to create baseline, chained, and ordinate dimensions, as these dimensions cannot be created in SmartMode (from the Drafting menu).

Note: You cannot select, reposition, or edit existing dimensions from the Dimension menu. Instead, display the Drafting menu, then use SmartMode and the Quick Edit options to select, reposition, and edit dimensions you wish to change.

Note

You can create a drafting note by choosing the Note option from the Drafting menu.

1. Choose Main Menu > Create > Drafting > Note
The Note dialog box opens.
2. Enter note text in one or more of the following ways:
Key In. Type the text directly into the Note text box. If desired, press [Enter] to create line or paragraph breaks.
File. Click Load File to open the Specify File Name to Read dialog box. Locate and select a file containing the text (ASCII, unformatted) you wish enter, then click Open.
Symbol. Click Add Symbol to open the Select Symbol dialog box, then click the symbol you wish to enter.
3. Select either of the following:
Single Note to create one instance of the note.
Multiple Notes to create multiple copies of the same note.
Note: You can choose Properties to edit the note's text properties using the Drafting Globals dialog box.
4. Choose OK. The Note dialog box closes.
5. Drag the note text to a desired position in the graphics window, then click to set its position. If you selected Multiple Notes in step 3, drag and position each additional note copy, then press [Esc] to exit the Note function.

Note: You can use the Quick Edit options to edit the note before setting its position.

Witness

Witness lines are simple, straight drafting lines (no arrowheads) used to indicate boundaries or connect entities. In dimensions, witness lines are paired parallel lines that indicate the end points of a dimension.

You can also create freestanding witness lines by choosing Witness from the Drafting menu. You can create freestanding witness lines anywhere in the graphics window. Freestanding witness lines can be associated with the geometry selected during their creation or by entering points relative to an entity, such as midpoints, endpoints, center points, quadrant points, etc.

Leader

Leader lines are drafting lines with single arrowheads that function as pointers. In Verisurf, leader lines most often point from a dimension's text to its witness lines. You

can also create freestanding leader lines by choosing the Leader option from the Drafting menu. You can use this option to create single or segmented leader lines. You can create freestanding leader lines anywhere in the graphics window. Freestanding leader lines can be associated with the geometry selected during their creation or by entering points relative to an entity, such as midpoints, endpoints, center points, quadrant points, etc.

Multi edit

You can use the Drafting menu's Multi edit function to edit parameters of selected drafting entities, using the Drafting Globals dialog box.

Edit text

Turning on the Drafting menu's Edit text function allows you to edit the text of drafting dimensions, notes, and labels without changing the entity's position in the graphics window.

Globals

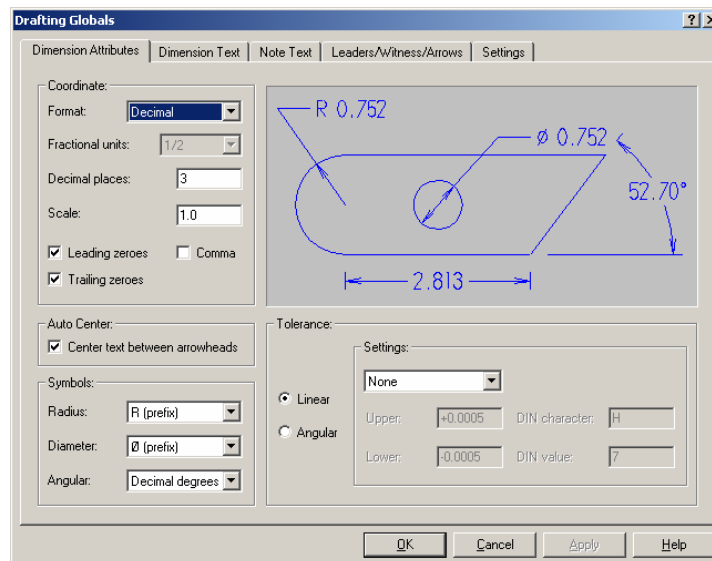
The Dimension Attributes tab of the Drafting Globals dialog box lets you set parameters that determine the following properties of drafting dimensions:

- .. Coordinate display format and scale
- .. Text centering
- .. Symbols or units used to display radius, diameter, and angular dimensions
- .. Tolerance settings for linear and angular dimensions

These parameters affect drafting dimensions that you create or edit, depending upon what function you are in when you access the dialog box.

Note: For additional help with individual parameter settings, click the ? in the upper-right corner, then click on a parameter.

Note: This menu can also be set at Main Menu > Screen > Configure > CAD Settings > Global.



Manipulating views in Verisurf

Verisurf allows you to work in any plane in 3D space, unlike 2D CAD/CAM systems that only allow you to work in a flat XY plane (similar to a piece of paper). Planes can be defined with any combination of coordinates in X, Y, and Z. These planes are also called views. A plane plus an origin is called a view.

Views can be used as graphics views (Gviews) to view the part and as construction planes (Cplanes) to orient geometry.

There are eight system views in the View list of the View Manager. These system views correspond to standard planes; Top (X, Y) Front (X, Z), Back (-X, Z), Left Side (-Y, Z), Right Side (Y, Z) Bottom (-X, Y), Isometric and Axonometric. These are available in any file and you cannot change them.

In addition to the system views, you can create your own views (sometimes referred to as "named" views). You can create as many views as you want, but you can only align the Work Coordinate System (WCS) with only one of them at a time. This becomes the view that determines the orientation of the Top plane. Cplanes and Gviews are defined relative to this view. In other words, changing the WCS to a different view changes the orientation of the Cplanes and Gviews.

Gviews

In Verisurf you can view parts from any vantage point. The Gview function in the Secondary Menu provides several predefined views and allows you to freely rotate your vantage point around the geometry. The predefined views are planes that match the predefined construction planes (Cplane). You may want to set the Gview to match the Cplane when constructing geometry.

You can also set the Cplane to automatically update to match the Gview when you change it. Choose Screen, Configure, Start/Exit to set this automatic update.

Gview does not move or otherwise translate the geometry. Rather, Gview moves your vantage point. However, it appears the geometry is rotating when you change the Gview, especially when using Gview, Dynamic.

Note: All planes are defined relative to the Work Coordinate System (WCS). If you reorient the WCS, all predefined planes are also reoriented.

Cplane

All geometry is created in a construction plane (Cplane). You select a construction plane by choosing Cplane from the Secondary Menu.

In 2D design, you work in a single Cplane. It can be any Cplane other than 3D. Usually the default plane (Top XY) is adequate for 2D work, so you do not usually need to change the Cplane or use Gviews.

In 3D design, you can work in any of the eight predefined planes or create planes as you need them. Top, Side, and Front are three of the predefined Cplanes provided by Verisurf. The following illustration shows the predefined construction planes.

The construction plane origin is set to 0,0,0 by default, although you can change it. See Construction origin overview for more information. If you change the Work Coordinate System (WCS) origin, the construction origin also changes.

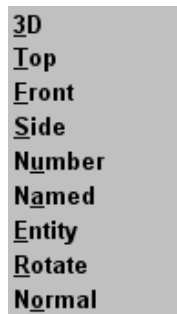
All planes are oriented with respect to the WCS. If you reorient the WCS, the planes will also be reoriented.

Note that in 3D design you need to be aware of the current Cplane or you will create the geometry in wrong and unintended orientations. Verisurf provides a 3D Cplane to allow you to create geometry that doesn't lie in a predefined plane, or geometry that lies in more than one plane. Changing to the 3D Cplane often resolves geometry misplacement. You can set the graphics view (Gview) to the same setting as the Cplane or view your work in the isometric or dynamic views.

Z depth is always referenced to the current Cplane. Always check the Z depth if the geometry you create isn't placed where you expect it to be. Z depth is measured along the Z axis from the construction origin. In the 3D construction plane, Z depth is referenced from the Top view.

When creating solids, Verisurf uses the Cplane of the selected geometry. For example, if you are creating a solid extrusion on geometry located in the Side Cplane, the extrusion will be created in the Side Cplane regardless of the current Cplane setting.

Press [Alt+F9] to display axes markers for the Cplane and the system coordinates.



Top, Front and Side have been shown and explained. The others are explained here.

3D – To set the 3D Cplane, choose Cplane, 3D from the Secondary Menu, or click on the 3D button on the toolbar.

The 3D Cplane allows you to draw in unrestricted 3D space instead of a 2D construction plane. The 3D Cplane allows you to create entities whose points lie in different planes, or to create entities (such as arcs or 2D splines) in planes other than the predefined planes.

When you need to create part geometry, for example fillets, on several faces of a 3D part, use the 3D Cplane so that you will not have to set the Cplane to match each face of the part.

3D Cplane uses the Top view as a reference for construction depth (Z depth) and for Xform moves (Xform, Translate function).

If an entity that you create in 3D is parallel to a defined plane, Verisurf creates the entity in the defined plane. Otherwise, Verisurf creates the entity in the Top plane at the current Z depth or at a depth specified by the entities that you used to create the new entity. For example, if you create an arc that is tangent to one entity, Verisurf uses the Z depth of the tangent entity to determine the construction depth of the new arc.

Note: When you create an arc or 2D spline in 3D that does not lie in a defined plane, Verisurf automatically creates a new plane and assigns it the next available number. This setting is used when measuring a part and the entities you are creating should be exactly where the probe picks them up at.

Number – Verisurf supports an unlimited number of Cplanes. Eight of the planes are predefined and can be identified by number:

1 - Top 5 - Right Side

2 - Front 6 - Left Side

3 - Back 7 - Isometric

4 - Bottom 8 - Axonometric

Views 1 through 6 correspond to the six faces of a cube. Isometric and axonometric are 3D views that show the top, front, and right sides of a model.

Note: All planes are defined relative to the Work Coordinate System (WCS). If you reorient the WCS, all predefined planes are also reoriented.

Verisurf supports Cplanes that you create, and assigns each new Cplane the next available number. You can retrieve the new Cplane by number or name it for retrieval by name.

Note: A user-defined Cplane must contain 2D geometry (such as an arc or a drafting note) for Verisurf to save it with the MC9 file. However, once the Cplane has been saved, you can delete the geometry from that Cplane and Verisurf still retains the Cplane. You can also save a user-defined Cplane by using the View Manager.

Named – If you want to change the origin and orientation of the Cplane, Tplane or Gview, you can set them to named views.

1. In the Secondary Menu, select WCS to open the View Manager.
2. Select a view in the list.
3. Right-click and select Set current.
4. Choose one of the options to set one or more of the system views to the current view.
5. Choose OK.

Verisurf updates the system view(s) you selected and changes the display in the Secondary Menu. For example, if you set the Cplane to "MyNamedView-Top," the Secondary Menu displays *nT next to Cplane. The asterisk (*) indicates the view origin is set to a point other than 0,0,0. The "n" indicates a named view being used.

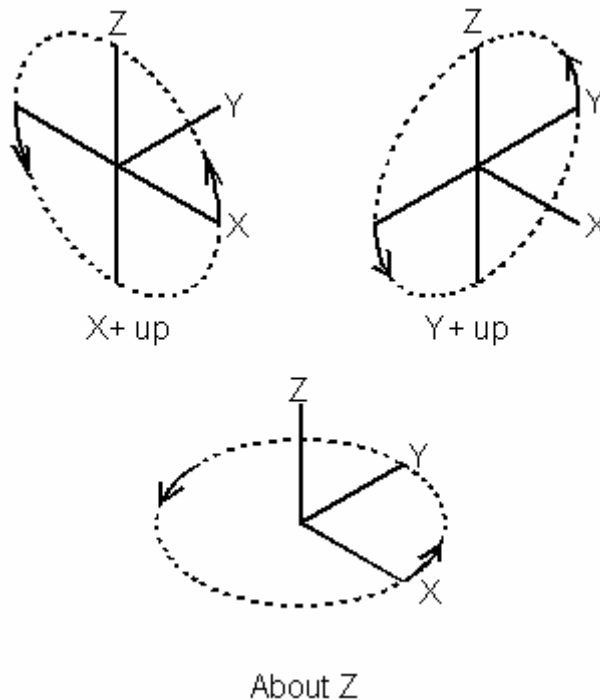
Entity – Choose Cplane from the Secondary Menu, then choose Entity to define a construction plane using existing entities in the graphics window. You can define the plane using one of the following selection options:

- A flat entity, such as a 2D spline or an arc, a solid face, or any other single entity that lies in only one plane.

- Two lines that lie in the same plane. The lines do not have to intersect, but they cannot be collinear. The order in which you select the lines is important. The first line you select sets the X positive direction and the construction depth (Z depth). The second line sets the Y positive direction.
- Three points that are unique and are not collinear.

Rotate – Choose Cplane from the Secondary Menu, then choose Rotate to rotate the current Cplane about an axis. You can choose both the axis and angle of rotation.

- X+ up - Rotates the positive X axis in a positive direction in the XZ plane (that is, about the Y axis) through the angle of rotation that you enter.
- Y+up - Rotates the positive Y axis in a positive direction in the YZ plane (that is, about the X axis) through the angle of rotation that you enter.
- About Z - Rotates the XY plane about the Z axis through the angle of rotation that you enter.



Normal – The normal of a line is perpendicular (90°) to it. Cplane, Normal lets you define a Cplane that is perpendicular to the selected line. The Z axis lies on the line that you select. Cplane, Normal creates an XY plane perpendicular to the line that you select

WCS

The Work Coordinate System (WCS) is the coordinate system in use by Verisurf at any given time. The WCS contains the orientation of the X-Y-Z axes plus the location of the zero point (the origin). It determines the orientation of the Top plane. Gviews and Cplanes are all oriented relative to the WCS.

As a default, Verisurf aligns the WCS with System View 1 – Top. However, you can use the View Manager to align the WCS with any system or user-defined view.

You can display the WCS origin icon in the graphics window at any time by selecting the "Display WCS XYZ axes" check box on the Screen tab of the System Configuration dialog box. This check box is deselected by default.

Verisurf's ability to redefine the orientation of the WCS is useful, but is only appropriate for specific applications. There are many circumstances that do not require a new WCS.

Cad shortcuts

Alt + 0	Set construction depth (Z depth)
Alt + 1	Set main color
Alt + 2	Set level manager
Alt + 5	Choose construction plane (Cplane)
Alt + 6	Choose graphics view (Gview)
Alt + S	Shading on/off
Alt + Z	Level Manager
Alt + ‘	Create two-point circle
Alt + -	Hide more entities
Alt + =	Unhide selected entities
Alt + F1	Fit geometry to screen
Alt + F2	Unzoom to 0.8
Alt + F3	Cursor tracking on/off
Alt + F4	Exit CAD
Alt + F5	Delete using window selection
Alt + F7	Blank entities
Alt + F8	System configuration
Alt + F9	Display all axes
Alt + F10	Maximize and minimize screen
Alt + arrow keys	Rotate Gview
F1	Zoom with window selection
F2	Unzoom
F3	Repaint
F4	Analyze entities
F5	Show Delete menu
F6	Show File menu
F7	Show Modify menu
F8	Show Create menu
F9	Show/hide part information and coordinate axes
F10	List all functions and execute selected
Esc	System interrupt or menu backup
Home **	Fit geometry to screen
Page up	Zoom in by 5%
Page down	Zoom out by 5%
Arrow keys	Pan