NASA Technical Memorandum 4431

P-97
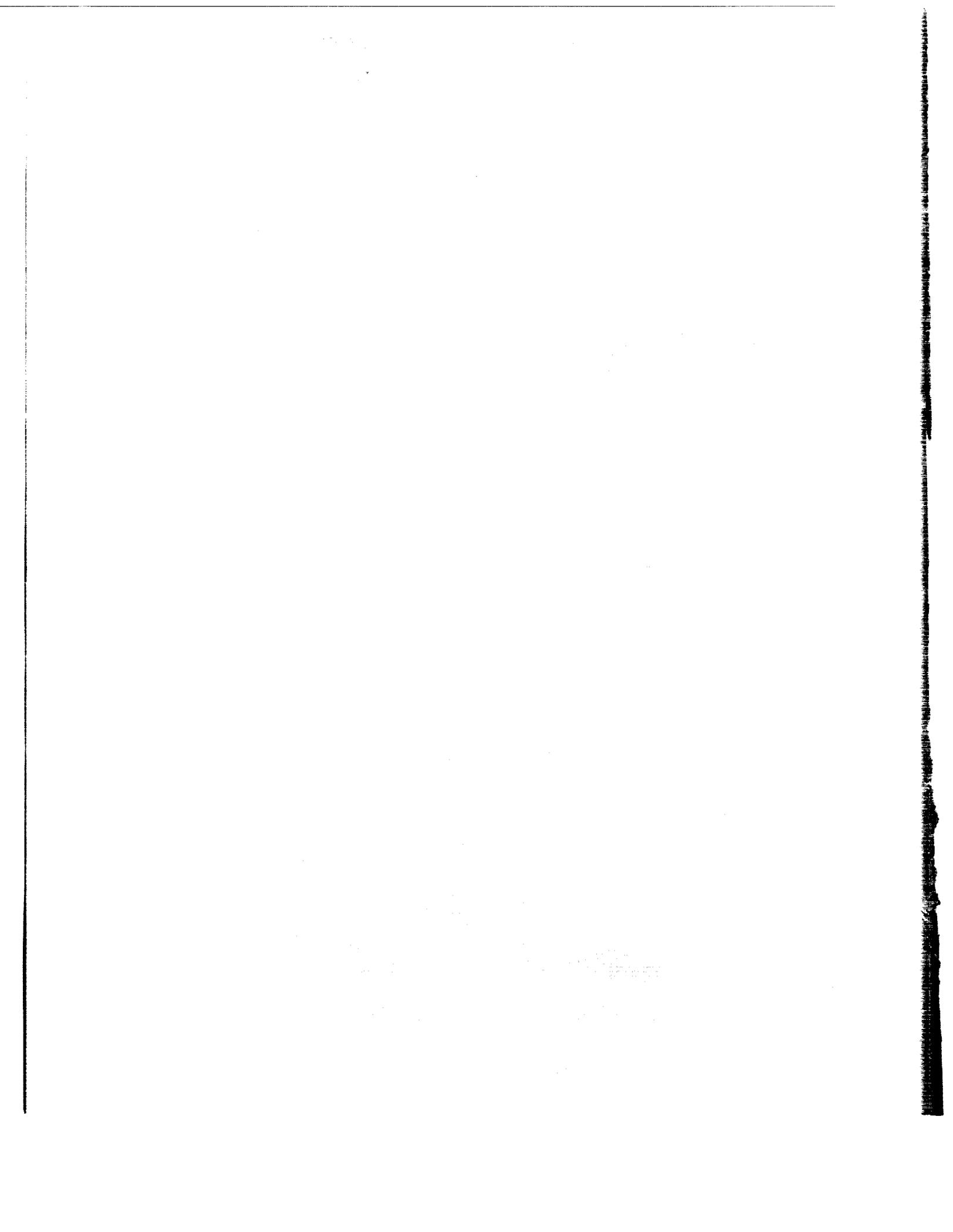
NASA Technical Memorandum 4431

Solid Modeling Aerospace Research Tool (SMART) User's Guide

Version 2.0

Mark L. McMillin
Langley Research Center
Hampton, Virginia

Jan L. Spangler and Stephen M. Dahmen
ViGYAN, Inc.
Hampton, Virginia

John J. Rehder
Langley Research Center
Hampton, Virginia

Contents

Preface

About the Software   The Solid Modeling Aerospace Research Tool (SMART) software package is used in the conceptual design of aerospace vehicles. It provides a highly interactive and dynamic capability for generating geometries with Bezier cubic patches. Features include automatic generation of commonly used aerospace constructs (e.g., wings and multilobed tanks); cross-section skinning; wireframe and shaded presentation; area, volume, inertia, and center-of-gravity calculations; and interfaces to various aerodynamic and structural analysis programs.

SMART was developed by the Vehicle Analysis Branch at NASA Langley Research Center (LaRC). It is also used by a number of organizations at LaRC, other NASA Centers, and at some aerospace corporations. To run SMART, you must have a 4D series workstation, which is manufactured by Silicon Graphics, Inc. For information about obtaining the software, you can send electronic mail to smart@vab01.larc.nasa.gov or send requests to the following address:

<div align="center">

MAIL STOP 365
NASA LANGLEY RESEARCH CENTER
HAMPTON VA 23681-0001

</div>

About the Guide   The *SMART User's Guide* serves as a basic description of the SMART program. It is intended to help you quickly review sections of the program and remind you of the more subtle details and algorithms in SMART. In the guide, we assume that you have some previous experience with SMART and that you have some basic understanding of aeronautical and aerospace design principles. For example, we do not explain structural concepts, such as spar, rib, and center of gravity; however, we do include a glossary of SMART terms.

The guide begins with the most basic concepts and then covers the more complex tasks, such as editing cross sections and converting files. Helvetica **boldface** type is used to reference SMART menus and other user-interface options. The guide is divided into the following chapters:

Chapter 1 "SMART User Interface" describes the program and the user interface. The layout of the screen, naming conventions, help facility, and general use of menus are described.

<div align="center">

xi

</div>

Chapter 2 "View Options" describes the various options available for viewing the created geometry.

Chapter 3 "TREE Mode" describes how SMART visually represents the relationship between the many objects in a model.

Chapter 4 "PRIMITIVES Mode" describes geometric primitives, which are the basic geometric shapes that are used to generate more complex objects. These primitives range from something as simple as a sphere to more complex objects, such as fuel tanks and wings.

Chapter 5 "CS SURFACE Mode" describes more complex geometry and how to use both Bezier (cubic) and Cartesian (point-by-point) curves to create cross sections in two dimensions and put them together in three dimensions to form surfaces.

Chapter 6 "IMAGES Mode" explains how to modify the appearance of your model, make certain parts of your drawing visible or invisible, and adjust the coloring of your model and the shading of its parts.

Chapter 7 "PACKAGING Mode" describes the process of putting all the pieces where they belong at the right size and orientation.

Chapter 8 "PROPERTIES Mode" covers SMART's ability to calculate weights, volumes, and areas of geometric objects and groups of objects.

Chapter 9 "PATCH TOOLS Mode" describes how to edit the individual patch structure that composes the object surface.

Chapter 10 "STRUCTURES Mode" describes how to add interior structure to wing primitives, such as ribs and spars.

Chapter 11 "PICTURES Mode" describes how to generate images of your model.

Chapter 12 "FILE Mode" describes how to save objects on data files on your disk and how to retrieve them later for editing.

Abbreviations

CG	center of gravity
CS	cross section
LE	leading edge
LMB	left mouse button
MMB	middle mouse button
OOSA	out-of-screen axis
RHP	relative height parameter
RMB	right mouse button
SGO	Silicon Graphics object
SMART	Solid Modeling Aerospace Research Tool
TE	trailing edge
TRP	transformation reference point
2-D	two-dimensional
3-D	three-dimensional

Chapter 1
SMART User Interface

This chapter describes a number of global features of the SMART user interface, including the different kinds of menus and input entry mechanisms. For an overview of the SMART program, you can perform the exercise in appendix A.

SMART Screen

Figure 1.1 shows the basic layout of the SMART screen. Menus and slider bars for each mode are displayed in the Menu Display Area. A number of special areas on the screen are described subsequently in this chapter.

TREE	PRIMITIVES	STRUCTURES		IMAGES	PROPERTIES	PICTURES	EXIT
FILE				PACKAGING	CALCULATOR		HELP

PLNR ROTA RCEN WIND Z-ZM X-ZM DLLY BGC DOFF GON PRSP 2-VU PLNR ROTA RCEN WIND Z-ZM X-ZM DLLY BGC DOFF GON PRSP 2-VU
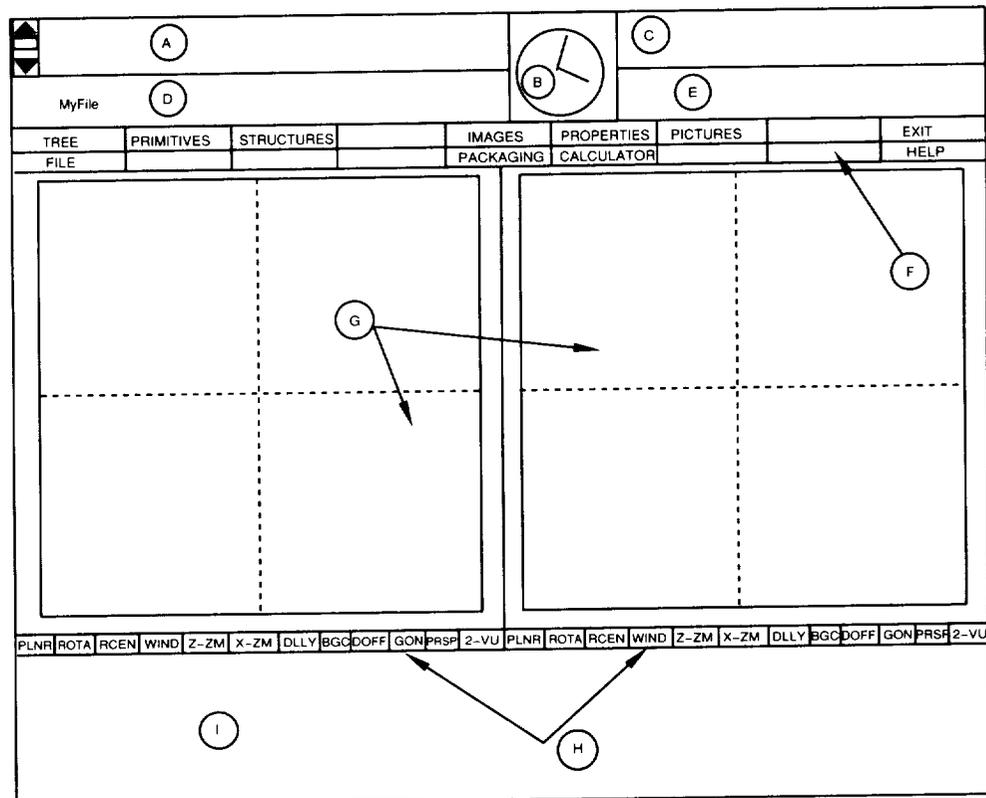
A. Textport
B. Clock
C. Function Name Area
D. Current Filename Area
E. Information Display Area
F. Mode Menu
G. View Windows
H. View Options Menu
I. Menu Display Area

Figure 1.1. Layout of SMART screen.

Menus

SMART has three kinds of menus: action menus, switch menus, and pop-up menus. Action menus cause something to happen in the SMART application, such as regenerate the shaded image. Switch menus change some parameter in the environment. After choosing an option from a switch menu, that option remains highlighted, unlike an action menu option. The STRUCTURES Mode uses pop-up menus. When you click on these menus, a series of choices "pop-up." To choose one of the choices, keep the mouse button depressed, move the cursor over your choice, and release the mouse button to indicate your choice. See the chapter on the STRUCTURES Mode for more information.

Most menus are displayed vertically; however, the Mode Menu and View Options Menu (fig. 1.1) are horizontal menus and are active in most of the modes. Note, you can leave any mode by selecting a new mode from the Mode Menu, even when you are in the middle of an entry procedure such as a wing definition.

Slider Bars

Slider bars are for selecting a value within a range. They allow you to make fine adjustments over a continuous range of values. The slider bar resembles a long, horizontal button with two small arrows on either side. Each slider bar displays a title and a current value. (See fig. 1.2.)

Figure 1.2. Slider bar.

Adjusting Values

To change the value, move the cursor on the slider bar and press one of the mouse buttons. As the mouse is dragged left or right within the slider bar, the value is updated appropriately. When you are finished, release the mouse button. The left mouse button (LMB) has the greatest resolution and changes the value slowly. The middle mouse button (MMB) has a medium resolution, and the right mouse button (RMB) has the least resolution and thus changes the value quickly. (See fig. 1.2.) The two small arrows on either side of the slider bar are used to make discrete changes to the value. Click on the left arrow to decrement the value or the right arrow to increment the value (the actual amount depends on which mouse button is used). You can also hold a mouse button down over an arrow. After an initial pause, the slider bar's value begins to increase or decrease. The value continues to change until either the mouse button is released or the minimum or maximum value is reached. When the maximum value is reached, the bar's value and title are displayed in green.

Using the Keyboard To Enter Values	Clicking on a slider bar when the Shift Key is pressed causes SMART to put a prompt in the Textport. Typing in a value after the prompt updates the value of the slider bar. As an alternative, you can use the Caps Lock key to toggle the keypad mode.
Entering Toggle Values	Slider bars are also used for entering toggle values. These slider bars display a title and either **True** or **False**. Clicking on the slider bar toggles the value between **True** and **False**.
On-Line Help	SMART provides a simple help facility that is toggled on and off by the **HELP** option on the Mode Menu. When activated, SMART gives you a quick summary of each menu option. Whenever you move your cursor over a menu or slider bar, SMART displays a message in the Information Display Area describing what happens when you click on that spot. For instance, if you move the cursor over **HELP**, the message "Turn on Help Facility" appears in the Information Display Area.
Cursor Wrap Around	In most SMART modes the cursor, which is displayed as a cross hair, wraps around the screen. If you continue moving the mouse to the right, even as the cursor reaches the rightmost boundary, it immediately jumps to the far left side of the screen and continues rightward. Thus, you can avoid moving the mouse long distances. For example, when the cursor is at the top of the screen and you need to select something at the bottom, move the cursor up and it wraps around to the bottom.
Special Screen Areas	Figure 1.1 shows the locations of the special screen areas described in this section.
Textport	The Textport displays information as SMART performs various functions as well as prompts you to input data (e.g., text strings or numbers). You can scroll the Textport forward and backward with the scroll bar on the left side of the menu.
Current Filename Area	The Current Filename Area displays the name of the file currently being edited. Note, if you write the model to a new file, the displayed filename does not change.
Function Name Area	The Function Name Area displays a message that describes the current mode of SMART. It also helps the development team find problems with the program.
Information Display Area	In addition to help messages, the Information Display Area also displays parameters such as point location, transformation data, and arc length while they are being edited.

3

Clock

The clock tells you the time based on the system clock. If the second hand stops moving, then SMART is performing a calculation. As long as the second hand moves, SMART is waiting for you.

Changing User Interface Colors

SMART lets you change the colors used in the interface through a *.smartrc* file in your home directory. Each line contains a string value specifying the color resource and three integer values (between 0 and 255) representing the red, green, and blue components of the desired color. Comment lines beginning with "#" and blank lines are allowed. The following table describes the color resources that you can change:

Resource	Description
BarColor	Base color of slider bars, Main Menu, and View Options Menu
BarHilite	Color of center panel of bar when cursor is over it
BarTextColor	Color of text on slider bars, Main Menu, and View Options Menu
MenuColor	Base color of menus
MenuTextColor	Color of text on menus
MenuHilite	Color of menu choice when mouse is over it
MenuModeOn	Color of Mode Menu choice when mode is active
MenuTitleColor	Color of background of menu titles
BarHandleColor	Color of bar indicator without mouse over it
BarHandleHilite	Color of bar indicator with mouse over it

Iconifying the SMART Window

Because SMART fills the entire screen, you must simultaneously depress the right mouse button and the Alt Key to access the window manager functionality. You should iconify SMART when you need to work in another window because SMART recognizes mouse clicks even when the SMART window is behind your current window.

Chapter 2
View Options

SMART allows you to change how you view your object or model. For example, you can use the View Options to change the angle or the magnification of an object. Figure 2.1 shows the layout of the View Options Menu. Although many of the View Options are discussed in other chapters, this chapter explains how to use them.

PLNR	ROTA	RCEN	WIND	Z–ZM	X–ZM	DLLY	BGC	DOFF	GOFF	PERSP	2–VU

Figure 2.1. View Options.

The view can be changed with any of the options located beneath each View Window. However, in some modes, such as CS Surface, some of these options are disabled. For the View Options with two labels (e.g., 2-VU/1-VU), you click the mouse on the View Option to switch between the two views. The label shown in the View Options Menu corresponds to the view that is currently active. For most of the other View Options, you click on the View Option and hold the mouse button down as you move the mouse to adjust the view. The following sections describe these View Options.

NOTE: *None of the View Options are saved with the model in the SMART data file.*

PLNR
(Planar View)

The **PLNR** View Option changes the View Window to a planar view, that is, a two-dimensional (2-D) projection of the model. You can access three standard aerospace views by using the three mouse buttons as follows:

1. The LMB displays the Y-Z plane from the $-X$-perspective.
2. The MMB displays the X-Z plane from the $+Y$-perspective.
3. The RMB displays the X-Y plane from the $+Z$-perspective.

By using the Shift Key with the mouse or by clicking twice on the **PLNR** View Option, you can access the same view from the opposite direction.

ROTA (Rotate)

Depending on which mouse button is held down, the **ROTA** View Option rotates the view in a variety of ways. The view is adjusted by keeping the mouse button depressed and moving the mouse. The rotation parameter that is being changed is displayed in the Information Display Area.

With the LMB depressed, the azimuth is adjusted by the left and right motion of the mouse, and elevation is adjusted by the up and down motion of the mouse. (This discussion assumes that you start with the X-Y planar view.) The easiest way to understand this step is to try it. The MMB works in a similar manner except that SMART shrinks and grows the image so that it fits in the window. With the RMB depressed, moving the mouse right twists the view clockwise, and moving the mouse left twists the view counterclockwise.

RCEN (Rotation Center)

The **RCEN** View Option adjusts the center of rotation (indicated by the axis system drawn in blue dashed lines). Press the LMB and hold it as you move the mouse to adjust the center along the X-axis. The MMB adjusts the Y-axis, and the RMB adjusts the Z-axis. You can use this option to zoom in on a particular area and keep that area in the window while you rotate the view.

WIND (Window)

By clicking the mouse on the **WIND** View Option, SMART sizes your drawing to fit completely in the View Window. The center of rotation is reset to be the center of the box formed by the grid.

NOTE: *You may need to use the WIND option after reading in a file.*

Z-ZM (Z-Zoom)

The **Z-ZM** View Option controls the near and far clipping planes (front and back of the View Box). Both planes move together as the LMB is kept depressed and moved. If the LMB and the MMB are pressed together, the mouse moves the clipping planes closer together or farther apart as the mouse moves from left to right. The clipping planes are displayed in both views whenever **Z-ZM** is selected, so you can use the other view to help visualize what portion of your model is between the clipping planes.

X-ZM (X-Zoom)

The **X-ZM** View Option lets you pan or zoom the view of your object. The LMB pans the view and the LMB and MMB together zoom the view.

DLLY (Dolly)

The **DLLY** View Option is only available in the perspective view mode. This option lets you use the LMB to adjust the distance between you (the viewer) and the object. With the camera model as an example, **X-ZM** is analogous to adjusting the zoom lens on the camera, and **DLLY** is analogous to moving the camera closer to the object. **DLLY** is not necessary in the orthogonal view because no perspective exists.

BGC
(Background Color)

When the **BGC** View Option is selected, the menus in the Menu Display Area are replaced by four slider bars—one for each color (red, green, blue) and one for intensity. You can then adjust these bars to create the background color you desire.

DOFF/DON
(Depth-Cueing
Off/On)

Clicking the mouse on the **DOFF/DON** View Option toggles the depth-cueing feature. Depth cueing means the model is displayed in a gray tone on a black background, which gives greater brightness to the portion of the drawing closest to the view point and adds a sense of depth to the view. Anti-aliasing is also performed; that is, jagged lines are smoothed. This option is not compatible with shading; when invoked with shading active, the display is interesting but no longer meaningful.

GOFF/GON
(Grid Off/On)

The **GOFF/GON** View Option toggles the coordinate grid off or on.

ORTH/PRSP
(Orthographic/
Perspective)

The **ORTH/PRSP** View Option toggles the projection between the perspective and orthographic options. The perspective view reduces those parts of the drawing that are farther from the viewer; the orthographic view does not.

2-VU/1-VU
(Two View/
One View)

The **2-VU/1-VU** View Option toggles the number of View Windows between one and two. In some SMART modes, this option is disabled.

Autowindow

In addition to the standard View Options available on the View Options Menu, an automatic windowing feature is available in many SMART modes. When turned on, **Autowindow** ensures that all objects are entirely visible in the View Windows.

NOTE: *The Autowindow option automatically resets the rotation center.*
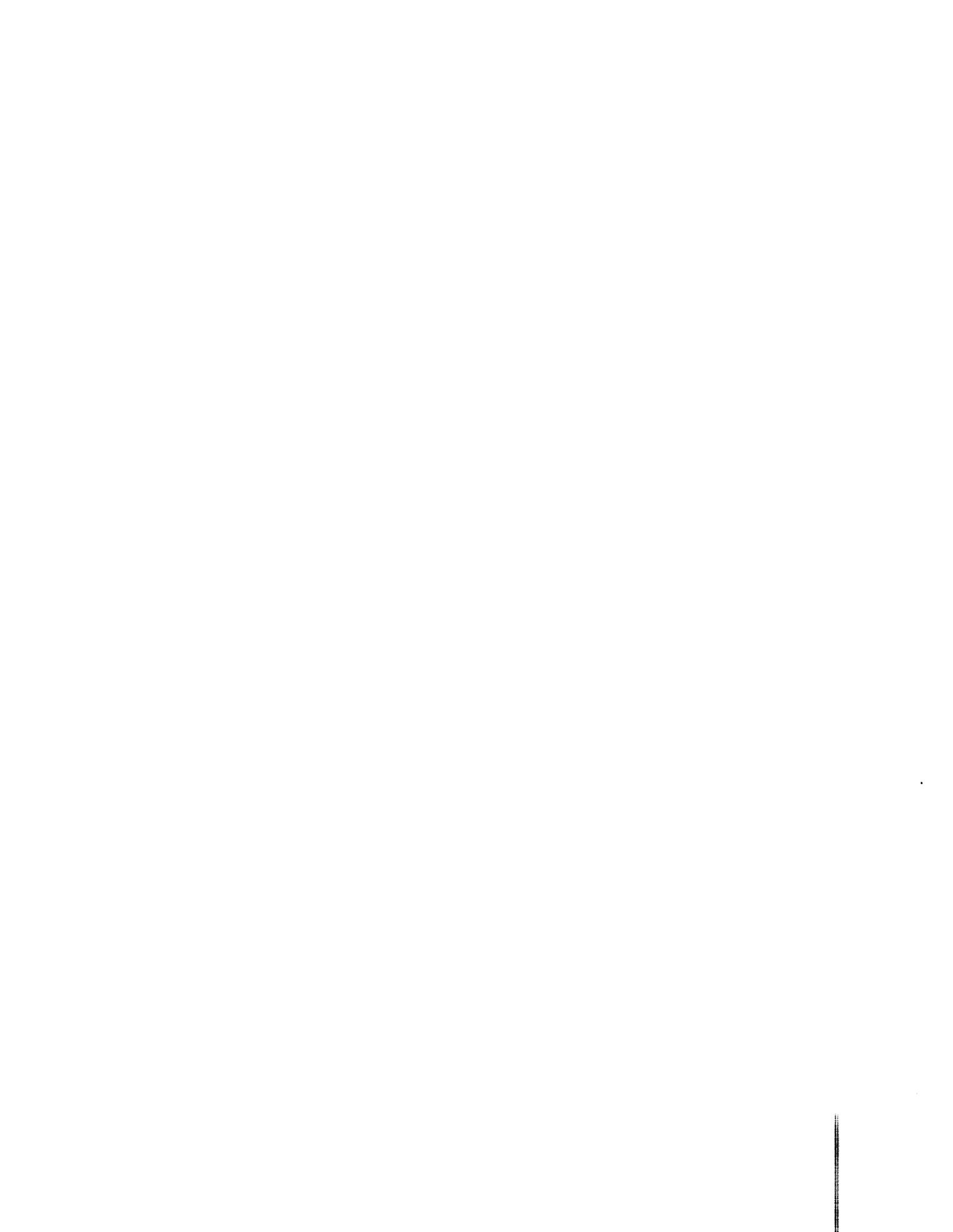
Chapter 3
TREE Mode

This chapter describes the Tree and the TREE Mode in SMART.

Defining the Tree

The Tree represents the hierarchical relationship between the components of your model. It allows you to group objects together to form more complex objects. The Tree has the following three components:

System node   The system node is the first node in the Tree; it can be considered the root of the Tree. It is part of the SMART system and cannot be deleted or moved. The system node is highlighted in light blue.

Group node   Group nodes are members of the Tree that contain other groups or objects. A group node can be collapsed, which means it hides the nodes beneath it. If the group node is collapsed, it is highlighted in light blue; otherwise, it is outlined in light blue.

Leaf node   Leaf nodes contain only a primitive or cross-section surface and are displayed in green.

Figure 3.1 shows a sample Tree with two group nodes and five leaf nodes.
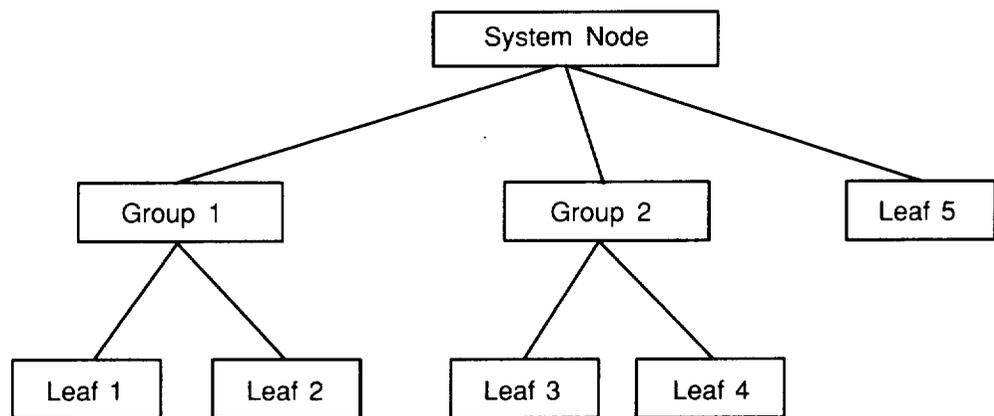
Figure 3.1. Sample Tree.

9

Viewing the Tree

Two different views of the Tree appear in the View Windows. In the left View Window, the entire Tree is displayed, along with the View Box, which shows the portion of the Tree displayed in the right View Window. By default, the two views are identical until changed with the zoom and pan features described subsequently. All user interaction occurs in the right View Window.

In a large model, the Tree can be so big that the node labels are too small to read. The View Box, used much like a magnifying glass, allows you to zoom and pan the Tree to let you see particular portions of the Tree. To help you select nodes, the cursor sticks to the nodes as you move it.

The MMB, while depressed, allows you to move the View Box to different parts of the Tree (translation), while the RMB zooms in and out on the Tree as you move the mouse left and right. These features are operative at all times in the TREE Mode to help you select nodes.

In some operations, only nodes of a specific type (light, dimension, etc.) are selectable. To select a node, simply put the cursor over the node (the node becomes highlighted) and use the LMB to indicate your choice. A small display in the lower left of the Menu Display Area shows the objects in the node that is beneath the cursor. Note, all of the node's children are highlighted when you rest the cursor on the desired group node.

SMART allows you to collapse parts of your Tree to hide unneeded detail. Any group node can be collapsed by holding down the right or the left Shift Key and clicking the LMB on the desired node. The Tree is then redrawn without all the subordinate groups and leaves, and the collapsed node is drawn in reverse video. To again access these hidden group and leaf nodes, simply repeat the shift-click procedure on the collapsed node, and the Tree is redrawn with all subordinate nodes visible.

Tree Options

With the exception of the add command, all of the Tree operations can be repeated after the initial command is selected. To abort any move or copy command, move the cursor outside of the right View Window. The following sections explain these commands.

• move

The **move** option allows you to rearrange your Tree. After choosing **move** from the menu, SMART waits for you to indicate which node (and all its children) that you want to move. When you make your selection, an image of the entire selected subtree becomes attached to the cursor and moves with the mouse. SMART then waits for you to select a group node to indicate the destination of your move. The

subtree that you have just moved then becomes the leftmost child of the node that you selected.

● **copy**　　　　　　　The **copy** option works the same as the **move** option except the original node is not removed from the Tree.

● **add**　　　　　　　You can create a new group node with the **add** option. After selecting the option, SMART prompts you for the name of your new group node, then SMART creates the new group as the leftmost child of the system node. The name, which must not contain blanks (use underbars), is truncated to 12 characters.

NOTE: *When a node is added, moved, or copied to a collapsed node, the operation still takes place, but the new node is not displayed.*

● **rename**　　　　　　Group and leaf nodes can be renamed (often desirable after **copy**) with this option. Simply select **rename** from the menu, select the node to rename, and type in the new name when prompted.

● **delete**　　　　　　The **delete** option deletes leaf or group nodes; SMART prompts you for verification after each selection before deleting it. If the system node is selected all the nodes in the Tree are deleted, but the system node itself remains.

● **combine**　　　　　The **combine** option combines the geometry of all the objects in a group node into one leaf node. This option is useful after creating an object whose individual parts you no longer need to edit. This option also helps reduce the size of the Tree. After selecting **combine**, select the node that you want to combine. Next, you are prompted for a name for the new node in the Textport. SMART then applies all transformations on the object and then places the object in the new node, which is the leftmost child in the Tree.

11

Chapter 4
PRIMITIVES Mode

Part I—Geometric Primitives

Part I of this chapter describes the geometric primitives that you can create in SMART. Basic shapes, such as cubes, wedges, and spheres, are available as well as more specialized aerospace shapes, such as wings and tanks. Primitives are defined by as few as 1 parameter (e.g., a sphere) or as many as 12 parameters (e.g., a tank). In SMART, primitives are constructed from Bezier patch surfaces and are used as building blocks in more complex constructions.

To include a basic or aerospace primitive in your model, select **PRIMITIVES** from the Mode Menu. Four menus appear in the Menu Display Area. (See fig. 4.1.) The left menu lists the miscellaneous primitives, the two middle menus list the basic primitives, and the right menu lists the aerospace primitives.

Misc. Primitives	Basic Primitives	Basic Primitives	Aerospace Primitives
text	sphere	pyramid	tank
light source	box	prism	wing
line dimension	tapered box	torus	
angle dimension	cylinder	toroidal segment	
	cone	spherical segment	

Figure 4.1. Menus for miscellaneous, basic, and aerospace primitives.

Adding Basic Primitives

The **Basic Primitives** menus provide you with 11 different shapes: sphere, box, tapered box, cylinder, cone, pyramid, prism, torus, toroidal segment, spherical segment, and tube. Once you select the desired shape, SMART prompts you to indicate a name and the dimensions for the primitive. For example, to add a box to your model, type in the following:

Enter name for box: big_cube

Enter length: 100

Enter width: 100

Enter depth: 100

13

PRECEDING PAGE BLANK NOT FILMED

A cube with dimensions of 100 units is added to the View Windows. As with all basic primitives, the box is positioned so that its center is at the origin (0,0,0). Adding a basic primitive automatically resizes the View Windows so that the created object lies within their boundaries.

Figure 4.2 shows the dimensions that define the various basic primitives. You can enter dimensions as either decimal or integer numbers. The subtended angles entered for toroidal and spherical segments are accurate to one decimal place.

| Sphere | Box | Tapered box | Cylinder |

| Cone | Pyramid | Prism | Torus |

| Toroidal segment | Spherical segment |
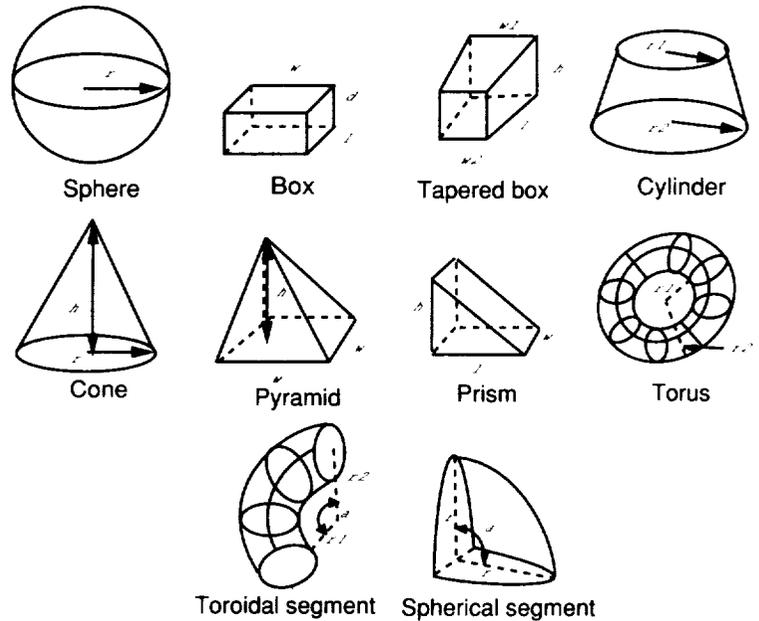
Figure 4.2. Basic primitives.

Tanks

This section describes how to create and edit tank primitives. When a tank is selected, SMART prompts you to perform one of the following commands:

1. Add a new tank to your model by typing in a name.

2. Edit an existing tank by selecting a tank primitive that you have already included in your model.

After you enter the requested information, two different views appear on the screen. The left View Window shows the tank with respect to the rest of your model. SMART draws the tank you have added or selected in white so that it is set apart from the other objects in your model. The right View Window provides a close-up view of the

tank. Only the tank being edited is shown. SMART lets you define tanks having one to eight bubbles. Each bubble is a chamber composed of two domes capping opposite ends of a cylindrical barrel.

A tank barrel is composed of three sections: the constant section, the blending section, and the taper section. These sections are shown in figure 4.3. The constant section is the portion of the barrel length that has a constant width, that is, the diameter of dome 1. The taper section is the portion of the barrel length where the width of the barrel is steadily narrowing to the diameter of dome 2. The middle section is called the blending section, which resolves the slope discontinuity between the constant section and taper section. Only one section exists (the constant section) when the diameters of dome 1 and dome 2 are equal.
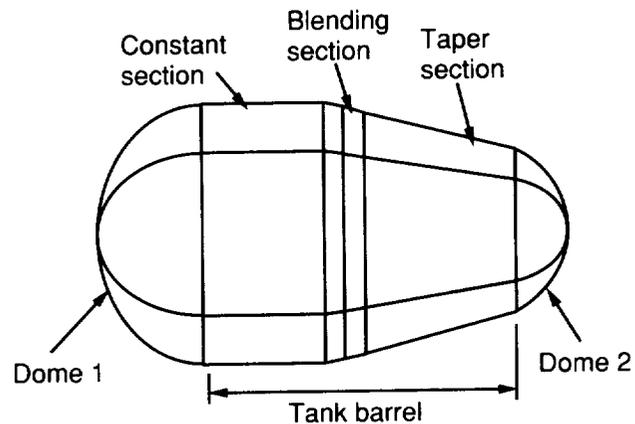
Figure 4.3. Tank bubble sections.

Calculating tank volume

The Menu Display Area has a **Calculate Volume** option. (See fig. 4.4.) Selecting this option causes SMART to calculate the area and volume of the tank. The results are shown in the Textport.

Slider bars for editing tank

This section describes the 12 slider bars that SMART provides for editing the shape of a tank. These slider bars are shown in figure 4.4. Most slider bars affect all the bubbles in a tank. For example, changing the **Barrel Length** changes the size of each barrel in the tank. A few slider bars, however, modify only one bubble at a time; these are the **Dome Radius**, **Center Angle**, and **Intersection Angle** slider bars. The **Current Bubble** slider bar lets you select the tank bubble that you want to modify. In the right View Window, SMART highlights the current bubble by drawing it in blue.

15

Calculate Volume

▲	No. of Bubbles = 2	▼		▲	Intersection Angle = 120.0	▼		▲	Barrel Length = 1.10	▼
▲	Current Bubble = 1	▼		▲	Center Angle = 0.0	▼		▲	Taper Factor = 1.00	▼
▲	Dome Radius = 0.275	▼		▲	X–Center Offset = .000	▼		▲	Taper Break = .5	▼
▲	Eccentricity = 0.707	▼		▲	Y–Center Offset = .000	▼		▲	Taper Blend = .100	▼

Only displayed when
taper factor is not 1
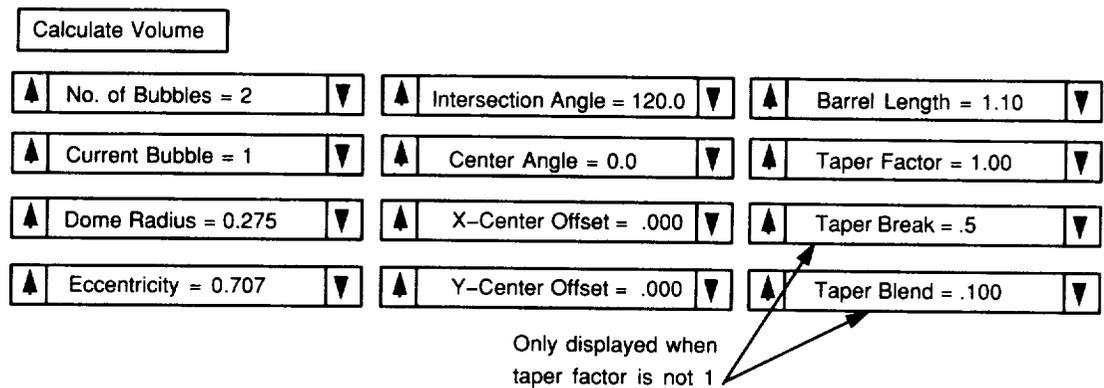
Figure 4.4. Slider bars for editing tanks.

Even a small change to a slider bar's value results in SMART performing a large number of calculations to redefine the tank's geometry. The message "Please Wait" appears in the Function Name Area while these calculations are being performed. The function of each slider bar is as follows:

● **No. of Bubbles**   Specifies the number of tank bubbles; range is 1 to 8.

● **Current Bubble**   Selects the tank bubble that you want to edit. Bubbles are numbered in increasing order according to their positions along the positive X-axis. For example, set this slider bar to 2 to modify the second tank bubble from the left. To edit all bubbles simultaneously, set the current bubble to 0.

● **Dome Radius**   Changes the dome radius, which is measured at the point where dome 1 meets the barrel.

● **Eccentricity**   Changes the ratio of the dome cap's height along the Z-axis to its width across the Y-axis. Eccentricity ranges from 0 to 1. For example, 0 defines dome caps that are completely flat, and 1 defines dome caps that are perfect hemispheres.

● **Intersection Angle**   Adjusts the included angle between tangents of adjacent barrels at the plane of intersection (fig. 4.5). The slider bar adjusts the angle between the current bubble and the next consecutive bubble. The intersection angles can range from 0° to 180°. An intersection angle of 0° completely separates the two tank bubbles and an intersection angle of 180° merges the two bubbles into one chamber. The default intersection angle between two adjoining bubbles is 120°.

16
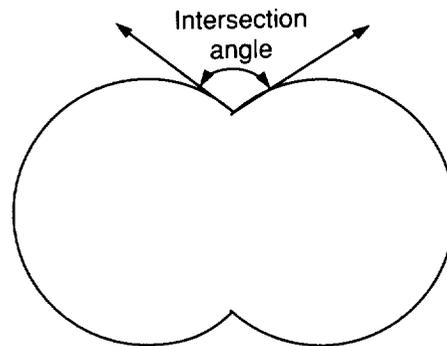
Figure 4.5. Intersection angle.

● **Center Angle**

Adjusts the angle at which the line connecting the centers of two adjoining bubbles (the current bubble and the next consecutive bubble) intersects the horizontal axis (fig. 4.6). Range is −45° to 45°.
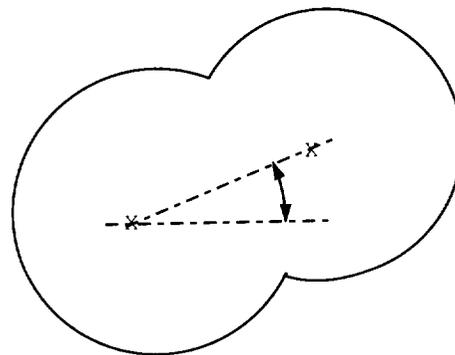
Figure 4.6. Center angle.

● **Y-Center Offset**

Changes how much the Y-coordinate of dome 1's center is offset from the Y-coordinate of dome 2's center.

● **X-Center Offset**

Changes how much the X-coordinate of dome 1's center is offset from the X-coordinate of dome 2's center.

● **Barrel Length**

Changes the length of the tank's barrels.

● **Taper Factor**

Adjusts the ratio of the radius of dome 1 to dome 2; range is 1 to 4. For example, a **Taper Factor** of 1 indicates that dome 1 and dome 2 have equal radii.

17

● **Taper Break**

Changes the length of the constant section; this value is specified as a percentage of the total barrel length. For example, a taper break of 0.5 means that the constant section is half the total barrel length. This slider bar is not displayed when the **Taper Factor** is 1.

● **Taper Blend**

Changes the length of the blending section; this value is specified as a percentage of the total barrel length. This bar is not displayed when the **Taper Factor** is 1.

Default tank

The default tank is a symmetric double-bubble tank that is centered at (0,0,0). The length of the tank lies along the Z-axis. The initial size of the tank is based on the size of the right View Window; barrel lengths are one-half the window width, and the diameters of the tank bubbles are one-fourth this distance. This guide refers to the dome cap positioned on the positive Z-axis as dome 1 and the dome on the negative Z-axis as dome 2 (fig. 4.3).

Wings

This section describes how to create and edit wing primitives. You can create wings with both single-delta and double-delta shapes. The distinction between single-delta and double-delta wings is that a break chord divides the span of the latter. SMART includes clipped-delta or trapezoidal shapes in the delta category.

Default wing

The default wing is a single-delta wing with the following characteristics:

Leading edge	60.00 degrees
Trailing edge	0.00 degrees
Area	600.00
Span	40.998
Aspect ratio	4.0
Root chord	46.53
Tip chord	9.31
Taper ratio	0.20

If you are adding a wing, this information is displayed in a small window in the lower left corner of the screen. If you are editing a wing, the window shows the results of your last edit. SMART updates the information as you make changes to the wing. At any point in time, the displayed information accurately reflects the characteristics of the wing displayed in the View Windows.

18

If the wing is a double delta, the leading- and trailing-edge angles are replaced by Inboard LE, Inboard TE, Outboard LE, and Outboard TE. Inboard refers to the inner span section and outboard to the outer span section. The information display also lists the length of the break chord.

Planform display

This section describes how you can change the shape of your wing by using the cursor to move the vertices of the planform. The right View Window displays the planform (fig. 4.7). It is a fixed planar view, and all View Options are turned off.
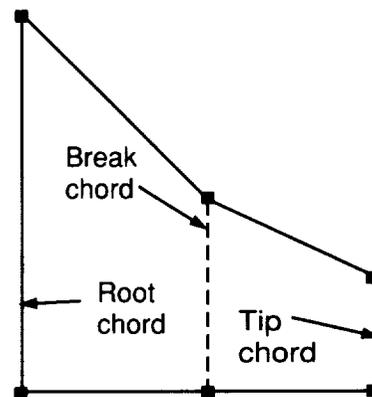
Figure 4.7. Planform display for double-delta wing.

As you move the cursor into the right View Window, the cursor changes from a cross hair to a small white square. The planform vertices are also marked by small squares. To lock onto a vertex, move the cursor over it and press the LMB. As long as the LMB remains depressed, you can move the vertex vertically by moving the cursor up or down.

You can also move the break or tip chord as a whole. Selecting either end point of the desired chord with the MMB lets you move the chord vertically. Selection with the RMB moves the chord horizontally.

Slider bars and menus for wing editing

As shown in figure 4.8, SMART provides the following seven slider bars for adjusting wing parameters:

19

Airfoils	Airfoils	Delta
00xx–63	00xx–64	single
00xx–(6.2)3	00xx–(5.777)4	double
00xx–(6.2)5	00xx–(4.6)4	Autowindow
00xx–55	00xx–(5.19)4	on
00xx–(4.125)5	00xx–(5.57)4	off
00xx–(5.171)5		

Only displayed when
wing is double delta

▲ T/C (root) = 0.100 ▼
▲ T/C (tip) = 0.100 ▼
▲ Dihedral Angle = 0.000 ▼
▲ Incidence Angle = 0.000 ▼
▲ Twist Angle = 0.000 ▼
▲ Scale Factor = 1.000 ▼
▲ Which Section = 1 ▼
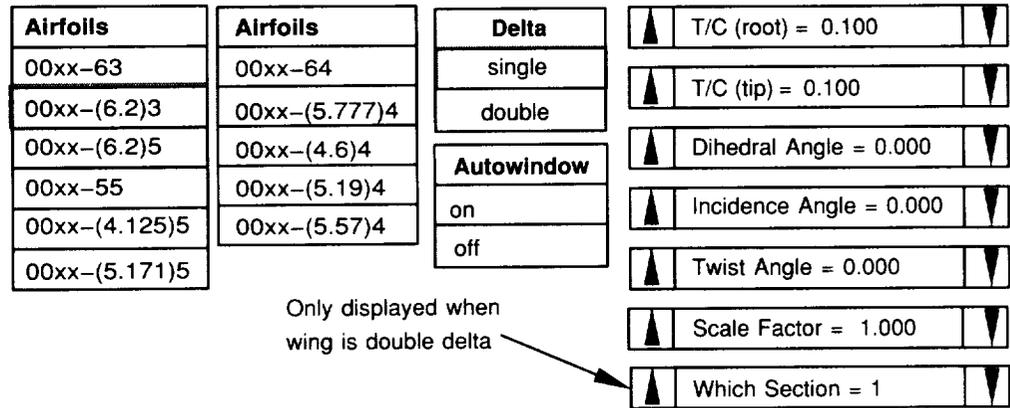
Figure 4.8. Menu and slider bars for editing wings.

- **Which Section**

Specifies which section to edit when adjusting the parameters for a double-delta wing; section 1 is the inner span section and section 2 is the outer span section. This slider bar is not displayed if you are working on a single-delta wing.

- **T/C (root)**

Airfoil thickness ratio for root rib; range is 0 to 0.3.

- **T/C (tip)**

Airfoil thickness ratio for tip rib; range is 0 to 0.3.

- **Dihedral Angle**

Range is $-45°$ to $45°$.

- **Incidence Angle**

Range is $-45°$ to $45°$.

- **Twist Angle**

Range is $-45°$ to $45°$.

- **Scale Factor**

Shrinks or expands the wing. Values with a magnitude greater than 1 expand the wing; values with a magnitude less than 1 shrink it. Range is 0.001 to 1000.

The following menus are located next to the slider bars:

- **Delta**

Specifies whether the wing has one or two sections. A break chord is added to the wing geometry when you change a wing to a double delta.

- **Autowindow**

Turns the autowindowing feature on and off. The chapter on View Options describes this feature.

● **Airfoils**   At the far left of the Menu Display Area, 2 menus list 11 National Advisory Committee for Aeronautics (NACA) airfoils: 00xx-63, 00xx-(6.2)3, 00xx-(6.2)5, 00xx-55, 00xx-(4.125)5, 00xx-(5.171)5, 00xx-64, 00xx-(5.777)4, 00xx-(4.6)4, 00xx-(5.19)4, and 00xx-(5.57)4. (See ref. 1.) The wing sections of the default wing use the 00xx-(6.2)5 airfoil. To change the wing sections used by your design, select another airfoil from the menus.

Part II—Miscellaneous Primitives

Part II of this chapter explains how to use PRIMITIVES Mode to add text, light, and dimension objects to your model. Light objects let you specify local light sources to use for shading your model. Text and dimensions are generally used to annotate the model. To add a light, text, or dimension to your model, select the desired option from the **Misc. Primitives** menu, which is shown in figure 4.1.

Text Objects

Text objects are used to add labels, captions, or other strings of text to the scene in the View Windows. Each text object is composed of 1 line of text, which includes up to 80 characters. SMART provides tools for editing your text, including the size, font, and color.

When you select **text** from the **Misc. Primitives** menu, SMART prompts you to enter a name for the text object and then enter the text. When you have finished typing, press <Enter>. SMART adds the text to the scene in the View Windows.

Initially, your text is positioned at the point (0,0,0); It is drawn in green, has a line width of two, and uses the simplex font. You can modify these default parameters by using the options in the PACKAGING and IMAGES Modes. By default, text does not rotate with the view. Instead, text remains facing forward so that it can be read. Once text has been rotated in PACKAGING Mode, even as little as one-tenth of a degree, it then rotates with the view.

Light Objects

Light sources are used when you shade your model in IMAGES Mode; however, the lighting effects are not visible when your model is displayed as a wireframe. These light sources shine light in every direction. You can create up to six lights for your model.

To create a light, choose **light** from the **Misc. Primitives** menu. As when creating any new object, SMART prompts you to enter a name. Type a name and press <Enter>. A light icon, represented as a white asterisk, is added to the scene in the View Windows. The appearance and effect of light sources can be modified with the options described in IMAGES Mode.

The first light source added to your model replaces the default light source positioned at (5000,5000,5000); however, when you add a new light source, it is initially positioned at (0,0,0). The advantage of replacing the default light source is that you can move the new light source to any position that you want with the options described in PACKAGING Mode. Moving the light changes where shadows and highlights appear.

Line Dimensions

Line dimensions are used to indicate distance measurements. To create a line dimension, select **line dimension** from the **Misc. Primitives** menu. SMART restricts the View Window to an *X-Y*, *Y-Z*, or *Z-X* planar view of your model. The following View Options are disabled while **line dimension** is selected: **ROTA, RCEN, WIND, Z-ZM, DLLY, ORTH/PRSP, 2-VU/1-VU**.

When you move the cursor into the View Window, a white square appears at the center of the cursor, and the Information Display Area shows the current cursor coordinates. Next, move the cursor to the starting point and click the LMB. As you move the cursor, a line spans the distance between the starting point and the current cursor position, and the length is displayed at the line's mid point. After clicking the LMB a second time, you are prompted to enter a name for the new line dimension.

SMART adds a black dimension line to the view. To avoid adding the line dimension to your model, press <Enter> without typing in a name. To exit the **line dimension** option, choose any item from the Mode Menu. You can modify line dimensions with the options described in IMAGES and PACKAGING Modes.

Angle Dimensions

You can create angular dimensions in the same way that you create line dimensions. The main difference is that you need three points to specify the angle. Simply follow these steps:

Click the mouse on **angle dimension** in the **Misc. Primitives** menu. Move the cursor into the View Window and click the LMB on the vertex of the angle. Then, move the cursor to where you want the angle to start and click the LMB. As you move the cursor, an arc is displayed with the measured angle shown in degrees. Now move the cursor to where you want the angle to end and click the LMB. You are then prompted to enter a name for the new angle dimension. SMART adds a black arc to the view. To avoid adding the angle dimension to your model, press <Enter> without typing in a name. You can modify angle dimensions with the options described in IMAGES and PACKAGING Modes.

23

Chapter 5
CS SURFACE Mode

This chapter describes procedures for the generation and manipulation of cross sections and surfaces. When **CS SURFACE** is selected from the Mode Menu, SMART prompts for the object name (unless an existing object in the Tree can be edited, in which case you are asked whether you want to edit an existing object). The CS SURFACE Mode menus are shown in figure 5.1.

Node			Edit	Edit	Edit	Input	
Bezier CS	▲ CS Width = 2 ▼		copy	move	reverse	cubic	▲ Rotate = 20.0 ▼
Cartesian CS	▲ CS Center = 2 ▼		convert	delete	break	conic	▲ Translate = 20.0 ▼
surfaces	▲ All CS ▼		sort	close	split	arc	▲ Scale = 20.0 ▼
	▲ Extract T = 2 ▼		flatten	key in	fillet	linear	
	▲ Extract S = 2 ▼		slope				
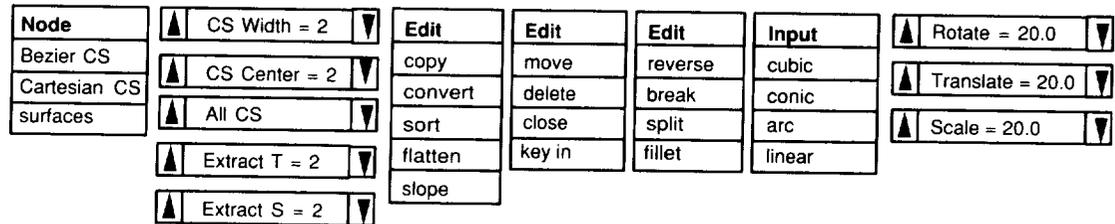
Figure 5.1. Menus for CS SURFACE Mode.

When working with cross sections, all the View Options are available in the left View Window. The right View Window, however, is restricted to planar views, although the selected edit plane can be rotated around the out-of-screen axis (OOSA) by using the RMB on the **ROTA** View Option. You can also use **X-ZM**, both pan and zoom. **ORTH** is the only view available, and you cannot use single view while working with cross sections.

SMART allows you to select more than one cross section at a time so that you can operate on them as a group. The selection is made with the **CS Center** and **CS Width** slider bars. The **CS Center** slider bar ranges from 0 to the number of cross sections in this object. The **CS Width** slider bar controls the number of cross sections on either side of the center curve; this number is called the edit range. If you have only a few cross sections, use the RMB to adjust the slider bar. For more cross sections, use the MMB, and for many cross sections use the LMB. You can also select all the cross sections by clicking on the **All CS** slider bar. The edit range appears highlighted in white in the left View Window and includes the only cross sections available for editing. These cross sections appear in the right View Window. The Bezier, Cartesian, and surface modes are specified with the Mode

menu. These modes, along with some common options, are described in the following sections.

Bezier CS

Each Bezier curve has four control points, each of which can be moved to change the shape of the curve. (For details on the underlying mathematics, see appendix B.) Each point is displayed in a color indicating its type or status. Yellow indicates a slope control point—each curve has two. Red indicates a curve end point with C0 (discontinuous slope) continuity. Green indicates that the two curves are C1 (slope) continuous at these end points. Blue indicates an end point of a cross section.

Inputting Bezier cross sections

To enter a new curve, select the input option from the **Input** menu (e.g., **cubic, arc, line, conic**). All input options begin by specifying the starting point of the cross section. Next, the Textport prompts you for the cross-section station (position on the OOSA). You can then enter any number of curves by using any input option, even switching between options for different parts of the curve. A dotted construction line appears on a curve as you append another curve to it. After a curve is defined, it appears with the rest of the cross sections in the left View Window. Note, although entered differently, all curves become cubic curves after entry and can be adjusted and modified as such. Special entry characteristics of the particular curves are lost.

Many parameters in curve entry are adjusted by moving the mouse. Note, the sensitivity (resolution) of the mouse is greatest at the top of the screen and least at the bottom. If you move the mouse to indicate a point placement and the Shift Key is depressed, SMART prompts you for exact coordinate values in the Textport. You can also enter in the points approximately and then depress the Shift Key and click on the mouse on each point in turn to move it to an exact position. The add and edit options are explained in the following paragraphs.

• cubic

The **cubic** input option is the default for Bezier curves. Input consists of four points, the middle two of which are slope control points. Points are selected with the LMB or the MMB. If you are beginning a new curve, the LMB and the MMB yield C1 and C0 continuity, respectively, with the previous curve at the new end point. The last slope control point entered can be adjusted with the RMB.

• arc

When the **arc** option is selected, a circular arc is created. The arc by default has a subtended angle of 90°. This angle can be adjusted with the RMB and the movement of the mouse. As the angle of the arc changes, it is displayed in the Information Display Area. The center of the arc is adjusted with the MMB, and its location is displayed in the Information Display Area.

Repeated use of the LMB creates new arcs with the same center, radius, and angle values as their predecessors having C1 continuity (i.e., each arc starts where the last one left off). Thus, four clicks of the LMB create a circle when the angle is set to 90°.

● **line**

The **line** input option is straightforward. Simply click the LMB to specify an arbitrary end point. The RMB draws the line segment vertically and the MMB draws a horizontal line segment.

● **conic**

When the **conic** option is selected, the conic is displayed, with its starting point fixed at the end of the current cross section. The LMB adjusts the second end point, the MMB adjusts the slope control point of the curve, and the RMB adjusts the relative height parameter (RHP in fig. 5.2). The RHP value is displayed in the Information Display Area.
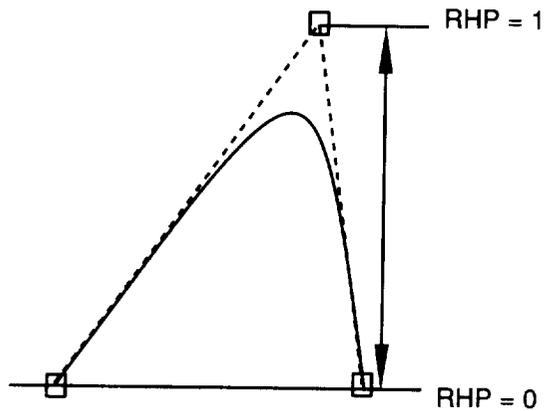
Figure 5.2. Relative height parameter.

The RHP value controls the classification of the conic section. The following table shows the type of conic that is created for each range of RHP value:

RHP	Type of conic	Precision
0	Line	Exact
>0 and <0.5	Elliptical arc	Approximate
0.5	Parabolic arc	Exact
>0.5 and <1.0	Hyperbolic arc	Approximate

• new CS

The **new CS** option allows you to start a new cross section or add a curve to an existing one, which you select by clicking the mouse on any control point on the desired cross section. The new curve is entered with the current option.

Editing Bezier cross sections

This section describes the options you can use to edit the Bezier cross sections. These options include **move**, **delete**, **close**, **key in**, **slope**, **split**, **fillet**, and **flatten**.

• move

The **move** option allows you to move points that define the curves. The LMB is used to move selected points. When you move an end point, the LMB breaks a curve where it joins the next, and the MMB ensures C0 continuity between adjacent curves. When you adjust a slope control point, the MMB also adjusts the control point of the next curve to maintain C1 continuity. Selecting a point with the Shift Key depressed allows you to move that point to an exact value, which you enter in the Textport.

• delete

With the **delete** option, you can delete one curve by clicking the LMB on a slope control point of the curve that you want. If you use the MMB, you are prompted to delete all selected cross sections.

• close

After choosing the **close** option, place the cursor over any control point, and SMART displays the new curve segment that results when you click the LMB. The MMB moves the current end point of the cross section so that it is coincident with the starting point and thus forces C1 continuity. If the slope of the end points of either cross section is changed, then you must close the cross section again to regain C1 continuity at the closure point.

• key in

The **key in** option allows you to enter exact points without relying on the mouse. First, enter approximate points with the mouse. To make points exact, choose **key in** and enter the coordinates of the point (either both on one line or one per prompt). Next, click the LMB on the point that you want to move to this location.

If you enter -999 for either the X- or the Y-coordinate, the current value of that coordinate is retained. This feature allows you to align a group of points along a horizontal or vertical line. For example, if you enter $(-999, 12)$ at the coordinate prompts and you click on two points $(4, 4)$ and $(8, 8)$, both points keep their original X-values and both Y-values change to 12. Thus, the resulting points are $(4, 12)$ and $(8, 12)$.

• slope

When generating a surface from your cross sections, you can use the **slope** option to control the line generated through corresponding points of consecutive cross sections. By clicking the LMB on an end point of a curve, you can change the longitudinal slope at the end point. With each click of the LMB, the slope cycles between continuous, discontinuous, right continuous, and left continuous, as displayed in the Textport. Any change made to a point is immediately reflected in the generated surface in the left View Window.

• reverse

With the **reverse** option selected, you can click the LMB on the slope control point of any curve of any cross section, and the parameterization is reversed.

• break/unbreak

With the **break/unbreak** command selected, clicking the LMB on any slope control point marks the curve as the end of a cross section. The MMB unbreaks the cross section. You can take advantage of this feature to insert a new curve between two existing ones.

• split

When the **split** option is selected and the cursor is placed on a slope control point of a curve, a small square box appears on the curve. This box is the point where the curve is split if the LMB is pressed. You can press the MMB and move the mouse to adjust the position of the split location.

• fillet

The **fillet** option creates a third curve in between two consecutive curves. By placing the cursor over any slope control point, you can preview the created curve. The LMB creates the fillet, and the MMB adjusts the fillet size.

• convert
(Bezier to Cartesian)

With the **convert** option, you can create Cartesian cross sections from Bezier cross sections. Select **convert** from the **Edit** menu and enter the number of points (the resolution) that you want in the Cartesian cross section. After the selected cross sections are converted, you can select **Cartesian CS** to manipulate them. All points in the Bezier cross sections that were discontinuous are present in the Cartesian counterparts, and they are marked as segment end points. (See fig. 5.3.)

29

Discontinuous end point
is retained

Continuous end point
is not retained
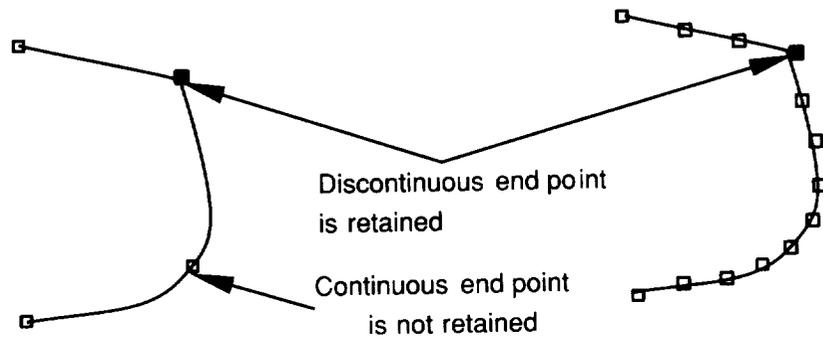
Figure 5.3. Converting Bezier to Cartesian curves.

• flatten

The **flatten** option linearizes Bezier curves. You can flatten one curve by clicking the LMB on a slope control point of the curve that you want. If you use the MMB, you are prompted to flatten all selected cross sections.

• plot normals

The **plot normals** option allows you to display the geometric smoothness of your Bezier cross section. When **plot normals** is toggled on, SMART prompts you for the number of normals desired. Your cross section is then redisplayed with the normal vectors and curvature values labeled on evenly spaced points along the cross section. If only one cross section is selected, a plot showing curvature versus arc length is drawn in the left View Window; this plot is updated as you edit the curve.

• Extract S and Extract T

You can use the **Extract S** and **Extract T** slider bars to extract cross sections from a surface. These bars are located just beneath the cross-section selection slider bars. Clicking the mouse on either of these bars creates a new cross section, which is coincident to either the S- or T-isoparametric lines of the surface. The isoparametric line corresponds to the value on the slider bar when the mouse button is released.

Cartesian CS

This section explains each of the **Cartesian CS** options.

• add/move

Since the Cartesian cross section is an ordered set of points, there is only one input option in SMART. To add a Cartesian cross section, select **add/move** from the **Edit** menu. Then click the MMB to start a cross section. SMART then prompts you for the OOSA coordinate. When in input mode, either the LMB or the MMB creates points. All points in this mode are white, except the end point of each cross section, which is blue.

In addition to being used to add Cartesian cross sections, the **add/move** option is used to edit cross sections. The LMB is used to move existing points. Selecting a point with the Shift Key depressed allows you to move that point to an exact value, which you enter in the Textport. The MMB adds a point after the highlighted point; this point follows the cursor until the LMB or the MMB is pressed to insert the point. Clicking the RMB or moving the cursor outside of the View Window ends the input of the cross section.

- **delete**

With the **delete** option, you can delete one point by clicking the LMB. If you use the MMB, you are prompted to delete all selected cross sections.

- **close**

The **close** option lets you create closed cross sections. Selecting any point on a cross section with the LMB moves the last point to the first. The MMB adds a point to the end of the cross section coincident to the first point.

- **key in**

The Cartesian **key in** option works exactly the same way as the Bezier **key in** option.

- **redistribute**

The **redistribute** option allows you to repeatedly select portions of existing Cartesian cross sections, calculate a C1 spline through the points, and replace them with n evenly spaced points, where you specify n through the Textport. To select portions, you click the LMB on the first and last point in the section. If you want discontinuities, redistribute points in sections. The MMB lets you change the number of points used in the redistribution.

- **reverse**

With the **reverse** option selected, you can click the LMB on any point on any cross section and the order of points is reversed.

- **break/unbreak**

When the **break/unbreak** option is selected, clicking the LMB on a point marks it as an end of a cross section. This operation splits the cross section in two. The **unbreak** option undoes what the **break** option did. You can take advantage of this option to add a new cross section between two existing ones.

- **segment**

The **segment** option lets you change the characteristics of a given point to prepare the cross section for subsequent conversion to a Bezier cross section. The LMB causes the point to be drawn green, which indicates that the point is to maintain slope continuity between consecutive Bezier curves. The MMB makes the end point red, which indicates that this point should be considered discontinuous with the adjoining curve. The RMB removes segment markers.

31

• convert
(Cartesian to Bezier)

Before converting selected Cartesian cross sections to Bezier cross sections, you can use the **segment** option to divide your Cartesian cross sections into segments. Using this option tells SMART how many curves you want to use to describe each of your cross sections. You must have at least three points between segments. The more segments that you have, the more accurate the conversion will be. When you are through adding segments, choose **convert** from the menu and look at the resulting Bezier cross sections. If you are not satisfied, adjust your segmentation points, delete the Bezier cross sections generated previously, and try again, or you can edit the Bezier cross sections with the **move** option. (See figs. 5.3 and 5.4.)

Cartesian to Bezier
with segmentation
indicated

Cartesian to Bezier
without segmentation
indicated
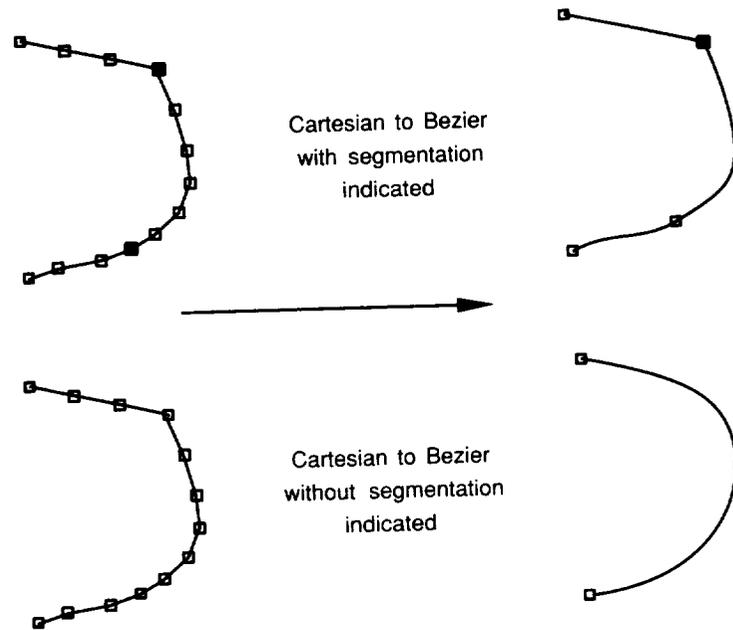
Figure 5.4. Segments versus curves.

• relink

The **relink** option is used in conjunction with the **break/unbreak** option to reorder points in a cross section. To use the **relink** option, click the mouse on the first point of each segment in the order that you want them joined. In between clicks, a point extending from the end point of the cross section that was previously selected moves with the cursor as you click on the next cross section in the sequence. If you click the mouse on the end point of a cross section, the points in that cross section are reversed automatically. The relink can be aborted at any time by moving the cursor out of the View Window. The description is much more complicated than the task; try out the option to understand how it works.

• clean

The **clean** option removes all duplicate points and resets segment flags.

NOTE: *If all the Cartesian cross sections in an object have the same number of points, SMART draws connecting lines between corresponding points on each cross section.*

Common Options

This section describes the options that work the same for **Bezier CS** and **Cartesian CS** modes.

• transformation slider bars

You can use the nine slider bars on the right side of the Menu Display Area to rotate, scale, and translate selected cross sections. The right View Window's rotation center is used as the reference point for rotations and scaling.

• template plot

The **template plot** option is located in the lower left corner of the Menu Display Area. When the **template plot** is on, SMART keeps the last-selected Bezier cross sections visible when you enter Cartesian mode and the last-selected Cartesian cross sections visible when you enter Bezier mode. With these curves visible, you can see how accurately the conversion algorithm converted the cross sections. Thus, you can edit them to more closely match the cross sections from which they were created.

• copy

After selecting **copy** from the **Edit** menu, SMART prompts you for an offset. After you enter the offset value, the selected cross sections are duplicated and offset along the OOSA.

• sort

Sorting is useful when you add new cross sections between previously created cross sections, and this addition causes them to become out of order. Sorting the cross sections places them in order along the OOSA. Select the cross sections that you want to sort (normally **all CS**) and select the **sort** option. The Textport tells you when the sort is finished.

Surface Generation

After selecting **surfaces** from the **Mode** menu, you are presented with options for generating surfaces from Bezier cross sections. These options are described in the following paragraphs.

33

• revolve

The **revolve** option for surface generation takes the cross section you have drawn and rotates it around one of the three coordinate axes. After choosing **revolve** from the menu, SMART shows you the surface of revolution. (The X-axis is the default.) The rotation axis, rotated cross section, and rotation angle can be adjusted by the slider bars and menus.

• skin

The **skin** option for surface generation creates a series of patches from stacked Bezier cross sections. The degree of longitudinal slope can be controlled by the auxiliary menu that is displayed while skinning. The **C0** and **C1** options are available. If the **C1** option is used, you can use the **slope** option in **CS Bezier** mode to change the longitudinal slope of each point on any of the cross sections; the surface is then redrawn to reflect your change. The longitudinal slope is only meaningful for intermediate cross sections not for end cross sections.

Since you often use the **skin** option when creating vehicle fuselages, SMART provides an easy method for creating a blunt nose on your vehicle. When you choose the **blunt nose** option, SMART prompts you for the location of the nose tip. SMART fits the nose based on the first cross section. You should plan your drawing so that the nose point is easily referenced, for example, (0,0,0). Another way to create a nose is to duplicate the cross section nearest the nose and scale it down. Use as many cross sections as needed to create the degree of bluntness needed for your configuration. The endmost cross section can be scaled down to a point.

• extrude

In SMART, extrusion is the process of taking one cross section and moving it through space along the path described by another. To extrude a shape along a cross section, first draw the extrusion path as you would any cross section. Make sure that all the points lie in the X-Y-plane. Next, draw the shape you want to extrude with a station larger than that of the extrusion path because SMART performs the extrusion based on the sorted order of the curves. If you accidently create the curves in the wrong order, you need to move the extrusion shape farther from the origin by using a translation slider bar. Next, select both of the cross sections and then sort them. Last, select the **extrude** option and SMART draws the extrusion. Note, slope-discontinuous points in the path cross section do not cause a discontinuous extrusion.

• delete

The **delete** option deletes all patches in the object. This option is the same as the **delete** option in **PATCH TOOLS** except that all patches are deleted rather than just those selected.

• off

The **off** option turns off SMART's automatic refitting of the surface when any cross section is changed in Bezier mode. This option is useful when you need to make many changes and you do not want SMART to recalculate the surface each time. The refitting mode is automatically turned on again when you explicitly regenerate the surface.

• from 4

The **from 4** option is a surfacing method that creates fewer patches than skinning but with less control over continuity between the patches. SMART starts with the first cross section and takes the cross sections in groups of four; this step creates groups equal to one plus the number of cross sections divided by three. For each group, SMART determines which has the fewest number of curves and calls this number cmin. The cmin patches are then created for that group. Since only the first cmin curves of each cross section are used, any extra data (curves) are ignored, including any trailing cross sections that do not permit a grouping of four.

Chapter 6
IMAGES Mode

This chapter explains the various options in IMAGES Mode that allow you to change imaging parameters. These imaging parameters affect the manner in which your model is displayed, such as its shading, visibility, and precision of drawing.

When you select **IMAGES** from the Mode Menu, the **Node** menu listing the node types is displayed in the Menu Display Area. Click the mouse on one of the four types to display the Tree. If no objects in the Tree match the chosen type, SMART beeps and redisplays the **Node** menu. All the available nodes of the type selected are highlighted for you to select from. Note, the Node menu appears in the submenus of all the node types. Any parameter changed on a group node affects all nodes of the selected type within that group.

Geometry Presentation Parameters

This section describes the submenus available when editing a geometry object. After selecting a geometry node, the various presentation menus appear in the Menu Display Area. (See fig. 6.1.) Now you are ready to edit the presentation parameters of the selected node. A black line separates the menus into two groups. On the left are the menus that affect the wireframe image, and on the right are the menus that affect the shaded image.

Node	Element		Wire Vis	Precision		Shade	Shade Vis	Color		
geometry	point	black	invisible	S lines		0	invisible		▲ Red = 20.0 ▼	
text	curve	red	visible	T lines		1	transparent		▲ Green = 20.0 ▼	
light	patch	yellow		pts/line		2	opaque		▲ Blue = 20.0 ▼	
dimension	quad	green		recalc auto		3			▲ Ambience = 20 ▼	
	all	blue				4			▲ Diffusion = 20. ▼	
		cyan							▲ Specularity = 20 ▼	
		magenta								
		white								
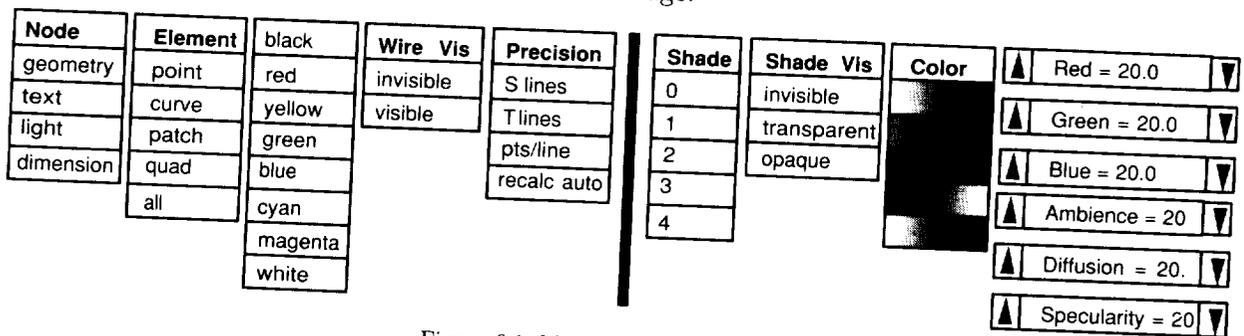
Figure 6.1. Menus for IMAGE Mode.

Element

The **Element** menu lists the various geometry types of an object: **point**, **curve**, **patch**, **quad**, and **all**. It is used in combination with the **Color** and **Wire Vis** menus. The item currently highlighted shows what element of the object's presentation is affected. The item highlighted on this menu is referred to as the current element.

Color

The **Color** menu shows the colors available for the various lines composing the wireframe representation of the object. By default, each element listed in the **Element** menu has a different color to aid interpretation of the many lines that compose a drawing. The color of the current element type (in the current object only) can be changed by selecting the desired color name. For example, to change the patches to the color white, select **patches** from the **Element** menu, then select **white** from the **Color** menu. If the patches are marked visible, you can see them change color on the screen. Even if the lines are not currently visible, the color is still changed.

Wire Vis

The **Wire Vis** menu turns the current element on (**visible**) or off (**invisible**). For example, if you shaded an object and wanted the quad lines to be invisible, choose **quads** from the **Element** menu, then select **invisible**. The quads should disappear.

Precision

The **Precision** menu changes the number of points or lines used to approximate the patch or curve geometry when displayed as a wireframe model. The **S-lines** and **T-lines** options change the number of lines in the S- and T-directions. (See appendix B.) The **pts./line** option changes the number of points used in each direction of the patch; a higher number generates smoother curves for the wireframe object, but it also increases display time on complex objects. All these options prompt you for a new value in the Textport.

If a model is complex, you may want to delay recalculation of S- and T-lines and the points per line until you have made all the precision changes. To delay recalculation, set the recalculation mode to **recalc man** by selecting the **recalc auto** option. When **recalc man** is on, none of the precision calculations occur until you select the **recalc man** option. If no precision values have been reset, selecting the **recalc man** option turns the recalculation mode back to automatic.

Shade

With the **Shade** menu, you can select the number of quads per patch to control the degree of resolution in the shading algorithm. A higher number indicates a higher resolution. Thus, use numbers **0** through **3** to create a quick display and use number **4** only to generate a final picture. A shading number of **1** or **2** is enough for most uses. Any time you change the shading precision, SMART recalculates the shaded model. Note, SMART displays the name of each object in the Textport as it is shaded. If one patch of an object does not appear shaded, then its normal surface may have been reversed. See the discussion on the **Parameterize** menu in the chapter on PATCH TOOLS Mode for details on correcting this situation.

Shaded Vis

The **Shaded Vis** menu controls the appearance of the shaded object. You can make the shaded portion of the model either **invisible**, **transparent**, or **opaque**.

Color

Each color map defines colors that SMART uses to shade an object. When you select one of the 12 predefined color maps, SMART reshades the object in that color.

Adjusting Color Maps

You can adjust a color map with the six slider bars in the Menu Display Area. The **Red**, **Green**, and **Blue** slider bars adjust the color. The shading can also be adjusted. The **Ambience** slider bar adjusts the ambient lighting level. The higher the value, the brighter the ambient light. The ambient light level refers to the level of light that is not related to a specific source of light.

The **Diffusion** slider bar adjusts the amount of light that the object itself reflects. The **Specularity** slider bar affects the shininess of the object. When combined with **Diffusion**, some interesting effects are generated. Note, the color maps are saved with the model when you write your file, so they are available the next time you read in the file and shade your model.

Light Parameters

When you select the **light source** option in the **Node** menu, SMART displays the **Light Parameters** menu; this menu allows you to edit the light nodes in the Tree. The **light source** can be made visible or invisible, which determines whether the symbol for the light source is drawn. The **light source** can also be turned on (**white**) or off (**black**) to add or remove its effect on the shaded image.

Dimension Presentation Parameters

You can change only the color and the visibility of a dimension object.

Text Presentation Parameters

When you select the **text** option on the **Node** menu, SMART displays menus for editing the color, visibility, font, and line width of the text. You can also edit your text with the replace text option.

39

Chapter 7
PACKAGING Mode

This chapter describes the PACKAGING Mode, which allows you to rotate, translate, and scale objects and groups of objects. SMART uses a matrix to keep track of how an object is transferred. When an object is created, the transformation matrix is set to the identity matrix. When you make transformations through the PACKAGING Mode, the transformation matrix is changed accordingly.

When you select **PACKAGING** from the Mode Menu, SMART shows you the Tree. Select either the group or leaf node that you want to transform. All visible items are then shown in the two View Windows with the transformation reference point (TRP) displayed as three vectors marked **X**, **Y**, and **Z**. The name of the group or leaf node selected is displayed in the Function Name Area throughout the editing session, except when a slider bar is activated, and the menus shown in figure 7.1 appear in the Menu Display Area.

Autowindow		Edit Options		Edit Mode		▲	X Rotation = 20.0	▼
on		select node		rotate		▲	Y Rotation = 20.0	▼
off		undo		translate				
		identity		scale		▲	Z Rotation = 20.0	▼
		reset tree						
		CG off						
		move						
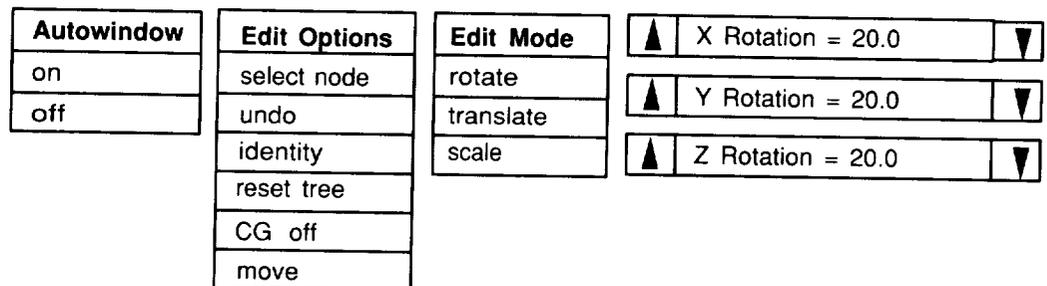
Figure 7.1. Menus for PACKAGING Mode.

Edit Options Menu

This section describes the following options on the **Edit Options** menu: **select node**, **undo**, **identity**, **reset tree**, **CG off**, and **move**.

● select node

The **select node** option lets you select a new node to edit. When you use this option, SMART highlights the node in white and saves its current transformation matrix.

● undo

The **undo** option resets the object's transformation matrix to the saved transformation matrix. Thus, any transformations that you have indicated since last selecting that node are undone.

• Identity

The **Identity** option resets the transformation matrix of the object to the identity matrix. Thus, any transformations that you have ever indicated are undone.

• reset tree

The **reset tree** option traverses the entire tree, transforms the geometry data, and resets each node's transformation matrix to the identity matrix. This option does not change the appearance of the object (unless you had reflections), but it causes all the slider bar values to be set to zero.

• CG off

The **CG off** or **CG on** option is a switch menu option, whereas the other options are action menu options. When this option is selected, you are prompted to select a node whose center of gravity is to be displayed. This node can be different from the node you are editing. SMART then performs the required geometric property calculations for the untransformed objects. These intermediate results are used to calculate the CG of the transformed objects. The CG is displayed as a small symbol at the CG location in the View Window and the exact CG coordinates are displayed in the Information Display Area. As objects are transformed, the CG is updated. If you nonuniformly scale an object, the CG display is turned off. Nonuniform scaling requires new intermediate values that will be calculated when you turn the CG display back on.

• move

When you select the **move** option, the Textport prompts you for X-, Y-, and Z-translation coordinates. After you enter the values, SMART translates the object to these coordinates.

Autowindow

The **Autowindow** option automatically adjusts the view so that all visible objects fit into the View Windows. This option is helpful when you are performing large transformations.

Edit Mode Menu

The **Edit Mode** lets you select the type of transformation to perform on the object when you use the slider bars. **Rotate**, **Translate**, **Scale**, and **Reflect** options are available.

Slider Bars

This section describes how you can use the slider bars and View Windows to transform your object. On the right side of the Menu Display Area are three slider bars, one for each of the X-, Y-, and Z-axes. These slider bars allow you to transform the selected objects in the current edit mode. The sensitivity of the slider bars depends on the degree of magnification used in the right View Window (i.e., the closer you are to the object, the more sensitive the slider bar).

• rotation slider bars

Three slider bars are present when the **rotate** mode is selected; these bars allow you to rotate your objects about any of the three axes.

• scale slider bars

Four slider bars are available when the **scale** mode is selected, one for each axis direction and a fourth to scale uniformly in all directions.

• translation slider bars

The translation slider bars appear when you enter PACKAGE Mode. The first set of slider bars (labeled **X, Y, or Z Local Trans**) controls motion in the local axis system. Since motion in the local axis system is relative to the TRP, the local slider bars are reset to zero after each translation.

The slider bars labeled **X, Y, or Z World** control motion in the coordinate system that you see defined by the grid. If you hold down the space bar before clicking the mouse on the world slider bars, the values are set temporarily to zero, which allows you to make relative translations. When you release the mouse button with the space bar depressed, the world slider bars are reset to zero and are ready for another relative translation. After the space bar is lifted, the world coordinates of the object again reappear.

• dragging objects

As an alternate way to translate an object, you can position the cursor in the View Window and press and hold the LMB while moving the mouse to indicate the desired position. Release the LMB when you are done. Thus, you can move in two directions simultaneously. Note, you must select **PLNR** from the View Options Menu after entering PACKAGING Mode to use the drag object feature.

• reflection slider bars

SMART provides the capability to reflect a node about any of the TRP axes. You can reflect an object about more than one axis. When the **reflect** mode is selected, you can click on each of the three slider bars to toggle between **True** and **False**. This value indicates whether or not an image, which is reflected about that axis, is generated. Note, new geometric data are not generated in this reflection—only a reflected image. Reflections in upper nodes duplicate reflections in nodes below them.

Transforming Groups

You can easily apply a single transformation to several objects by placing all the desired objects (leaf nodes) into a single group node and then performing the desired transformation operations on the group node. The group-level transformation is added to the objects' existing transformations. Because of this hierarchical approach to building transformations, you can conveniently change an object's TRP for rotation and scaling operations. To change the reference point, place

43

an object under a group, translate the object to the negative values of the desired reference point, and then perform rotations and scalings on the group node. You can also change a node's TRP with the **reset** Tree option in this mode and the **combine** option in TREE Mode.

NOTE: *Changing an object's location in the Tree with the* **move** *option in TREE Mode changes the transformations that are applied to it.*

Special Primitives

When a dimension object is translated, leader lines are drawn that connect the ends of the dimension lines to the original measured points. Once created, dimension objects cannot be made to measure a distance different from the one it originally measured. However, as you scale an object, the dimension value changes to reflect the new size. The **rotate** and **reflect** modes are not active for dimension objects. If you want to transform an object along with a dimension, put both in a group and transform the group.

Recall from PRIMITIVES Mode that rotations are performed on a text object, the text rotates with the object when you use the View Options. Otherwise, the text ignores view rotations and remains facing you. Translation is the only transformation that is applicable to light objects, although SMART does not complain if you try to perform other transformations.

Chapter 8

PROPERTIES Mode

This chapter describes how to generate on-screen reports that list the physical properties of your model. SMART can calculate the following physical properties: surface area, area centroids, volume, volumetric centroids, mass, and mass centroids. SMART can also calculate moments, products of inertia, and projected areas. Area, volume, and length properties are calculated by integrating an object's patch and curve geometry. Point objects are composed of Cartesian cross sections. Except for polygonal areas, SMART cannot calculate properties for point objects. Mass properties are derived from the area, volume, and length calculations and corresponding unit densities. If the chosen object is a group, then only mass properties are displayed.

When you select **PROPERTIES** from the Mode Menu, three menus appear in the Menu Display Area. (See fig. 8.1.) The **Density** menu lets you assign density to an object, the **Physical Properties** menu lets you generate on-screen reports of an object's physical properties, and the **Reference Point** menu lets you set the reference point about which the moments of inertia are calculated.

Reference Point	Physical Properties	Density
arbitrary	properties	mass/unit length
nodes's CG	projected areas	mass/unit area
default (0,0,0)	polygon area	mass/unit volume
		inquiry

Figure 8.1. Menus for PROPERTIES Mode.

Density Menu

This section describes how to use the **Density** menu options to assign unit density values to objects. The densities assigned to a group node are assigned to the appropriate nodes in that group. To change the density value of an object, select an option from the **Density** menu. SMART then displays the Tree and outlines in red all nodes that are applicable to that option. Click on a red node, and SMART displays the following prompt: "Enter mass/unit value." At the prompt, type in a value and press <Enter>. You can then assign a density to another node or select another option from the menu. The Density menu has the following four options:

45

- **mass/unit length**   Only curve objects have mass per unit length; the default value is 1. The option lets you model pipes and other slender objects as curves.

- **mass/unit area**   Patch objects have an assigned density per unit area; the default value is 1. A tank, for example, can be assigned an area density to represent the structural weight.

- **mass/unit volume**   Patch objects have a density per unit volume as well as a density per unit area; the default value is 0. An object representing a fuel tank, for example, can be assigned a volume density to represent the mass of the fuel it contains.

- **inquiry**   The **inquiry** option lets you check the densities that are currently assigned to an object. A small window in the lower left corner of the screen lists all the unit densities that are assigned to that node.

Physical Properties Menu

This section describes the three options on the Physical Properties menu.

- **properties**   The **properties** option generates an on-screen report that details the physical properties of an object. Figure 8.2 shows the report generated for a sphere with its center at (0,0,0) and a radius of 1. The area and volume densities are also both set to 1.

To generate an on-screen report, first select **properties** from the **Physical Properties** menu, then click on any red node to select the object that you want. If the node's properties have not been calculated, the following message appears in the Textport: "Please wait while properties are calculated." Each time SMART completes the calculations for a patch or curve, SMART prints a bar (|) in the Textport. The properties are displayed in the left View Window and a small picture of the object is displayed in the lower left corner of the screen. A symbol marks the position of the CG. When you are finished viewing the properties report, you can select another node from the Tree or another option from the displayed menus.

NOTE: *The correctness of the physical properties cannot be guaranteed when you have scaled a group node nonuniformly; however, nonuniform scaling of leaf nodes is not a problem. Inaccurate results can also occur when the surface of the object is "open" or when all surface normals are not oriented in an outward direction from the object's center.*

```
┌─────────────────────────────────────────────┐
│ PHYSICAL PROPERTIES OF SPHERE                │
├─────────────────────────────────────────────┤
│ AREA PROPERTIES                              │
│ total        12.5686                         │
│ unit wgt     1.0000                          │
│ total wgt    12.5686                         │
│ centroid     (0.0000,   0.0000,0.0000)       │
│ Ixx    8.3830                                │
│ Iyy    8.3830                                │
│ Izz    8.3830                                │
│ Ixy    0.0000                                │
│ Ixz    0.0000                                │
│ Iyz    0.0000                                │
├─────────────────────────────────────────────┤
│ VOLUME PROPERTIES                            │
│ total        4.1902                          │
│ unit wgt     1.0000                          │
│ total wgt    4.1902                          │
│ centroid     (0.0000,   0.0000,0.0000)       │
│ Ixx    1.6778                                │
│ Iyy    1.6678                                │
│ Izz    1.6678                                │
│ Ixy    0.0000                                │
│ Ixz    0.0000                                │
│ Iyz    0.0000                                │
├─────────────────────────────────────────────┤
│ MASS PROPERTIES                              │
│ total        16.7587                         │
│ total wgt    16.7587                         │
│ centroid     (0.0000,   0.0000,0.0000)       │
│ Ixx    10.0608                               │
│ Iyy    10.0608                               │
│ Izz    10.0608                               │
│ Ixy    0.0000                                │
│ Ixz    0.0000                                │
│ Iyz    0.0000                                │
├─────────────────────────────────────────────┤
│ reference point is    (0.000, 0.000, 0.000)  │
└─────────────────────────────────────────────┘
```

Figure 8.2. Physical properties report for sphere.

• projected areas

You can use the **projected areas** option to calculate areas of projection to the X-Y, X-Z, and Y-Z coordinate planes for leaf nodes only. To generate an on-screen report of an object's projected areas, select **projected areas** from the **Physical Properties** menu, and click on any red node. The projected area is then displayed in the left View Window.

47

● polygon area

You can use the **polygon area** option on objects represented by a single planar Cartesian cross section to calculate the area bounded by the cross section. Select **polygon area** from the **Physical Properties** menu and click on any red node. The area is then displayed in the left View Window.

Reference Point Menu

The coordinate values of the reference point appear at the bottom of the mass properties report. The default reference point for calculating inertias is (0,0,0); however, you can move the reference point by selecting one of the following options from the **Reference Point** menu.

● arbitrary

To specify the X-, Y-, and Z-coordinates, select **arbitrary** from the **Reference Point** menu. At the prompt, type in the values of the desired reference point.

● node's CG

To move the reference point to the CG of an object, select **node's CG** from the **Reference Point** menu. Click on a red node in the Tree, and the reference point is set to be the CG of the selected node.

● default

To reset the reference point to (0,0,0), select the **default** option. Click on a red node in the Tree and the reference point of the selected node is set to (0,0,0).

Chapter 9
PATCH TOOLS Mode

This chapter describes the PATCH TOOLS Mode, which lets you manipulate individual patches in any geometry object. After selecting **PATCH TOOLS** from the Mode Menu, SMART displays the Tree and asks you to select the object whose patches you want to edit. The selected object is then displayed by itself. The menus shown in figure 9.1 appear in the Menu Display Area, and you are now ready to manipulate patches.

Select	Edit Patch	Edit Object	Display
node	split S	move patches	shrink 1
patches	split T	copy patches	shrink 2
points	split S & T	delete patches	shrink ?
	move points		normals
	join patches	**Parameterize**	
	fillet patches	switch S & T	
		reverse	

Figure 9.1. Menus for PATCH TOOLS Mode.

Select Menu

The **select menu** lets you choose objects, patches, or points for editing. Patches and points are selected for editing with the LMB and deselected with the MMB. In the right View Window, hold down the mouse button and drag the cursor over the items to be selected or deselected. When you release the mouse button, these items are added to or removed from the current patch or point set. The current patch set is highlighted in white. Points in the current point set are drawn as squares; unselected points are shown as dots. Points are selected or deselected only from the current patch set.

• node

The **node** option lets you select a new object from the Tree for patch editing.

• patches

The **patches** option lets you select and deselect patches from your current patch set.

49

- **points**   The **points** option lets you select and deselect points from your current points set.

Display   The **Display** menu has four options that control how the patches are drawn in the View Windows. The **shrink** options let you see patch divisions more clearly. Although the patches are drawn smaller, the size of the patches is not altered. The highlighted lines of the current patch set still appear full size.

- **shrink 1**   The **shrink 1** option shrinks the patches by about 10 percent.

- **shrink 2**   The **shrink 2** option shrinks the patches by about 30 percent.

- **shrink ?**   The **shrink ?** option lets you specify the shrink percentage.

- **no shrink**   The **no shrink** option redraws the full-sized patches.

- **normals**   The **normals** option toggles the display of the normal vector, the S- and T-directions, and parametric origin for all patches in the current patch set.

Parameterize   The **Parameterize** menu lets you modify the orientation of the parametric coordinates (S and T) for each patch.

- **switch S & T**   The **switch S & T** option switches the S- and T-directions for all patches in the current patch set. With each click of the mouse, the parametric origin of each patch moves to the next clockwise corner, and the S- and T-directions are switched. Thus, the surface normal direction remains the same.

- **reverse**   When you select the **reverse** option, the surface normal is reversed, the S- and T-directions are swapped, and the patch origin is maintained. When you reshade the patches, you should see the effect of using this option.

Edit Menu   The following options on the **Edit** menu let you divide, join, blend, and deform patches.

- **split S**   The **split S** option creates a curve that indicates where you want to split the patch. One end point is on each S-edge of the patch. This curve can be adjusted by moving the mouse; up-and-down motion adjusts one end point and left-to-right motion adjusts the other. When you click any mouse button, SMART displays the new patches.

● **split T**

The **split T** option works the same as the **split S** option except in the T-direction.

● **split S & T**

The **split S & T** option divides the patch into two or four pieces. After you select this option, two split lines are shown on each patch for you to adjust the split in each direction. As with the **split S** option, any click on the mouse button causes SMART to divide the patch along your chosen split lines and redisplay the four new patches. If the S-split line is made coincident with one of the patch edges, the patch is divided into two parts in the T-direction, and vice versa.

● **join patches**

The **join patches** option requires that you select only two patches and that they share a common edge. SMART then combines these two into one. It is the inverse operation of the **split** options.

● **fillet**

The **fillet** option creates a curved connection between two adjacent patches. To use the option, select two patches that share a common edge, then select **fillet** from the menu. SMART draws a patch that is responsive to the motion of the mouse and displays the fillet percentage in the Information Display Area. As you move the mouse from left to right, the fillet becomes more pronounced. When you are satisfied with the fillet, click any mouse button, and the two patches are replaced by three, including the fillet. If you plan to repeat the option on a number of similar patch pairs, remember to note the fillet percentage. If the fillet cannot be calculated, the message "UNSUCCESSFUL FILLET OPERATION" is displayed in the Textport.

● **move points**

The **move points** option lets you translate a patch. After selecting the desired points, click on the **move points** option. The LMB moves the points in the X-direction, the MMB in the Y-direction, and the RMB in the Z-direction. To exit this mode, move the cursor out of the View Window.

Edit Objects

The **Edit Objects** menu lets you delete patches as well as move and copy patches from one leaf node to another.

● **move**

Moving patches means to take a set of patches from one leaf node and place them into another. To move patches, first define your current patch set. Next, select the **move** option. SMART then presents the Tree, and you must select the destination node for the current patch set. After selecting the node, you are returned to PATCH TOOLS Mode and the current patch set is no longer visible (it is in fact, deleted).

When you view the destination node in PATCH TOOLS, IMAGES, or PACKAGING, the new patches have been added.

NOTE: *The transformations currently in effect for both the source node and the destination node of the moved patches are not applied to the patches; thus, in some instances the patches do not appear transformed as expected.*

● **copy**

The **copy** option works the same as the **move** option, except the original patch set is not deleted.

● **delete**

The **delete** option removes the current patch set from the node.

Chapter 10
STRUCTURES Mode

This chapter describes the options available in STRUCTURES Mode. This mode simplifies the construction of wing geometry that is suitable for finite-element model mesh generation. You use the mouse and menus to build a structural model by following a four-step process: (1) define the wing box, (2) create a rib pattern, (3) create a spar pattern, and (4) select Done. When you enter STRUCTURES Mode, you are prompted to select a wing from the Tree. You are restricted to single delta wings generated in PRIMITIVES Mode. The planform of your wing, which is similar to the one shown in figure 10.1, appears in the right View Window.
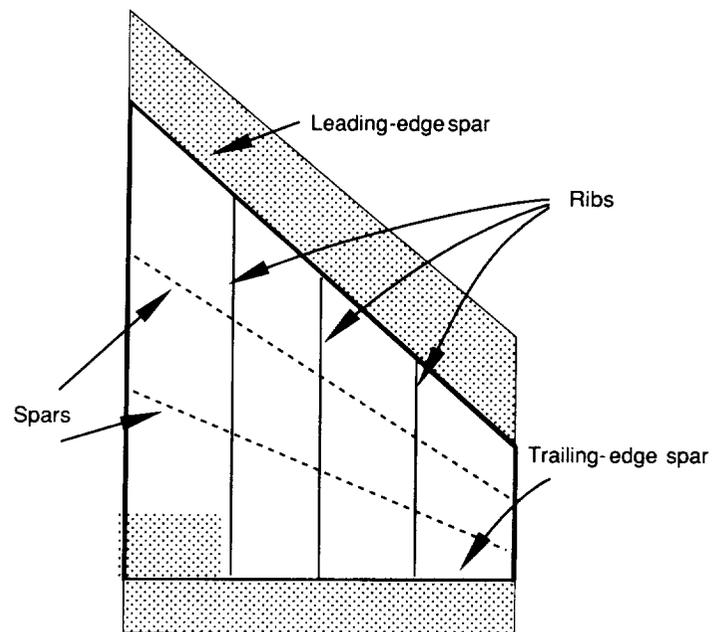

Figure 10.1. Wing planform and internal structure.

In figure 10.1, the planform is the outline of the *X-Y* projected area of the wing. The dashed lines that run spanwise across the planform are called spars, and the solid lines that run chordwise across the

53

planform are called ribs. The leading-edge spar is the major load-bearing structure located behind the wing's leading edge, and the trailing-edge spar is the major load-bearing structure located in front of the wing's trailing edge. The root rib is the inner-most rib, and the tip rib is the outer-most rib. The wing box, which is shown by the thick lines, is defined by the trapezoid bounded by the leading-edge spar, the trailing-edge spar, the root rib, and the tip rib.

Figure 10.2 shows the menus for STRUCTURES Mode. The **Modify** menu is used to select the type of structure to edit. The **Which Edge** menu is used to indicate which spar to edit when defining the wing box. The **Edit Mode** menu is used to indicate whether to create, move, or delete the structure. The **Pattern** menu is used to select predefined patterns for creating ribs or spars. These pop-up switch menus work differently from other SMART switch menus. The menu title is shown on the left, and the current mode is shown on the right. To select the options, hold the LMB down while pointing at the menu, pull down to the desired option, and release the LMB. Some **Pattern** menu options display up and down arrows to the right of the menu for adjusting numeric values. If you are creating ribs or spars, a slider bar is presented below the menus and is used to specify how many ribs or spars to create. The Done button is located to the left of the pop-up menus.
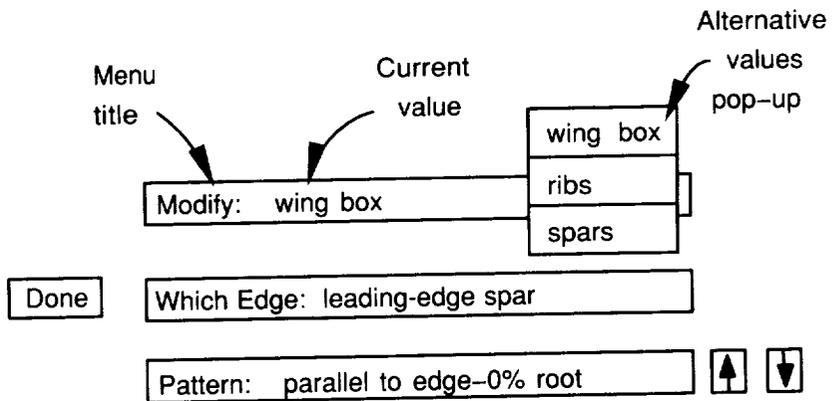
Figure 10.2. Menus for STRUCTURES Mode.

Modify: Wing Box

When you set the **Modify** menu to **wing box**, you can move the leading- and trailing-edge spars. The root rib and tip rib cannot be moved. By default, the leading-edge spar is coincident with the planform leading edge, and the trailing-edge spar is coincident with the planform trailing edge. To change the position of the leading-edge spar, set the **Which Edge** menu to **leading-edge spar**. To change the position of the trailing-edge spar, set the **Which Edge** menu to **trailing-edge spar**.

When you set the **Pattern** menu, you can move either spar so that it remains parallel to or is skewed to a planform edge.

You can use the arrows located to the right of the **Pattern** menu to specify how far back from the leading edge or how far forward of the trailing edge to place the spar. The specified value represents a percentage of the chord length. The leading edge is 0 percent and the trailing edge is 100 percent. When **Pattern** is set to **skew**, you are provided two Pattern menus and two sets of arrows. The top Pattern menu specifies a percentage length of the root chord, and the bottom menu specifies a percentage length for the tip chord.

An alternate method is to use the mouse to place a parallel spar. Move the cursor into the right View Window, and SMART highlights the root chord. Position the cursor on the root chord, and click the LMB to place the spar at the indicated position. You can also use the mouse to indicate where you want a skewed spar to intersect the root and tip ribs. Move the cursor into the right View Window. SMART now highlights both the root chord and the tip chord. Position the cursor on the root chord where you want one end of the spar to be located. While keeping the LMB pressed, move the cursor to the spar's tip chord location and release the LMB.

NOTE: *If you try to change the wing box after ribs or spars have been entered, SMART asks you whether you want to delete the existing ribs and spars. If you answer no, you cannot edit the wing box.*

Modify: Ribs

This section describes how you can design rib patterns inside the wing box. To access these options, set the **Modify** menu to **ribs** and the **Edit Mode** menu to **create**. Then, set the **Pattern** menu to one of the following input options:

● **Pattern: parallel root rib**

The **parallel root rib** option creates ribs parallel to the root rib. The number of ribs is input with the slider bar, and the ribs are equally spaced along the span.

● **Pattern: perpendicular chord**

The **perpendicular chord** option creates ribs that are perpendicular to the line that intersects the root and tip ribs at a specified percentage of the chord length. To change the percentage, click on the arrows to the right of the menu. As with the preceding option, you can use the slider bar to create a number of equally spaced ribs.

Modify: Spars

This section describes how you can design spar patterns inside the wing box. To access these options, set the **Modify** menu to spars and the **Edit Mode** menu to create. Then set the **Pattern** menu to one of the following input options.

• Pattern: equal spacing

The **equal spacing** option creates spars that intersect the root and tip ribs at an equal percentage of the chord length. You can use the slider bar to enter the number of spars.

• Pattern: parallel chord

The **parallel chord** option creates spars that are parallel to the line that intersects the root and tip ribs at the specified percentage of the chord length.

• Pattern: perpendicular ribs

The **perpendicular ribs** option creates spars that are perpendicular to existing ribs and are located where the rib intersects the leading-edge spar. To create ribs one at a time, click the LMB on the end point of any rib. SMART enters a spar that is perpendicular to the rib. At the top of the View Window is an **ALL** button. Clicking **ALL** creates a perpendicular spar for each of the ribs.

Modify: Ribs or Spars

The following **Edit Mode** options work identically for ribs and spars.

• Edit Mode: create

When the **Pattern** menu is set to **free hand**, you can define ribs or spars one at a time by pointing to the end points on the wing box. Move the cursor into the right View Window. Position the cursor on the wing box where you want one end of the rib to be located. Press the LMB, move the cursor to another edge of the wing box, and release the LMB. This process is identical to the one that specifies skewed spars for the wing box.

The rib patterns: **parallel root** and **perpendicular chord** and the spar patterns **equal spacing** and **parallel chord** generate equally spaced lines when the slider bar is used. With these options, you can also enter ribs or spars one at a time by pointing to the desired position. In the right View Window, position the cursor on the wing box where you want the rib or spar to be located and click the LMB. The ribs or spars are no longer equally spaced.

NOTE: *Adjusting the slider after adding ribs or spars in this way causes them to revert to equal spacing.*

● **Edit Mode: move**

When you set the **Edit Mode** menu to **move** and move the cursor into the right View Window, SMART draws the structure type that you are editing in yellow and the other type in white. (The type is set on the **Modify** menu.) If you are moving spars, for example, the spars are displayed in yellow and the ribs are displayed in white. Movable end points are marked with small squares. Position the cursor on the end point that you want to move. Press the LMB, drag the end point to a new location on the wing box, and release the LMB. The action is ignored when it causes lines to cross or when you move the end point off the wing box.

● **Edit Mode: delete**

When you set the **Edit Mode** menu to **delete** and move the cursor into the right View Window, SMART again draws the structure type set on the **Modify** menu in yellow. To delete an individual item, position the cursor on either end point and click the LMB. To delete items select the **Delete All** button at the top of the right View Window.

● **Done Button**

When you complete the definition of the wing box, rib pattern, and spar pattern, the interactive portion of the design work is done. The final model, however, is not calculated until you select the **Done** button. SMART then uses the pattern information that you have specified to create the structural wing geometry model. First, each skin element is checked. Skin elements are the quadrilateral or triangular regions on the upper and lower surfaces, which are created by the intersections of ribs and spars.

For instance, the wing shown in figure 10.1 contains 12 elements. SMART gives you an error message if any of the elements have more than four sides. Five-sided elements are highlighted in red, and six-sided elements are not highlighted. Highlighted elements, which appear to be four-sided, actually have five sides, but one side is too small to be seen on the screen. You must edit the rib or spar patterns so that you have no five- or six-sided elements. Then select **Done** again. The wing is resurfaced so that the upper and lower patch boundaries coincide with the underlying internal structural rib and spar elements that are generated. The leading-edge and trailing-edge portions of the wing outside the wing box are not present in the completed structural model.

Finally, you are prompted to enter a name for your model. As shown in figure 10.3, a group node with the name that you entered is added to the Tree. The node contains both the original geometry

57

and the new wing structure. The resulting model can be written to a PATRAN neutral file. (See the chapter on FILE Mode for more information.)
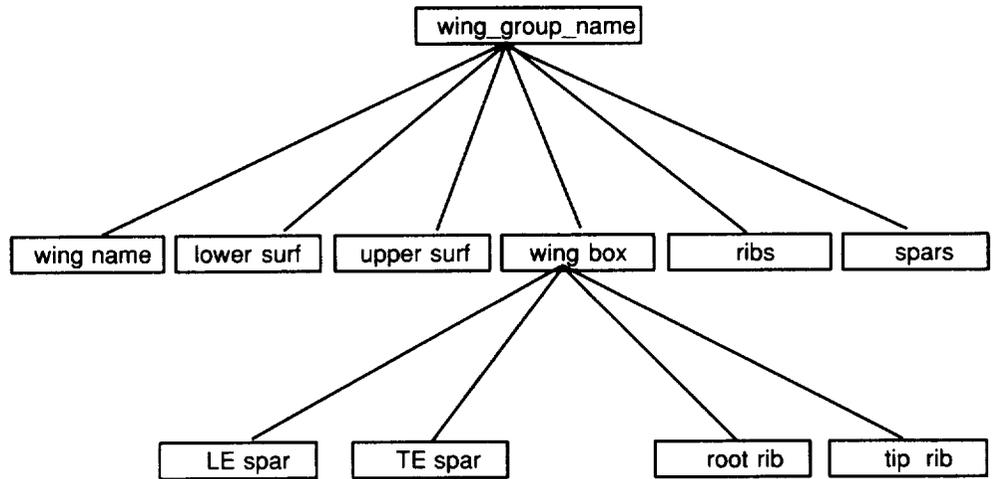
Figure 10.3. Group node hierarchy of structural wing.

Chapter 11
PICTURES Mode

PICTURES Mode lets you draw full-screen pictures of your model. To enter PICTURES Mode, select **PICTURES** from the Mode Menu. The **Display Options** and **File Options** menus appear in the Menu Display Area. (See fig. 11.1.) On the **Display Options** menu, the first three menu options draw full-screen pictures of your model. The last option, **file**, displays images in the Silicon Graphics image format. On the **File Options** menu, the **write** option saves the next displayed picture in an image file. The other three menu options compress, uncompress, and delete existing image files.

File Options	Display Options
write	no background
compress	with background
uncompress	four view
delete	.rgb file

Figure 11.1. Menus for PICTURES Mode.

Display Options Menu

Most options on the Display Options menu draw a full-screen picture based on the display in the right View Window. Before selecting from the menu, check to see that you are satisfied with the view in the right View Window. To display a full-screen picture, perform the following steps:

1. Select one of the following options from the **Display Options** menu:

   • The **no background** option draws exactly the scene that is in the right View Window.

   • The **with background** option draws the scene that is in the right View Window superimposed on a background read from a image file. While scanning the screen for the picture to superimpose, SMART displays your model on a black background. Thus, anything in your model that is black is not superimposed on the background.

59

- The **four view** option draws a split picture that provides four different views of your model. The upper left corner duplicates the view displayed by the right View Window and the other three screen quadrants show the top, side, and front views.

- The **from file** option displays an image that is read from an image file.

2. If you select **with background** or **from file** you must use the File Chooser to indicate the image file that you want. Otherwise, this step is skipped.

3. SMART draws the full-screen picture. Click the LMB to return to the SMART screen.

File Options Menu

To save full-screen pictures in an image file, perform the following steps:

1. Select **write** from the **File Options** menu.

2. Select an option (**no background**, **with background**, or **four view**) from the **Display Options** menu.

3. Enter a name for the image file. SMART appends an *.rgb* extension to the filename that you enter.

A full-screen picture appears and the contents of the screen is saved in the image file. Once the image is saved, the regular SMART screen returns. The image file is stored in SMART *program_data/pictures* directory.

To avoid wasting your computer memory, you should compress image files that are not being used.

1. Select **compress** from the **File Options** menu.

2. Use the File Chooser to select the image file that you want to compress. Compression reduces the size of the selected file and appends *.Z* to the filename. (See the chapter on FILE Mode for more information about the FILE chooser.)

Compressed files cannot be displayed on the screen. To restore a compressed file to its original form, perform the following steps:

1. Select **uncompress** from the **File Options** menu.

2. Use the File Chooser to select the image file to uncompress.

To delete an image file, perform the following steps:

1. Select **delete** from the **File Options** menu.

2. Use the File Chooser to select the image file to delete.

Chapter 12
FILE Mode

When you select **FILE** Mode, three menus, shown in figure 12.1, appear in the Menu Display Area: **File Type**, used for selecting the import or export file format; **File Options**, used for performing storage and retrieval operations on data files having the selected file format; and **Directory Options**, used for organizing projects.

File Type	File Type	File Options	Directory Options
GRIDGEN	SMART	move	change
PLOT3D	LAWGS	read	create
WAVE	PATRAN	write	delete
Property	APAS	append from	
SGO		replace	
		delete	

Figure 12.1. Menus for FILE mode.

The routines in SMART are not translators. Geometry stored in leaf nodes can be either Cartesian cross sections (discrete) or Bezier curves and surfaces (analytic) or both. When generating a file, SMART extracts from your model only the data that are already in the proper form for the specified file format. As the user, you are responsible for meeting the restrictions of a given format. You can use the utilities in SMART to manipulate the geometry into the correct form before you output the data via the desired interface format. For example, patch data can be converted into Cartesian cross sections with the **extract** and **convert** options in CS SURFACE Mode. With the exception of SMART's own format, geometric data are transformed before output. Hierarchy, as represented by group nodes, is not preserved; however, leaf node grouping of data usually is.

The File Chooser shown in figure 12.2 lists files in the current directory. It uses file name extensions to determine a file's format. Files whose extensions differ from those of the selected file format are filtered out. Therefore, files created from an outside source must be named with the proper extension or you cannot select the files. The current file format appears on the title bar of the File Chooser. Each listing in the File Chooser consists of a file's name, its size (in bytes), and the date that it was last modified. The Information Display Area

shows the pathname of the current directory, and a scroll bar on the right lets you search large directories.

Title bar		Scroll bar

SMART MODEL FILES

File pending selection is highlighted

all	10000 bytes	Tuesday	Jan 8	14:51	1992
alphaorbiter	2000 bytes	Saturday	July 8	12:00	1991
beta_orbiter	46840 bytes	Monday	Nov 5	14:51	1991
fuselage	35035 bytes	Monday	Nov	12:10:00	1991
payload	36202 bytes	Tuesday	Nov 20	12:00	1991
tank	22202 bytes	Wednesday	Nov 14	22:22	1991
wing	46840	Saturday	Dec 15	14:51	1991

Figure 12.2. File chooser.

When you execute commands that require you to select an existing file or directory (e.g., **read**, **append**, **move**, **replace**), the File Chooser appears in the center of the screen with the cursor positioned in its center. You select a file by double clicking the mouse on its entry. When you click on the file name the first time, the entry is highlighted. In most cases, a brief description of the file appears in the Textport. Clicking the mouse again selects the file. You can cancel the operation by moving the cursor anywhere outside the View Window.

When you save new files and create project directories, SMART prompts you to enter a unique name. The appropriate file extension is automatically appended to that name. The name can contain up to 12 characters. If you enter an invalid or nonunique name, SMART displays a warning and prompts you to enter another name. You can cancel the operation by pressing <Enter> without typing a name.

The following table shows some characteristics of the different formats:

Format	Extension	ASCII	Binary	Read	Write	Discrete	Analytic
SMART	.s	1	0	1	1	1	1
PATRAN	.pat	1	0	1	1	1	1
LaWGS	.lwg	1	0	1	1	1	0
APAS	.aps	1	0	1	1	1	0
GRIDGEN	.grda	1	0	1	1	1	0
GRIDGEN	.grd	0	1	1	1	1	0
PLOT3D	.fmt	1	0	1	1	1	0
PLOT3D	.bin	0	1	1	1	1	0
Object	.obj	1	0	0	1	0	1
SGO	.sgo	0	1	0	1	1	1
Property	.prp	1	0	0	1	0	0
Image	.rgb	0	1	1	1	0	0
Project directory	.sd	0	0	0	0	0	0

File Type

This section briefly describes the file formats that are supported by SMART and are listed on the **File Type** menus.

• SMART

You can use SMART's own format to store all or a portion of your model. Color maps are also saved. Thus, when you read in a file, any color maps currently defined are overwritten. The View Options and the polygons that are created when you shade an object are not saved.

• LaWGS

Langley Wireframe Geometry Standard (LaWGS) formatted files contain data in (X-, Y-, and Z-) Cartesian coordinate form, describing one or more geometric components. For a complete description of the LaWGS file format, see reference 2.

• PATRAN

You can read in files from PATRAN, a product of PDA Engineering. A PATRAN neutral file consists of up to three sections. Information is contained in over 30 different types of data packets. The following phase-I packet types correspond to SMART Bezier geometry and are input and output: 25 (title), 26 (date), 32 (cubic curve), 33 (bi-cubic patch), 22 (named component), and 99 (end of file). The following phase-II packet types are supported for reading only: 2 (nodes) and 4 (quads, tris, bars). A new packet type 6 is included to provide six channels of analysis data for each element. SMART may halt if you try to read in a PATRAN neutral file that contains packets other than those listed here. PATRAN named components correspond to a leaf node in the Tree. Hierarchy, as represented by SMART's group nodes, is not preserved. A complete description of the format of each data packet in the PATRAN neutral file can be found in reference 3.

65

● **APAS**

SMART geometry in the form of Cartesian coordinates can be exported to or imported from the Aerodynamic Preliminary Analysis System (APAS), which was developed by Rockwell International Corporation. Data to and from SMART flow through the apasin/apasout programs, which convert the ASCII files to and from binary formats for APAS. A number of conditions must be met before your model can be transferred o APAS for analysis. SMART checks your model for these conditions (subsequently discussed) and does not write the file if they are not met. While SMART attempts to write your file, any diagnostics appear in the Textport.

Cross sections must be planar (X-Y, Y-Z, or Z-X). If points are out of the plane by a small amount, SMART makes them planar. Cross sections must be sorted, and they must have the same number of points. If not, SMART breaks the component into multiple components at cross sections where the point count changes. Segment points are ignored. SMART makes each fuselage cross section one APAS segment and each wing cross section two segments.

Fuselages must be aligned along the positive X-axis and the Y-axis out the right wing. Points should start from the top (maximal "z") and be listed in a clockwise direction (looking from back to front). SMART reorders the points if they are in a counterclockwise direction. You should have no more than 25 cross sections in a fuselage and no more than 25 points in a cross section.

Wings must span the Y-axis with points running from the leading edge to the trailing edge along the top and then back to the leading edge along the bottom. This default is used in the wing generator within SMART. You should have no more than 11 cross sections in a wing and no more than 40 points in a cross section. (See ref. 4 for more information about APAS.)

● **GRIDGEN**

SMART can read and write single and multiblock GRIDGEN files in binary and ASCII formats. (GRIDGEN was developed by General Dynamics Corporation.) This technique writes all the coordinates in the X-direction, then the Y-direction, and then the Z-direction. SMART can automatically detect whether a file is binary or text when reading, but SMART prompts you for your preference when writing a file. For more information on GRIDGEN file formats, see reference 5.

● **PLOT3D**

SMART can read and write binary or ASCII PLOT3D files in either single or multiblock format. SMART can automatically detect ASCII files when reading but prompts for your preference when writing files. In both cases, all the coordinates in the X-direction are written to the file, then the Y-direction, and then the Z-direction. To read a PLOT3D

file, SMART prompts you for the file format (single or multiblock). If you answer incorrectly, the geometry is severely distorted. Rereading the file with the correct information rectifies this problem. SMART writes all PLOT3D files (even those containing only one element) in multiblock format. For more information on PLOT3D file formats, see reference 6.

• Object

You can export your SMART model, represented by Bezier patches, to Advanced Visualizer, an image generation product of Wavefront Technologies, Inc. The object files generated by SMART contain geometric vertices, element connectivities, and grouping information. Elements are represented as three- or four-sided elements corresponding to the polygons formed by the wireframe rendering of your model. Object files are written in an ASCII format. For a full description of the Object file format, see reference 7.

• SGO

SMART allows you to use the Silicon Graphics Object (SGO) file format to export your model to IRIS Showcase, a product of Silicon Graphics, Inc. Objects contained in the SGO files generated by SMART are constructed from a list of quadrilaterals. SMART provides two options for writing the SGO file. If your model contains only Cartesian data, a wireframe representation is written. No normal vectors are written to the SGO file; thus, you cannot shade the model inside Showcase. If your model contains Bezier patch data, the SGO file that is written is based on the subdivision of the patches into quadrilaterals, that is, the process that takes place when SMART shades an object. SMART only includes objects that have been shaded in the SGO file. SMART writes the components of the normal vector at each vertex as well as the components of the vertex itself. This means that you can shade your model inside Iris Showcase. The SGO format, which is binary, is fully described in reference 8.

• Property

SMART generates reports that list the physical properties of your model. The itemized properties include volume, area, total weight, center of gravity, and moments of inertia. If you select a group node, the physical properties of all nodes beneath it are also included in the report.

File Options

This section explains the options on the **File Options** menu. The format highlighted on the **File Type** menu is used in all options. Thus, before selecting an option from this menu, be sure that the desired format is selected.

67

● **move**

The **move** option lets you rename a file and optionally relocate it in another subdirectory. First, select **move** from the **File Options** menu, then select the file you want to rename or move from the File Chooser. Next, select the project directory to which you want to move the file from the File Chooser. This step is skipped when no project directories are present. Last, enter a new name for the file at the prompt. If you want the name to stay the same, press <Enter> without typing a name. You can now move another file or exit by moving the cursor outside the View Window.

● **read**

To read a file, select **read** from the **File Options** menu. If you have any objects already defined, SMART asks you "OK to overwrite Tree?" To continue, select yes from the **Verify** menu and select a file to read from the File Chooser. The message "Please Wait" appears in the Function Name Area while the file is read. When the data have been read, the new Tree is displayed in the View Windows. When you read or append a file, a group node, named after the input file and having one or more leaf nodes, is added to the top-left node of the Tree.

As a short cut, the -f command-line option causes SMART to read in the specified model file at program start-up. For example, smart -f <mymodel> causes the model that is stored in mymodel.s to be read. Remember, you cannot read object, SGO, or property file formats. You should rewindow and check visibility (see the chapter on IMAGE Mode) to view geometry after reading or appending a file.

● **write**

The **write** option lets you produce a new file from the current Tree. To write a file, select write from the **File Options** menu. Then use the LMB to click on the node corresponding to the portion of the model that you want to output. Selecting the system node saves your complete mode. At the prompt, type in a file name. In most cases, SMART asks you to enter a brief file description (80 characters or less). If you selected the system node, SMART then prompts you to type a new node name before writing the file. The message "Please Wait" appears in the Function Name Area while the file is written.

If **Object** is selected from the **File Type** menu, you are provided a menu for indicating whether you want the geometry to be written as three- or four-sided elements. If SGO is selected, you are provided a menu for indicating whether you want to export a wireframe or a shaded geometry to Showcase.

If the **File Type** menu is set to property, SMART displays the Reference Point menu. Set the reference point by selecting an item. SMART then displays this message: "Please wait while properties are

calculated." At the "Include projected areas?" prompt, select either yes or no from the **Verify** menu.

● **append**

The **append** option is similar to the **read** option except that the current model is not overwritten. Instead, SMART appends the data stored in the file to the existing model. This option can be used to copy objects from one SMART file to another.

● **replace**

The **replace** option is similar to the write option except that you choose an existing file.

● **delete**

The **delete** option lets you remove files. You are prompted to verify each file selection.

NOTE: *If program execution is interrupted, SMART first attempts to write a file named RECOVER.s in the current directory. You can often recover your work by reading this file. However, the best way to protect your model from being lost is to frequently replace it.*

Directory Options

This section explains the options on the **Directory Options** menu and describes the directory organization and manipulation utilities.

● **change**

The current directory is the one that SMART uses for file storage and retrieval. While in FILE Mode, the current directory is displayed in the Information Display Area. The default current directory is set to be the home directory. Unless modified by one of three mutually exclusive command-line options, your home directory is the name of the directory keyed to your UNIX login id, stored in the file $(SMART)/program_data/users.dat. To change your current directory, select change from the Directory Options menu, then select a directory from the File Chooser. Your home directory is the last item listed.

The following command-line options change your home directory: -c uses your working directory as the home directory, which is the directory name that is printed if you type "pwd" at the UNIX prompt; -projname uses a directory named $(SMART)/projects/<projname> as the home directory. This option is designed to let multiple users access the same files; -username uses another user's home directory (as given in users.dat) as your home directory. For example, smart -u bob allows you to read data files that are stored in Bob's home directory. However, you cannot write files unless Bob has set his permissions to

allow write access to users other than himself. If you have at least read permission on the home directory, SMART will execute. If you do not have write permission, SMART tells you so. If you have neither read nor write permission, SMART does not run. With the exception of picture files, all files are stored and retrieved from the current directory. Image files (.rgb) that SMART reads and writes while in PICTURES Mode are stored in $(SMART)/program_data/pictures.

● **create**

You can **create** one level of subdirectories beneath your home directory, these subdirectories are referred to as project directories. You can have as many project directories as you want. These directories are different from home directories specified by the -projname option previously discussed. To create a new project directory, select **create** from the **Directory Options** menu. At the prompt, type in a name for the new directory.

● **delete**

With the delete option, you can remove a file or a project directory and any data files that it contains. To delete a project directory, select **delete** from the **Directory Options** menu. Use the File Chooser to select the project directory to delete. If the directory is not empty, SMART asks you to confirm the deletion. You can continue to delete project directories until either all the project directories are removed or you move the cursor outside the View Window, however, you cannot delete the current directory.

Appendix A
Getting Started Exercise

If you are familiar with graphical user interfaces, then the easiest way to become familiar with SMART is to follow the exercise in this appendix. Performing the exercise gives you an introduction to the PRIMITIVES, PACKAGING, TREE, IMAGES, and FILE Modes as well as an introduction to the user interface. If you have the software installed, simply type "smart" at the command line. (See appendix C for a description of the command-line options.) An animation appears on the screen until you hit any key, then the main screen appears.

Choose **PRIMITIVES** from the Mode Menu. Create a few primitive objects by selecting any menu choice in the Menu Display Area (the bottom section of the screen). For now, avoid creating tanks and wings; just use the standard shapes. The Textport prompts you for the names and dimensions of your objects.

Notice that all shapes are created with their center points at (0,0,0). To change the relationships of the objects with one another, you need to use the PACKAGING Mode. Choose **PACKAGING** from the Mode Menu and experiment with rotation, scaling, and translation of the various objects. Click and drag the mouse over the **ROTA** and **PLNR** View Options beneath the right View Window to obtain different views of your objects. The **WIND** View Option assures that you can see all the objects.

Next, choose the **TREE** Mode from the Mode Menu. You now see a Tree showing all the objects that you created. You can create new nodes that can be used to group objects together so that you can edit them as one unit. Create a new node with the **add** menu option, then use **move** or **copy** to place some of your objects into the new group. Delete one or more objects by using the **delete** option.

Next, go to the IMAGES Mode by choosing **IMAGES** from the Mode Menu. Choose your group node from the Tree. Click on your favorite color map (on the right side of the screen), then on the number **3** on the menu just left of the maps. SMART redraws the objects and shades those objects that you previously placed in the group. Use the **ROTA** and **PLNR** View Options to examine your creation; note the three-dimensional (3-D) effect of the shading.

71

As another experiment, make the various object presentation elements (the menu with **point**, **curve**, etc.) invisible and visible. Finally, choose **FILE** from the Mode Menu and save your file on disk by answering the various prompts in the Textport.

Appendix B
Mathematical Basis for Curves and Surfaces

This appendix describes the mathematical basis for SMART's representation of curves and surfaces.

Curve Representation in SMART

SMART uses parametric cubic curves. A parametric cubic curve is one for which x, y, and z are defined as third-order polynomials for some variable t.

Parametric Cubic Equations

Most shapes are too complex to be represented by just one curve; thus, curve segments are pieced together. Parametric cubic equations are used because they allow up to C2 continuity and handle infinite slope conditions.

Bezier Cubic Curves

A cubic parametric curve can be defined many ways; SMART uses the Bezier formulation. Four points, called control points, specify the Bezier curve. The first and fourth control points form end points of the curve. The two other points, which are generally not on the curve, indirectly define the tangents at the curve's end points. The four control points are referred to collectively as the control polygon of the Bezier curve. Figure B1 shows the control polygon for a Bezier curve segment.
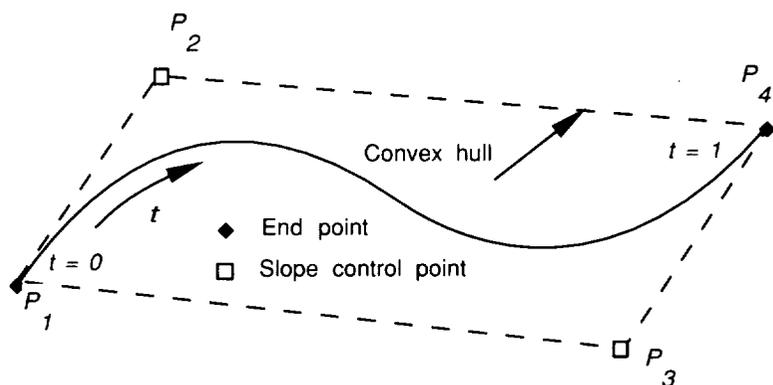
Figure B1. Bezier parametric segment control polygon.

73

One reason that the Bezier form is widely used in graphics systems is that this representation provides the most intuitive control over the shape of the curve. Thus, you can easily mold a curve to its desired shape by moving control points with the mouse.

Curve Mathematics

For a single Bezier curve segment, C, the equations are as follows:

$$C_x(t) = P_{1_x}F_1(t) + P_{2_x}F_2(t) + P_{3_x}F_3(t) + P_{4_x}F_4(t)$$

$$C_y(t) = P_{1_y}F_1(t) + P_{2_y}F_2(t) + P_{3_y}F_3(t) + P_{4_y}F_4(t)$$

$$C_z(t) = P_{1_z}F_1(t) + P_{2_z}F_2(t) + P_{3_z}F_3(t) + P_{4_z}F_4(t)$$

where P_i represents the Bezier control points of the individual curve and $F_i(t)$ represents the blending functions given by the following equations:

$$F_1(t) = (1 - t)^3$$

$$F_2(t) = 3t(1 - t)^2$$

$$F_3(t) = 3t^2(1 - t)$$

$$F_4(t) = t^3$$

where t is the local parameter varying between 0 and 1.

Note, when $t = 0$, then $F_1 = 1$ and $F_2 = F_3 = F_4 = 0$. This fact ensures that the curve passes through point P_1. By similar reasoning, the curve must also pass through point P_4 at $t = 1$. This characteristic of Bezier curves is called end point interpolation.

Points P_2 and P_3 are called slope control points because they lie on the lines containing the tangents to the curve at points P_1 and P_4. Figure B2 shows how moving a slope control point changes the shape of a curve segment.
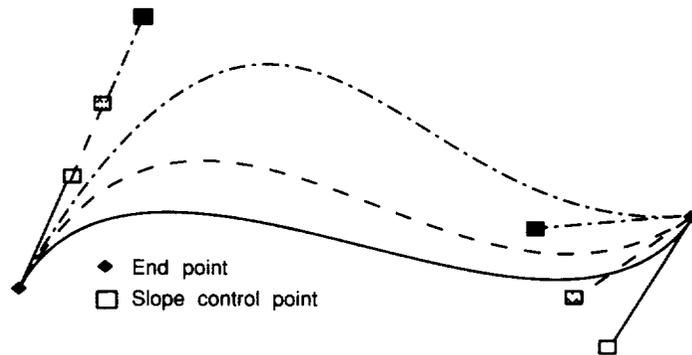

Figure B2. Effect of moving slope control point on curve shape.

Figure B3 shows that for curve A to be continuous with curve B, points A_4 and B_1 must coincide. For C1 continuity, the control points A_3, A_4, B_1, and B_2 must also be colinear.
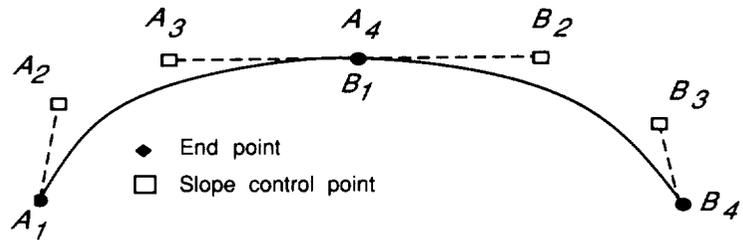

Figure B3. Continuity conditions of piecewise Bezier curve.

The convex hull property is an important characteristic of Bezier curves. This property states that the curve generated from a set of control points lies completely within the convex polygon surrounding the points. Although not of great practical use to the designer, this property is exploited in devising efficient processing algorithms to improve response time in SMART's interactive environment.

Surface Representation in SMART

This section extends the information about cubic curve representation to bicubic surface representation.

Bicubic Surfaces as Bezier Patches

In SMART, surfaces are composed of Bezier patches. As with curves, patches can be joined together in a piecewise manner to form more complex surfaces. Each Bezier patch is defined by cubic equations of two parameters, s and t. Varying both parameters from 0 to 1 defines all points on the surface patch. Bicubic surfaces can provide continuity of position, slope, and curvature at the points where two patches meet. In addition, bicubic parametric equations allow for handling the large gradients that can be encountered in dealing with arbitrarily shaped surfaces. This formulation results in each coordinate specification being a function of one or two monotonically increasing independent variables.

Control Net

Sixteen control points are required to specify a bicubic Bezier patch. The collection of all 16 points is referred to as the control net. Figure B4 shows the control net for a surface patch. The primary reason for using the Bezier representation is that the control net provides the designer with an intuitive handle for making shape modifications.
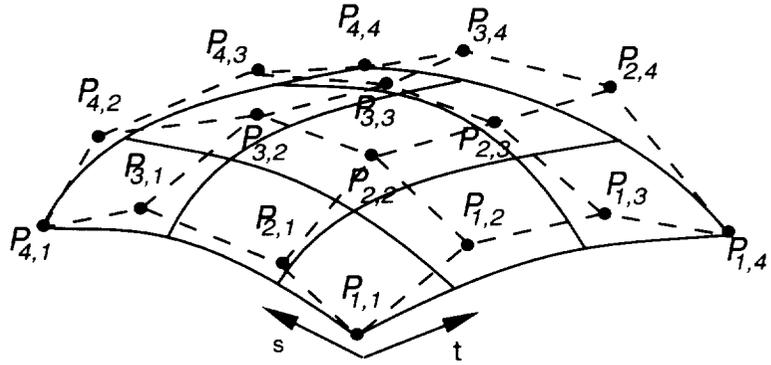
75

Figure B4. Control net for Bezier bicubic surface patch.

Patch Mathematics

The equation of a patch, P_i, is as follows:

$$\mathbf{P}_i(s,t) = \mathbf{S}\mathbf{M}_b\mathbf{G}_i\mathbf{M}_b^T\mathbf{T}^T$$

where $\mathbf{G}_i$ is the matrix of the 16 control points and $i = x, y$, or z, and the matrices $\mathbf{S}$ and $\mathbf{T}$ are as follows:

$$\mathbf{S} = [s^3, s^2, s, 1]$$

$$\mathbf{T} = [t^3, t^2, t, 1]$$

and the cubic Bezier basis matrix, $\mathbf{M}_b$, is as follows:

$$\mathbf{M}_b = \begin{bmatrix} 1 & 3 & -3 & 1 \\ 3 & -6 & 3 & 0 \\ -3 & 3 & 0 & 0 \\ 1 & 0 & 0 & 0 \end{bmatrix}$$

Reference 9 provides a more detailed discussion of curves, surfaces, and their interesting properties (e.g., affine invariance, variation diminution, symmetry, linear precision, continuity requirements).

Appendix C
UNIX Command-Line Options

Name	smart—Solid Modeling Aerospace Research Tool
Synopsis	**smart [-wchxF]**— [[**-u** user] — [**-p** *project*]] [**-f** [*dir.sd/*]*file*]
Description	The Solid Modeling Aerospace Research Tool (SMART) software package is used in the conceptual design of aerospace vehicles. It provides a highly interactive and dynamic capability for generating geometries with Bezier cubic patches. Features include automatic generation of commonly used aerospace constructs (e.g., wings and multi-lobed tanks); cross-section skinning; wireframe or shaded presentation; area, volume, inertia, and center-of-gravity calculations; and interfaces to various aerodynamic and structural analysis programs. SMART is used by a number of organizations at the Langley Research Center, other NASA Centers, and at some aerospace corporations.

Options

-w	Use the current window as the Textport rather than the default of creating its own.
-c	Use the current working directory rather than the directory listed in the *users.dat* file.
-x	SMART normally starts with a graphics display, showing the program name and version number. When any key or mouse button is depressed, the program then proceeds. When invoked with the -x option, this display is omitted.
-F	Normally, SMART does its best to be sure that you are running on a console before launching SMART and disrupting the actual user of the console, since it is easy to accidently run SMART from a window opened to another host or even from a text-based terminal. This option overrides that check and executes SMART, regardless of who SMART thinks is at the console.

77

-p project When properly configured by the SMART adminis-
trator, SMART supports multiple-user projects, so
many users can access the same set of SMART files.
If no project is specified, SMART assumes you want
to use your default directory specified in the special
users.dat file. If your login name or project is not
found in the *users.dat* file, SMART warns you and
uses the current working directory to read or write
files. If you want to set up a SMART project, see
your SMART administrator.

-u user Use another *user's* files rather than your own.
Unless they change permissions on their SMART
directory, then you can only read not write files.

-c The **-c** option tells SMART to use the current
working directory as the source for the SMART files
and ignore the *users.dat* file.

-f The **-f** option allows you to specify an initial model
file with an optional project subdirectory (with the
.sd suffix) if needed. If the file is found, SMART
reads in the file before starting, so you do not have
to use the FILE menus to get your file. If you
plan to work many sessions on the same model,
this option is useful when combined with the *alias*
feature in C-shell. You should not use wildcards or
the *.s* file suffix on the command line.

Example The following command

smart -xf rocket.sd/Atlas6

starts SMART without initial graphics and automatically reads in the
"Atlas6" smartfile from the "rocket" subdirectory in the default project
directory.

Environment The exported shell variable SMART must be defined in order to work
properly. Normally, this variable is automatically set up by the system
administrator.

78

Files

$SMART	SMART root directory
$SMART/Smart	SMART executable
$SMART/smart	SMART shell wrapper
$SMART/doc	SMART manpage source files
$SMART/projects	Group project area
$SMART/program_data/users.dat	Default logins and projects
$SMART/program_data/fonts	The fontfiles used by SMART
$SMART/program_data/airfoils	Files that define NACA airfoils
$SMART/program_data/pictures	Storage area for background image files and image files generated in SMART
$HOME/.smartrc	An optional file to redefine the color scheme used in SMART

Glossary

action menu—Selecting an option from an action menu executes an operation.

aerospace primitives—Specialized shapes, such as wings and tanks.

autowindow—Centers displayed objects in the View Window.

azimuth—Rotation about vertical axis.

basic primitives—Simple shapes, such as cubes, wedges, and spheres.

Bezier bicubic patch—A four-sided curved surface having 16 control points.

Bezier cubic curve—Defined by four control points. The curve passes through the first and fourth control points and the slope is determined by the second and third control points.

bubble—One chamber of a multilobe tank.

center of rotation—The point about which the ROTA View Option rotates the view.

C0, C1, C2—Curves are C0 continuous if they are joined, C1 continuous if in addition their tangents are continuous, and C2 continuous if their second derivatives are continuous.

color map—A continuous range of color shades used to represent surface characteristics.

control points—Three-dimensional points that control the shape of a curve. See Bezier cubic curve.

CS segment—A set of one or more curves within a cross section that have guaranteed slope (C1) continuity.

CS SURFACE Mode—Provides commands for (1) generating cross sections that are composed of a sequence of either curves or points and (2) using cross sections to create surfaces.

current directory—SMART uses this directory to store and retrieve data files.

depth cueing—Changes display in a View Window so that the intensity of a line varies with its distance from the viewer. The points on a line farther from the eye are darker, so the line seems to fade in the distance.

dimension object node—A leaf node used to indicate a distance.

dolly—To change the distance between the user and the object.

element—A geometric entity such as points, curves, patches, and quads. In STRUCTURES Mode, element also refers to the polygons formed by the intersection of ribs and spars inside the wing box.

enrich—Subdivide a patch into smaller patches.

extrusion—A method of generating a tube-like surface by translating a cross section along a path so that the plane of the cross section remains perpendicular to the tangent of the path.

File Chooser—Menu that shows the file names in the current directory, file sizes, and modification dates.

FILE Mode—Provides commands for storing, retrieving, and organizing SMART data files.

fillet—A curve created to smoothly join two other curves.

from 4—A surfacing option that creates patches for every four cross sections.

Function Name Area—Identifies the function or operation that SMART is currently performing.

Geometry object—A leaf node containing geometry elements.

grid—Optional display that shows the world coordinate system.

group node—Allows nodes to be grouped together so that they can be manipulated as one object.

help facility—Displays brief information about available options.

home directory—The highest level directory that SMART can use for storing and retrieving data files.

IMAGES Mode—Provides commands for changing the display attributes (e.g., color, visibility, precision, shading) of an object.

Information Display Area—Displays help messages and other information.

intensity—The brightness of a light source.

Langley Wireframe Graphics Standard (LaWGS)—A file format that contains data in X-, Y- and Z-coordinate form describing one or more geometric components.

leaf node—A terminal node in the Tree that represents an individual component in the model.

light object—A leaf node that defines a light source.

longitudinal slope—The slope of a T-line (at a curve end point) that connects two cross sections.

Menu Display Area—Area where menus and slider bars are displayed.

Mode Menu—Horizontal menu used to access categories of related operations available in SMART.

model—The collection of objects that are stored in the Tree.

moment of inertia—The resistance of body to rotational acceleration.

object—Represented by a node in the Tree.

orthographic projection—A projection used to show an object in two dimensions.

PACKAGING Mode—Provides commands for moving, scaling, and rotating objects.

PATCH TOOLS Mode—Provides commands for editing patches; for example, a command that subdivides a patch into smaller patches.

perspective projection—A projection method in which the farther an object or part of an object is from the viewer, the smaller it is drawn.

PICTURES Mode—Provides commands for displaying full-screen pictures of your model and saving pictures in .rgb image files.

pop-up menu—Selecting an option from a pop-up menu changes discrete parameter settings.

83

PRIMITIVES Mode—Provides commands for creating 3-D objects from a limited number of parameters.

project directories—Subdirectories of the home directory.

PROPERTIES Mode—Provides commands that display the linear, area, volume, or mass properties of an object.

reflecting—The creation of a mirror image of an object.

TREE Mode—Provides commands for editing the hierarchical organization of the model.

Tree—Graphical interface that represents the model's hierarchical organization.

quad—A quadrilateral.

reference point—The point about which a moment of inertia is calculated.

revolving—A method of generating a surface by revolving a cross section about an axis.

.rgb image files—Digital image file format.

rib—Line or structure that crosses the wing box chordwise.

shading precision—Controls the number of quads that are generated from each patch. The selected number is the power of four.

skinning—A method of generating a surface from a set of ordered cross sections.

slider bar—Allows selection of a value over a continuous range of values.

s-lines—A family of isoparametric curves on a surface.

spar—Structure that spans the wing box.

STRUCTURES Mode—Provides commands for adding interior structure to a wing.

System Node—The group node that is the root (i.e., top-most node) of the Tree.

switch menu—Selecting an option from a switch menu changes discrete parameter settings.

Text object—Leaf node that contains one line of text (up to 80 characters) aligned around an origin.

T-lines—A family of isoparametric curves on a surface.

twist—The angle of rotation of the view about on axis normal to the screen.

Verify Menu—A menu with yes and no options.

View Options Menu—Controls the manner in which the model is drawn on the screen.

View Window—Used for graphic displays of the model.

View Box—Displays the portion of the Tree that can be selected from.

wing box—Area inside the wing's planform where ribs and spar are defined.

wireframe—Rendering that shows only the edges of polygons.

world space—The 3-D global coordinates in which objects reside.

References

1. Abbott, Ira H.; and Von Doenhoff, Albert E.: *Theory of Wing Sections—Including a Summary of Airfoil Data.* Dover Publ., Inc., c.1959.

2. Craidon, Charlotte B., compiler: *A Description of the Langley Wireframe Geometry Standard (LaWGS) Format.* NASA TM-85767, 1985.

3. *PATRAN Plus User Manual,* Release 2.3. Publ. No. 2191020, PDA Engineering, July 1988.

4. Divan, P.: *Aerodynamic Preliminary Analysis System II. Part II— User's Manual.* NASA CR-165628, 1981.

5. Steinbrenner, John P.; Chawner, John P.; and Fouts, Chris L.: *The GRIDGEN 3D Multiple Block Grid Generation System. Volume II: User's Manual.* WRDC-TR-90-3022, Vol. II, U.S. Air Force, Apr. 23, 1991. (Supersedes WRDC-TR-90-3022, Vol. II, July 1990.)

6. Walatka, Pamela P.; Buning, Pieter G.; Pierce, Larry; and Elson, Patricia A.: *PLOT3D User's Manual.* NASA TM-101067, 1990.

7. *The Advanced Visualizer*^{TM} *User's Guide. Volume 1—Overview Director Model Digitizing.* Wavefront Technologies, Inc., c.1992.

8. *IRIS*® *Showcase*^{TM} *User's Guide.* Doc. No. 007-1303-020, Silicon Graphics, Inc., Nov. 1991.

9. Farin, Gerald: *Curves and Surfaces for Computer Aided Geometric Design: A Practical Guide.* Academic Press, Inc., 1988.

		Form Approved
REPORT DOCUMENTATION PAGE		*OMB No. 0704-0188*

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY*(Leave blank)*	2. REPORT DATE July 1993	3. REPORT TYPE AND DATES COVERED Technical Memorandum	
4. TITLE AND SUBTITLE Solid Modeling Aerospace Research Tool (SMART) User's Guide *Version 2.0*			**5. FUNDING NUMBERS** WU 506-49-11-01
6. AUTHOR(S) Mark L. McMillin, Jan L. Spangler, Stephen M. Dahmen, and John J. Rehder			
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Langley Research Center Hampton, VA 23681-0001			**8. PERFORMING ORGANIZATION REPORT NUMBER** L-17007
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			**10. SPONSORING/MONITORING AGENCY REPORT NUMBER** NASA TM-4431

11. SUPPLEMENTARY NOTES

McMillin and Rehder: Langley Research Center, Hampton, VA; Spangler and Dahmen: ViGYAN, Inc., Hampton, VA.

12a. DISTRIBUTION/AVAILABILITY STATEMENT ████████ Subject Category 61	12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

The Solid Modeling Aerospace Research Tool (SMART) software package is used in the conceptual design of aerospace vehicles. It provides a highly interactive and dynamic capability for generating geometries with Bezier cubic patches. Features include automatic generation of commonly used aerospace constructs (e.g., wings and multilobed tanks); cross-section skinning; wireframe and shaded presentation; area, volume, inertia, and center-of-gravity calculations; and interfaces to various aerodynamic and structural analysis programs. This report provides a comprehensive description of SMART and how to use it.

14. SUBJECT TERMS Mechanical CAD; Surface generation; Vehicle design; Three-dimensional geometry modeling			15. NUMBER OF PAGES 100
			16. PRICE CODE
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT

NSN 7540-01-280-5500

Standard Form 298(Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102

NASA-Langley, 1993