

AD-A238 837



WL-TM-91-316 - FIMM

13G VIRGO

Interactive Graphics for Geometry Generation
and
Visual Interactive Rapid Grid Generation

USER'S MANUAL

Howard T. Emsley
WL/FIMM
Wright-Patterson AFB, Ohio
45433-6553

Aerodynamic Methods Group
Aerodynamics and Airframe Branch
Aeromechanics Division

June 1991

Approved for Public Release, Distribution
Unlimited

Flight Dynamics Directorate
Wright Laboratory
Wright-Patterson AFB, Oh 45433-6553

91-06479



91

102


FOREWORD


This report was prepared under Work Unit 240410A1 entitled "Aerodynamic Design and Analysis Methods" by Howard T. Emsley of the Aerodynamic Methods Group, Aerodynamics and Airframe Branch, Aeromechanics Division, Flight Dynamics Directorate, Wright Laboratory, Wright-Patterson Air Force Base, Ohio.

This report replaces WRDC-TM-90-317.

The report documents the additional capabilities that the Interactive Graphics for Geometry Generation (I3G) Program, originally developed under Air Force Contract F33615-84-C-3001, has acquired between May 1990 and May 1991.

This document has been reviewed and approved.


Howard T. Emsley
Aerospace Engineer


Dennis Sedlock
Chief, Aerodynamics and Airframe Branch

A-1



TABLE OF CONTENTS

	<u>Page</u>
INTRODUCTION.....	1
GENERAL INFORMATION.....	2
DATA STORAGE MODIFICATION.....	6
IGES FORMAT FILE DESCRIPTION.....	7
I3G/VIRGO ERROR MESSAGES.....	8
I3G/VIRGO SAMPLE PROGRAM EXECUTION.....	11
I3G/VIRGO HELP.....	28
1 TOP	28
1.1 TOP / FILE	28
1.1.1 / FILE / OPEN FILE	29
1.1.2 / FILE / TRANSFER SURF	29
1.1.3 / FILE / DELETE SURF	29
1.1.4 / FILE / RENAME SURF	29
1.2 TOP / MANIPULATE	29
1.2.1 / MANIPULATE / BREAK	30
1.2.1.1 / BREAK / SURF AT LINE	30
1.2.1.2 / BREAK / SURF AT POINT	30
1.2.1.3 / BREAK / SURF AT CURVE	30
1.2.1.4 / BREAK / EXTRACT CURVE	30
1.2.1.5 / BREAK / CURVE AT POINT	31
1.2.1.6 /BREAK / EXTRACT SURFACE	31
1.2.2 / MANIPULATE / JOIN	32
1.2.2.1 / JOIN / ABUT SURFACES	32
1.2.2.2 / JOIN / ABUT CURVES	32
1.2.2.3 / JOIN / BUILD	32
1.2.3 / MANIPULATE / CHANGE	33
1.2.3.1 / CHANGE / REORDR N LINES	33
1.2.3.2 / CHANGE / REORDR M LINES	33
1.2.3.3 / CHANGE / REORDER BOTH	34
1.2.3.4 / CHANGE / SWITCH N AND M	34
1.2.4 / MANIPULATE / INTERSECT	34
1.2.5 / MANIPULATE / TRANSFORM	35
1.2.5.1 / TRANSFORM / SCALE	35
1.2.5.2 / TRANSFORM / TRANSLATE	35
1.2.5.2.1 / TRANSLATE / PICK LOCATION	35
1.2.5.2.2 / TRANSLATE / X,Y,Z TRANS	35
1.2.5.3 / TRANSFORM / ROTATE	35

TABLE OF CONTENTS (Continued)

		<u>Page</u>
1.2.6	/ MANIPULATE / SMOOTH	36
1.2.7	/ MANIPULATE / CUT PLANE	36
1.2.8	/ MANIPULATE / SURFACE_OFFSET	37
1.3	TOP / GEN_SURFACE	37
1.3.1	/ GEN_SURFACE / POINT	37
1.3.1.1	/ POINT / MANUAL_INPUT	37
1.3.1.2	/ POINT / OFFSET	37
1.3.1.3	/ POINT / SURFACE_POINTS	38
1.3.2	/ GEN_SURFACE / CURVE	38
1.3.2.1	/ CURVE / MANUAL_INPUT	38
1.3.2.2	/ CURVE / OFFSET	38
1.3.2.3	/ CURVE / BEZIER	39
1.3.2.3.1	/ BEZIER / RELATIVE	39
1.3.2.3.2	/ BEZIER / BLENDED	39
1.3.2.3.3	/ BEZIER / EXISTING	40
1.3.2.4	/ CURVE / CONIC	40
1.3.2.4.1	/ CONIC / CIRCLE	40
1.3.2.4.2	/ CONIC / ELLIPSE	40
1.3.2.4.3	/ CONIC / SPACE_ELLIPSE	41
1.3.3	/ GEN_SURFACE / SURFACE	41
1.3.3.1	/ SURFACE / MANUAL_INPUT	41
1.3.3.2	/ SURFACE / SURFACE_TFI	41
1.3.3.3	/ SURFACE / COONS_PATCH	42
1.3.3.4	/ SURFACE / BODY_OF_REV	42
1.4	TOP / GRID_SURFACE	42
1.4.1	/ GRID_SURFACE / PANEL	42
1.4.1.1	/ PANEL / RELATIVE	43
1.4.1.2	/ PANEL / ABSOLUTE	43
1.4.1.3	/ PANEL / BLENDED	44
1.4.1.4	/ PANEL / CURVATURE	44
1.4.1.5	/ PANEL / USER_SPECIFIED	44
1.4.1.6	/ PANEL / EXISTING	45
1.4.1.7	/ PANEL / EQUAL_X_Y_OR_Z	45
1.4.1.8	/ PANEL / LIKE_OPPOSITE	45
1.4.2	/ GRID_SURFACE / MOVE_POINT	45
1.4.2.1	/ MOVE_POINT / NODE_POINT	45
1.4.2.2	/ MOVE_POINT / KEYBOARD	46
1.4.2.3	/ MOVE_POINT / ARBITRARY_FNT	46
1.4.3	/ GRID_SURFACE / ABUT_POINT	46
1.4.4	/ GRID_SURFACE / DELETE_POINT	46
1.4.5	/ GRID_SURFACE / DELETE_LINE	47
1.4.6	/ GRID_SURFACE / ADD_LINE	47

TABLE OF CONTENTS (Continued)

		<u>Page</u>
1.5	TOP / BLOCK_BUILD	47
1.5.1	/ BLOCK_BUILD / POINTS	47
1.5.2	/ BLOCK_BUILD / EDGES	48
1.5.3	/ BLOCK_BUILD / SPLINE	48
1.5.4	/ BLOCK_BUILD / TFI	49
1.5.5	/ BLOCK_BUILD / ELLIPTIC GRID	50
1.5.5.1	/ ELLIPTIC GRID / SURFACE	52
1.5.5.2	/ ELLIPTIC GRID / PLANE	53
1.5.5.3	/ ELLIPTIC_GRID / PARAMETRIC	53
1.5.6	/ BLOCK_BUILD / HYPERBOLIC GRID	53
1.5.7	/ BLOCK_BUILD / METRIC ANALYSIS	54
1.5.7.1	/ METRIC_ANALYSIS / DX/D(XI)	55
1.5.7.2	/ METRIC_ANALYSIS / DY/D(XI)	55
1.5.7.3	/ METRIC_ANALYSIS / DZ/D(XI)	55
1.5.7.4	/ METRIC_ANALYSIS / DX/D(ETA)	55
1.5.7.5	/ METRIC_ANALYSIS / DY/D(ETA)	55
1.5.7.6	/ METRIC_ANALYSIS / DZ/D(ETA)	55
1.5.8	/ BLOCK_BUILD / SURFACE MAPPING	55
1.5.8.1	/ SURFACE_MAPPING / CONSTANT_PLANE	55
1.5.8.2	/ SURFACE_MAPPING / INTERPOLATE	56
1.5.9	/ BLOCK_BUILD / BEZIER	56
1.5.9.1	/ BEZIER / CREATE BEZIER	56
1.5.9.2	/ BEZIER / CURVE FIT	57
1.5.9.3	/ BEZIER / DEG_ELEVATION	57
1.6	TOP / OUTPUT	57
1.6.1	/ OUTPUT / PANEL CODES	57
1.6.1.1	/ PANEL_CODES / HESS	58
1.6.1.2	/ PANEL_CODES / MCAERO	58
1.6.1.3	/ PANEL_CODES / PANAIR	58
1.6.1.4	/ PANEL_CODES / VSAERO	58
1.6.1.5	/ PANEL_CODES / S-HABP	59
1.6.1.6	/ PANEL_CODES / QUADPAN	59
1.6.2	/ OUTPUT / GRID GENERATRS	59
1.6.2.1	/ GRID_GENERATRS / EAGLE	59
1.6.2.2	/ GRID_GENERATRS / PGRID	59
1.6.2.3	/ GRID_GENERATRS / PRE-SCRAMG	60
1.6.2.4	/ GRID_GENERATRS / SCRAMG	60
1.6.2.5	/ GRID_GENERATRS / WGRID	61
1.6.2.6	/ GRID_GENERATRS / ZGRID	61
1.6.3	/ OUTPUT / FLO59	61
1.6.4	/ OUTPUT / FLO67	62
1.6.5	/ OUTPUT / WIBCO	63
1.6.6	/ OUTPUT / AERHET(DSCEOM)	63
1.6.7	/ OUTPUT / PNS(QUICKII)	64

TABLE OF CONTENTS (Continued)

	<u>Page</u>
1.6.8 / OUTPUT / BLOCK	64
1.6.9 / OUTPUT / GRIDGEN	64
1.6.10 / OUTPUT / IGES	64
1.7 TOP / END	65
2 PERMANENT FUNCTIONS	66
2.1 PERMANENT FUNCTIONS / CENTER_DSPLY	66
2.2 PERMANENT FUNCTIONS / CNTR_SCL_DSPLY	66
2.3 PERMANENT FUNCTIONS / SCALE_DSPLY	66
3 IMMEDIATE FUNCTIONS	67
3.1 IMMEDIATE FUNCTIONS / RESET	67
3.2 IMMEDIATE FUNCTIONS / HELP	67
3.3 IMMEDIATE FUNCTIONS / ANALYSIS	67
3.4 IMMEDIATE FUNCTIONS / SHELL	67
4 GENERAL NOTES	69
4.1 GENERAL NOTES / PROGRAM_DIMENSIONS	69
4.2 GENERAL NOTES / SCREEN_DISPLAY	69
4.3 GENERAL NOTES / SIDE_NUMBERING	70
4.4 GENERAL NOTES / SURFACE_NORMAL	70
4.5 GENERAL NOTES / SURFACE_NAMING	70
4.6 GENERAL NOTES / TEKTRONIX_4115	70
4.7 GENERAL NOTES / IRIS_TERMINALS	71
INSTALLATION.....	A-1
REFERENCES.....	R-1

TABLE OF FIGURES

<u>Figure No.</u>	<u>Title</u>	<u>Page</u>
1	I3G/VIRGO Capabilities.....	13
2	I3G/VIRGO Surface Nomenclature.....	14
3	I3G/VIRGO Surface Display.....	15
4	Greeting Screen 1.....	16
5	Greeting Screen 2.....	16
6	I3G/VIRGO User Interface.....	17
7-a	IGES Point Surface Definition.....	18
7-b	IGES Point Surface Definition (Cont.).....	19
8-a	IGES Example File.....	20
8-b	IGES Example File (Cont.).....	21
8-c	IGES Example File (Cont.).....	22
9	OPEN / FILE.....	23
10	IGES to TEMP.....	24
11	ANALYSIS.....	25
12	PANEL.....	26
13	SURFACENEW.....	27

**THE INTERACTIVE GRAPHICS FOR GEOMETRY GENERATION PROGRAM (I3G)
AND
THE VISUAL INTERACTIVE RAPID GRID GENERATION PROGRAM (VIRGO)**

INTRODUCTION

Advances in the computational aerodynamics field have provided the aerospace community with tools to analyze complex configurations in a wide variety of flight regimes. Attempts to apply these analysis tools to specific problems, however, have indicated that the process of geometry generation and manipulation is often the most time consuming part of a computational aerodynamic analysis. Recognizing the need for rapid geometric modeling, the Interactive Graphics for Geometry Generation Program (I3G) was developed under Air Force contract F33615-84-C-3001 and the Visual Interactive Rapid Grid Generation Program (VIRGO) was later developed by Lt. David Amdahl of WRDC/FIMM. These programs have been modified, expanded and combined, over the past several years into a tool capable of performing very complex manipulations of geometric configurations. I3G/VIRGO, is designed to aid the engineer in the generation, manipulation, and output of computational aerodynamic models for a variety of flow solvers.

I3G/VIRGO uses interactive computer graphics to display the geometric model as it is being manipulated or built. The user views, picks and manipulates geometries via mouse and keyboard inputs and is provided the flexibility to generate both the surface and field grids of complex configurations. Because the user visually sees what is being generated, errors are easier to detect, and fewer are made.

Discussions were held with experienced analysis code users prior to the design of I3G and VIRGO, and continual inputs during development have assured an optimal human interface. The operation of I3G/VIRGO is intimately tied to its use of menu selecting and prompting to allow both the experienced and uninitiated user to input information easily. The program also contains an on-line help capability that can be requested by the user at any point during program operation.

The basic philosophy behind the operation of I3G/VIRGO is to supply low

1. I3G is documented in AFWAL-TR-87-3064 (Vols. I and II) "Configuration Data Management System". For additional information on VIRGO the reader is directed to "Interactive Multi-Block Grid Generation" in Numerical Grid Generation In Computational Fluid Mechanics '88 from the Second Conference on Grid Generation in Computational Fluid Dynamics.

level geometry manipulation functions that are interactively driven by the user. This approach gives the user a large degree of control over the geometry modeling process. Figure 1 shows the current list of geometry modeling options provided to the user. An explanation of the various program options and capabilities is provided in the I3G/VIRGO HELP section which contains a listing of the information in the on-line help file that is used within the program.

The code is written in standard FORTRAN-77, with the exception of; machine extensions to allow inclusion of external files into the body of the source code, subroutines which serve as graphics drivers, and a C routine which allows the program to open an external shell. I3G/VIRGO is not limited to the current options. Due to its modular nature, extensions can be added to the code with minimal effort.

I3G/VIRGO has six major functional areas: surface manipulation, surface grid-point generation and smoothing, surface and far-field generation, display functions, file management, and output functions. In each of these areas, capability was designed to surpass that afforded by existing manual and computerized methods, with specific emphasis on surface point generation, surface manipulation and graphic display. In the area of surface generation, I3G/VIRGO does not try to compete with CAD systems, which are more suited for performing this function. Limited capability is provided within I3G/VIRGO to create new surface descriptions.

GENERAL INFORMATION

Once the user has entered the program, I3G/VIRGO can operate on one or two geometry files at a time. A geometry file can be opened by the program as either a **temporary** or **permanent** file, however, only one of each can be open at the same time. The choice of **temporary** versus **permanent** is decided by the operations that the user wants to use in a particular session. A permanent file has a restricted group of operations that are valid: surface renaming, deleting and transferring. In a **temporary** model all of the operations are valid, and surface definitions may be brought into the model through Initial Graphics Exchange Specification, IGES, formatted files. Once a file is opened, it remains open until a different file is opened in its place, or the session ends.

I3G/VIRGO allows the user to manipulate three-dimensional surfaces. These surfaces can be either a full surface made up of one or more curves, a curve made up of one or more points, or a point defined by a three-dimensional coordinate. Surfaces made up of more than one curve do not have to have the same number of points on all of the curves. When dealing with surfaces that contain discontinuities, such as kinks or creases, the user is advised to break the surface along the discontinuity, manipulate the individual pieces and then rejoin the surfaces. I3G/VIRGO has the functions to perform these operations

quickly, and breaking the surface first will ensure that the discontinuity is maintained and not smoothed out.

Surface names should be limited to 15 alphanumeric characters. Duplicate surface names are not allowed and if the generation of one is attempted I3G/VIRGO will automatically generate a new name, ZSURF*n*.

I3G/VIRGO's surface nomenclature is shown in Figure 2. The curves that define a surface are called N-lines, and the curves that connect the points going from one N-line to the next (assuming there are the same number of points) are the M-lines. The first N-line of the surface is side 1, and the first line segment of side 1 (which is highlighted) touches side 2. Sides 3 and 4 follow in order.

Figure 3 illustrates the different surface representations within I3G/VIRGO and how they are displayed. A rectangular surface having the same number of points on each defining curve is displayed as quadrilateral elements. For non-rectangular surfaces not having the same number of points on each curve, the defining curves (N-lines) are displayed along with the first and last M-lines. Surfaces defined only as points with no connectivity are displayed as points. Note that the actual surface that the program is working with is the non-linear fit through the displayed points and that the display is a representation of the surface using straight-lines to connect the points.

Upon executing the program, informational greeting screens should appear. The first of these screens, Figure 4, indicates the size limitations of the surfaces produced in I3G/VIRGO, and the second screen, Figure 5, (if operating on an IRIS Workstation) informs the user of special key and mouse functionality. New users should pay special attention to this screen until they become familiar with its information. After the second screen is removed, the I3G/VIRGO window (Figure 6) will appear on the screen. This window will take up the entire screen, but it can be pushed, popped, iconified and closed by moving the cursor to the uppermost part of the window and pressing the right mouse button. The last option, **Quit i3gvir** should only be used as a last resort if the workstation becomes "locked up". Additionally, the window should not be resized or moved due to the pixel defined nature of the program.

The I3G/VIRGO display window is subdivided into a number of different windows (Figure 6).

The file area of the display indicates the current active model (**PERMANent** or **TEMPorary**).

The Mode indicates whether the Surface List area is being used to **INPUT** surface names into the program or to toggle on and off the **DISPLAY** of surfaces.

The names of the surfaces in the current active model are shown in the Surface List area and the Scroll box is used to scroll this list **UP** or **DOWN**.

The Axis represents I3G/VIRGO's right-handed coordinate system and is used to eliminate the ambiguity of the surface display. On several systems, depthcuing of the axis is used to aid the user in establishing the direction of the axis.

The box at the bottom of the display contains user Prompts, user Keyboard Input and Error Messages. It should be noted that the error message area is occasionally used to display notes and hints as particular operation are selected.

Immediate functions, found in the lower right corner of the display, allow the user to **RESET** the display view to the initial orientation used by the program, access the **HELP** file, do an **ANALYSIS** of a surface, or open a **SHELL** outside of the program.

Permanent functions, found above the immediate functions, modify the display of surfaces (for additional information see part 2 of the help section).

Some menu items contain a list of Options which are shown in the options box.

Menu items are listed in the menu box and the trail of user selected menu items is shown in the Menu Trail.

Finally, there is the Geometry Display box which is the display window for the surfaces.

Once a geometric model has been generated in I3G/VIRGO, the **OUTPUT** function is used to output the surfaces in the proper format for the particular analysis code selected by the user. The output file name is defaulted to I3G.OUT, however the user can specify a different name by opening an output file (see section 1.1.1) prior to outputting the model.

At the end of an I3G/VIRGO session (when **END** is picked), the user is questioned as to whether the I3G log file is to be deleted. The I3G log file (named I3Gx.LOG, where x is either absent or has a value between 1 and 9) is a log of the interaction between the user and program during the session. Under normal operation the log file need not be saved. It is, however, useful in understanding why problems may have occurred and therefore, should be retained if problems were encountered during program operation. In addition, the **TEMP**orary or **PERMAN**ent file that was modified or created in the session is stored in a compressed direct access format that should only be accessed through the program. To view any of the surface coordinates outside of the program, the surfaces in question should be output in the manner discussed above.

*******ATTENTION*******

If the session prematurely aborts for any reason (program "locks up" or stops without picking **END**), the user should be wary of the resulting **TEMP** file. The old surfaces have not been damaged but, partially

written records stored on the file will confuse I3G/VIRGO the next time the file is accessed. To correct this situation, I3G/VIRGO should be started, the file opened and the session ended correctly. This will rewrite the file and "clean up" up any partially written records.

DATA STORAGE MODIFICATION

I3G/VIRGO has been hosted on a variety of different platforms during its development. The Fortran-77 coding is extremely portable, however, data storage structure and graphical interfaces are two major areas where platform differences appear. I3G/VIRGO uses a direct access file format to store surfaces in the TEMP/PERM files for easy location, retrieval, addition and deletion. Due to this format however, I3G/VIRGO is subject to direct access differences between the Silicon Graphics' Iris and the previous hosts.

Recent Iris operating system upgrades have modified the direct access format and its word/byte/bit definition to conform to existing 'standards' used by other systems. In lieu of these changes, the most recent version of I3G/VIRGO has been modified to reduce the amount of empty space contained within each TEMP and PERM file. Unfortunately, this makes the old files unreadable by the new version. To alleviate this problem, a code (**packfile.f**) has been written to directly convert old TEMP/PERM files into the new smaller TEMP/PERM files directly.

To convert a file, **packfile** is executed, and the user is prompted for the old file name and a new file name. When running, **packfile** will display diagnostics about each surface in the file. Upon completion, the user is informed of the number of surfaces read and written. If the numbers are not equal then some surface(s) in the original file is not readable. In most cases this is caused by erroneous data that may have been present in the original file for some time. If the user is concerned that something has been lost, the diagnostics provide a list of the surfaces that were readable.

After execution of **packfile**, the new TEMP file should be approximately half the size of the original. This new file is now ready for use in version 4.75 of I3G/VIRGO.

IGES FORMAT FILE DESCRIPTION

The I3G/VIRGO system has been developed to accept data from a number of geometry sources. This wide range of sources is manageable by developing interfaces which all output into a single versatile format. The format that was chosen is that of the Initial Graphics Exchange Specification (IGES).

Since point surfaces, which are currently used within I3G/VIRGO, are not specifically supported by IGES, the IGES User Defined Entity types were employed to define the IGES format for a point surface.

A complete IGES format description is beyond the scope of this documentation, but is available in Reference 1. Briefly, the IGES file for a point source has a Directory section which includes one entry for each entity in the Parameter section. Directory section entries each consist of two, eighty character records, which contain entity information and a pointer into the Parameter section. The Parameter section contains the actual entity data. Entity type 406 is used to carry surface names in the IGES file.

Under the initial contract a new IGES entity type 5001 was established for a point definition surface to be used with I3G. Point definition surface entries in the Parameter section contain the following information:

Entity type	- 5001
2D-3D flag	- specifies 2-D or 3-D coordinates in surface description (1=2D or 2=3D)
Number of curves	
Array of number of points on each curve	
Total number of points	
Z	- Third coordinate if 2-D points input, else 0.0
Coordinates	- X, Y, (Z)....
Number of Pointers	- =0 not used
Number of Parameters	- =1 if entity 406 is used for name, else =0
Parameter pointer	- Pointer to proper 406 entity in Directory section

The details of the point surface IGES format are given in Figures 7(a,b), and Figures 8(a,b,c) show an abbreviated version of the IGES file used in the sample program execution.

I3G/VIRGO ERROR MESSAGES

I3G/VIRGO is structured such that for most errors that are encountered during program operation, a message is printed out for user viewing and then the program automatically returns the user environment to a higher level in the program - very few errors will cause the program to abort. Most of the error messages are self-explanatory, and can be avoided by inputting the correct information at the prompt. Those which may not be clear are listed below.

1) 'Not valid surface type for this operation'

Explanation: Certain operations require that the surface be a rectangular surface.

User Action: Generate a rectangular surface to operate on.

2) 'IGES file nested too deep'

Explanation: The IGES file being processed has too many nested groups.

User Action: See the system manager to increase the program dimensions.

3) 'Attempt to read beyond IGES directory'

Explanation: Bad pointers in the IGES file have caused a read error in the directory portion of the file.

User Action: The error in the IGES file must be corrected.

4) 'Error reading IGES file'

Explanation: An error has occurred in reading the information from the IGES file.

User Action: Check the IGES file, and possibly regenerate the file.

5) 'Breaks at surface edge or outside are not allowed'

Explanation: In breaking the surface at a point, the break point input cannot be on the surface edge or outside the surface.

User Action: Input a proper break point.

6) 'Error in calculating point parameters'

Explanation: Incorrect parametric values have been determined based on a X, Y, Z coordinate input.

User Action: Input point again, making sure point is on desired surface.

7) 'Max number of surfaces to join reached'

Explanation: The maximum number of surfaces that may be joined at one time has been exceeded.

User Action: The join operation will continue to completion with the maximum number of surfaces allowed. The user should then form another join operation with the new surface and the remaining ones to be joined. Check the number of surfaces that can be joined in i3g.prm.

8) 'Error retrieving side point'

Explanation: An error has occurred in a low level surface handling function.

User Action: Retain the I3G log file and see the system manager.

9) 'Error in calculating surface point'

Explanation: A coordinate X,Y,Z point on a surface could not be calculated from the parametric values.

User Action: Redo the operation, making sure that the information input through the picking operation is correct.

10) 'Error - Surface not a point definition surface'

Explanation: The operation is not valid for non-point definition surfaces, i.e., the surface should be made up of points that combine into one or more curves.

User Action: Generate a point definition surface using the **GEN SURFACE** function or correctly input the surface again if it was incorrectly entered the first time.

11) 'Error opening requested file'

'Error opening output file'

Explanation: An error opening a file was detected. This could be either an IGES file or an output file. Typically the file is already in use, or an invalid name has been input.

User Action: If opening an IGES file was attempted, make sure the IGES file exists (correct file name) and then reissue the open request. If opening an output file was attempted, check that a valid file name was entered.

12) 'Active file type and file option do not match'

Explanation: The active file, temporary or permanent, and the selected file option do not match.

User Action: Switch to other active file or select other file option.

13) 'Error cutting surface(s) with plane(s)'

Explanation: An error has occurred while executing the cut plane option.

User Action: Review the operation requested for correctness, and if no error is found save the I3G log file and see the system manager.

14) 'Error opening I3G log file'

Explanation: An error occurred while trying to open a session log file.

User Action: Delete or rename an existing log file, and rerun.

15) 'There are no single-segment cuts'

Explanation: The cut operation will abort if the segments generated from cutting the surfaces with the various cut planes cannot be combined into a single curve for each cut plane.

User Action: Reevaluate the surfaces being cut and modify as needed.

16) 'Surface name modified to conform to standard'

Explanation: An invalid surface name has been input.

User Action: Read the HELP file for naming conventions, and rename the surface generated with a valid name.

I3G/VIRGO SAMPLE PROGRAM EXECUTION

A sample execution of the I3G/VIRGO program (Silicon Graphics IRIS version) to open a new model file, read in data from an IGES file and repanel a surface, is presented below. An IGES file has been created and the surfaces stored in "demo.igs".

Once the user has initiated the execution of I3G/VIRGO, two informational screens (Figures 4 and 5) are displayed to provide the user with surface size limitations and special keyboard functionality that are in effect during the I3G/VIRGO session. This information is also available in the **GENERAL NOTES** section of the **HELP** file. After these screens, the user interface screen appears with its menu set to **FILE>OPEN>TEMP** as indicated by the menu trail on the right side of the screen (see Figure 9). The user is prompted for the name of the file in which a geometry is to be stored and manipulated. "demo.tmp" is provided.

With a **TEMP** file open, the user can load IGES geometry definitions into it by selecting the **IGES TO TEMP** option and providing a file name in which IGES surfaces are stored (in this case, "demo.igs"). This step is repeated if more than one file contains the required IGES surfaces. As the surfaces are read into the **TEMP** file, the names of the surfaces are displayed on the left hand side of the screen (see Figure 10). The user is now ready to manipulate the surfaces.

Since the surface list mode located in the upper left portion of the screen is defaulted to the display mode (as indicated by the asterisk), the user need only pick the surface name in the list with the mouse to display it on the screen. Once picked, the surface appears on the screen, an asterisk is placed next to the surface name (indicating that it is displayed), and the color of the surface name is changed from white to the displayed surface's color. The surface can be removed from the display at any time, by picking the surface name in the list again.

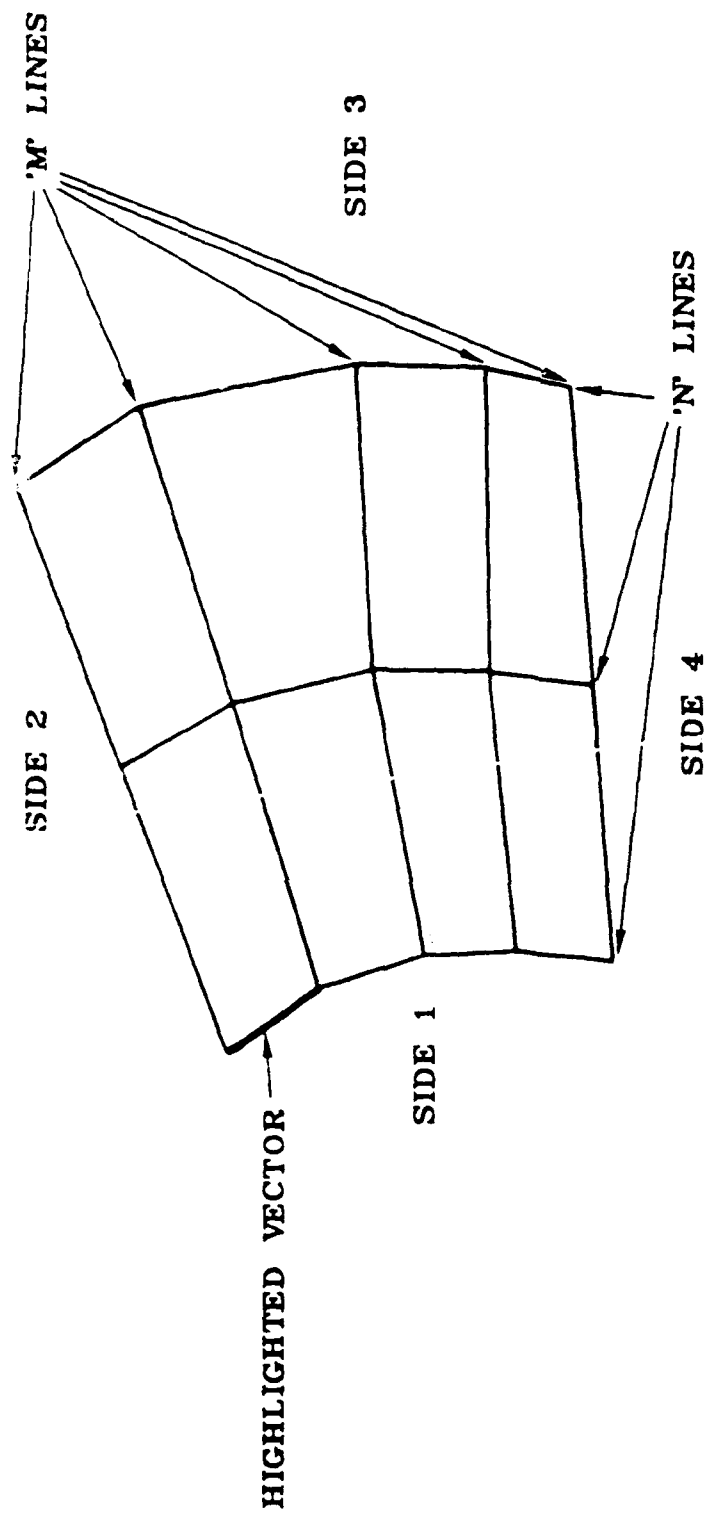
By picking **ANALYSIS ON** in the lower right corner of the display, the user can view any displayed surface's present dimensions (see Figure 11). When this mode is in operation, the file list is temporarily replaced by the chosen surface's information. All others functions are inaccessible until the user returns to the standard mode with **ANALYSIS OFF** (additional information on analysis can be found in section 3.3). To begin the manipulation, **TOP** is picked to reset the menu trail. From here the user is able to choose the options required for manipulation. To repanel the displayed surface, **GRID SURFACE** and then **PANEL** are selected in order. The menu trail in the upper right corner will be updated as each choice is made (see Figure 12). By following the prompts and the directions in the on-line **HELP** (or section 1.4.1), a repaneled surface is created with equal spacing on side 1 (indicated by the darker vector on the upper left corner of the surface), existing spacing on side 2 (the side adjacent to side 1), cosine spacing on side

3 (opposite side 1) and equal spacing on side 4. The new surface is given the name **SURFACENEW** (Figure 13).

To end the session the user now picks **END**. It is important to note that the surfaces in the **TEMP** file are now stored in **demo.tmp** and upon entering the program the next time the surfaces are accessed by reopening the **TEMP** file.

FILE	SURFACE MANIPULATION	SURFACE GENERATION	SURFACE GRID POINT GENERATION	BLOCK BUILD (VIRGO ROUTINES)	OUTPUT
<ul style="list-style-type: none"> OPEN FILE TEMP PERM IGES TO TEMP OUTPUT TRANSFER SURF TEMP TO TEMP TEMP TO PERM DELETE SURF RENAME SURF PERM TEMP 	<ul style="list-style-type: none"> BREAK SURF AT LINE SURF AT POINT SURF AT CURVE EXTRACT CURVE CURVE AT POINT EXTRACT SURFACE JOIN ABUT SURFACES ABUT CURVES BUILD CHANGE REORDER N LINES REORDER M LINES REORDER BOTH SWITCH M AND N INTERSECT TRANSFORM SCALE TRANSLATE PICK LOCATION X, Y, Z TRANS ROTATE SMOOTH CURVE SURFACE CUT PLANE YZ PLANE XZ PLANE XY PLANE ARBITRARY PLANE SURFACE OFFSET 	<ul style="list-style-type: none"> POINT MANUAL INPUT OFFSET SURFACE POINTS CURVE MANUAL INPUT OFFSET BEZIER RELATIVE BLENDED EXISTING CONIC CIRCLE ELLIPSE SPACE ELLIPSE SURFACE MANUAL INPUT SURFACE TFI COONS PATCH BODY OF REV 	<ul style="list-style-type: none"> PANEL FIXED ALONG M LINES ALONG N LINES VARIABLE ALONG M LINES ALONG N LINES ALONG BOTH <div> RELATIVE <ul style="list-style-type: none"> GEOMETRIC PROGRESSION HYPERBOLIC SIN HYPERBOLIC TAN ABSOLUTE <ul style="list-style-type: none"> GEOMETRIC PROGRESSION HYPERBOLIC SIN HYPERBOLIC TAN BLENDED <ul style="list-style-type: none"> EQUAL ARC LENGTH SINE COSINE CENTER PACKED COSINE <ul style="list-style-type: none"> CURVATURE DEPENDENT USER SPECIFIED LIKE EXISTING EQUAL X, Y OR Z LIKE OPPOSITE </div> <ul style="list-style-type: none"> MOVE POINT NODE POINT KEYBOARD ARBITRARY POINT ABUT POINT DELETE POINT DELETE LINE ADD LINE 	<ul style="list-style-type: none"> POINTS EDGES SPLINE TFI ELLIPTIC GRID PLANE LAPLACE POISSON POIS/CURV PARAMETRIC LAPLACE POISSON POIS/CURV HYPERBOLIC GRID C GRID O GRID METRIC ANALYSIS DX/D(XI) DY/D(XI) DZ/D(XI) DX/D(ETA) DY/D(ETA) DZ/D(ETA) SURFACE MAPPING CONSTANT PLANE INTERPOLATE BEZIER CREATE BEZIER CURVE FIT DEG ELEVATION 	<ul style="list-style-type: none"> IGES OTHERS
				<ul style="list-style-type: none"> TRANSLATE ROTATE SURFACE ON/OFF ANALYSIS ON/OFF RESET HELP SHELL 	

Figure 1. 135/VIRGO Capabilities

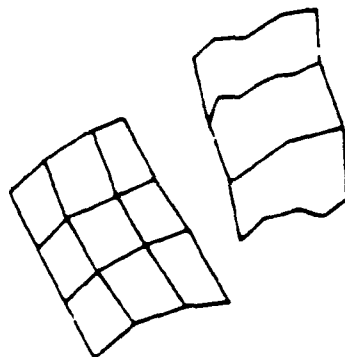


NOTE: N-LINES ARE 'CURVES' OF SURFACES

Figure 2. I3G/VIRGO Surface Nomenclature

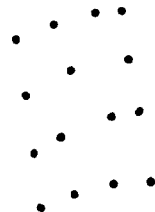
• RECTANGULAR

EVERY CURVE HAS SAME
NUMBER OF POINTS



• NON-RECTANGULAR

CURVES HAVE DIFFERENT NUMBERS OF
POINTS



• POINTS

NO ASSUMED CONNECTION

• NOTE: THE SURFACE IS A
NON-LINEAR FIT THROUGH THE
DISPLAYED POINTS

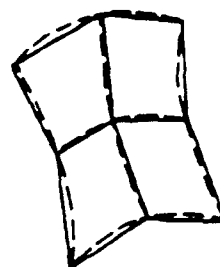


Figure 3. I3G/VIRGO Screen Display

```

      IIIIIIII  3333333  GGGGGGGG
      II  II  II  33  33  GG  GG
      II  33  GG
      II  3333  GG  GGGGG
      II  33  GG  GG
      II  II  II  33  33  GG  GG
      IIIIIIII  3333333  GGGGGGGG
VV  VV  VV  IIIIIIII  RRRRRRR  GGGGGGG  0000000
VV  VV  VV  II  RR  RR  GG  GG  00  00
VV  VV  VV  II  RR  RR  GG  GG  00  00
VV  VV  VV  II  RRRRRR  GG  GGGGG  00  00
VV  VV  VV  II  RR  RR  GG  GG  00  00
VV  VV  VV  II  RR  RR  GG  GG  00  00
VVV  IIIIIIII  RR  RR  GGGGGGG  0000000
*****
MAX # OF POINTS/SURFACE = 9000
MAX # OF POINTS/CURVE   = 300
MAX # OF CURVES/SURFACE = 300
MAX # OF SURFACES/FILE  = 300

```

ON-LINE HELP AVAILABLE, SEE LOWER RIGHT OF SCREEN.

Figure 4. Greeting Screen 1

LEFT BUTTON: controls the picking of surfaces from the display or choices or choices from the menus

MIDDLE BUTTON: controls the scaling of displayed surfaces

RIGHT BUTTON: controls the center and center/scale options on displayed surfaces

F4 KEY: toggles the display of the x,y,s coordinates of the center, the x,y,s rotations, and the scale factor of the display

L	M	R
LLL	MMM	RRR
LLL	MMM	RRR
LLL	MMM	RRR
L	M	R

ROTATIONS AND TRANSLATIONS: these functions are controlled with a dial box or keys F3 and F5 thru F12. When keys are used, the reverse direction of each function is performed when F8 or F12 is also held down. 90 degree rotations are performed when F3 is pressed while using the rotation keys.

ADDITIONAL INFORMATION ON SPECIAL FUNCTION KEYS IS AVAILABLE UNDER GENERAL_NOTES/IRIS_TERMINALS OF THE HELP FILE.

Figure 5. Greeting Screen 2

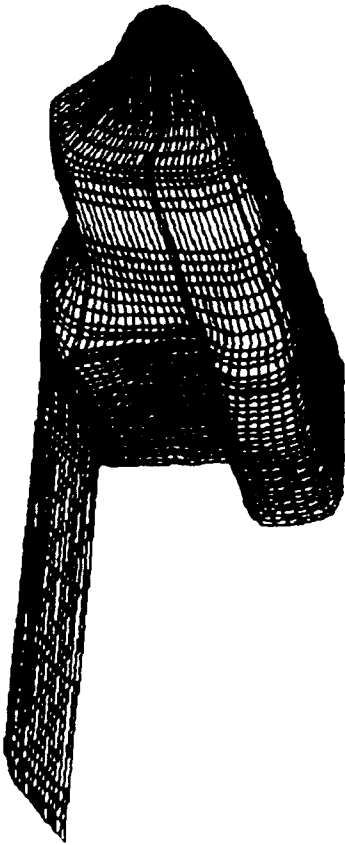
13G / VIRGO		MODEL MODE		TOP MANIPULATE MENU TRAIL	
SURFACE LIST UP DOWN AXIS		GEOMETRY DISPLAY 		MENU •BREAK JOIN CHANGE INTERSECT TRANSFORM SMOOTH CUT PLANE SURFACE OFFSET	
SCROLL		PERMANENT IMMEDIATE		OPTIONS •SURF AT LINE SURF AT POINT SURF AT CURVE EXTRACT CURVE CURVE AT POINT EXTRACT SURFACE	
Select break option, then hit RETURN KEYBOARD INPUT ERROR MESSAGES & HINTS		PROMPTS		•CENTER DISPLAY CONTRAST DISPLAY SCALE DISPLAY RESET HELP ANALYSIS ON SHELL	

Figure 6. 13G/VIRGO User Interface

DIRECTORY SECTION

TWO 80 CHARACTER RECORDS OF 10 FIELDS, (8 CHARACTERS EACH FIELD)

POINT SURFACE - ENTITY 5C01

FIELD	ENTRY	FIELD	ENTRY
1	5001	11	5001
2	Pointer to Parameter Section	12	0
3	1	13	1
4	1	14	Number of Parameter Recds
5	1	15	1=2-D Pts 2=3-D Pts
6	-	16	-
7	0	17	-
8	-	18	-
9	0	19	-
10	D000000N	20	D000000(N+1)

PROPERTY ENTITY - ENTITY 406 (NAME)

FIELD	ENTRY	FIELD	ENTRY
1	406	11	406
2	Pointer to Parameter Section	12	0
3	1	13	1
4	1	14	1
5	1	15	9
6	-	16	-
7	0	17	-
8	-	18	-
9	0	19	-
10	D000000N	20	D000000(N+1)

Figure 7-a. IGES Point Surface Definition

PARAMETER SECTION

80 COLUMN RECORD

COL 65-72	Pointer to Directory entry
COL 73	'P'
COL 74-80	Parameter section sequence number
COL 1-64	Free-field data area - delimited by ',' - ended by ,,

POINT SURFACE - ENTITY 5001

Entity type	5001
NC	Number of curves
NP	Number of points on each curve
.	.
NT	Total number of coordinate sets
Z	3rd coordinate for FORM 1 (2-D pairs)
x	Coordinate sets
y	.
z	.
y	.
z	.
.	.
0	Number of back pointers
1	Number of properties
'Dxx'	Pointer to property entity containing name

PROPERTY ENTITY - ENTITY 406 (NAME)

406	Entity number
4	
NAME	Surface name

Figure 7-b. IGES Point Surface Definition (Cont.)

5001,2, 32, 53, 53, 53, 53, 53, 53, 53,	5P	828
53, 53, 53, 53, 53, 53, 53, 53,	5P	829
53, 53, 53, 53, 53, 53, 53, 53,	5P	830
53, 53, 53, 53, 53, 53, 53, 53,1696, 0,	5P	831
142.0000 , 0.0000000E+00, 100.2000 , 142.0000 ,	5P	832
0.0000000E+00, 100.2000 , 142.0000 , 0.0000000E+00,	5P	833
100.2000 , 142.0000 , 0.0000000E+00, 100.2000 ,	5P	834
142.0000 , 0.0000000E+00, 100.2000 , 142.0000 ,	5P	835
0.0000000E+00, 100.2000 , 142.0000 , 0.0000000E+00,	5P	836
366.2000 , 11.91198 , 195.7941 , 366.2000 ,	5P	2101
7.154313 , 196.1140 , 366.2000 , 2.385963 ,	5P	2102
196.2236 , 366.2000 , -0.2912192E-21, 196.2417 ,	5P	2103
0,1, 7;	5P	2104
406,4,15HNOSE , , , ;	7P	2105
5001 2106 1 1 1 0	0D	9
5001 0 1 239 2	D	10
406 2345 1 1 1 0	0D	11
406 0 1 1 9	D	12
5001,2, 26, 12, 12, 12, 12, 12, 12, 12, 12,	9P	2106
12, 12, 12, 12, 12, 12, 12, 12,	9P	2107
12, 12, 12, 12, 12, 12, 12, 12,	9P	2108
12, 12, 312, 0,	9P	2109
366.2000 , 49.00090 , 187.5047 , 366.2000 ,	9P	2110
44.31867 , 188.3535 , 366.2000 , 39.87741 ,	9P	2111
190.1040 , 366.2000 , 35.38519 , 191.7090 ,	9P	2112
366.2000 , 30.75002 , 192.8673 , 366.2000 ,	9P	2113
26.00097 , 193.8115 , 366.2000 , 21.37605 ,	9P	2114
466.0000 , 9.896465 , 191.7612 , 466.0000 ,	9P	2341
5.938232 , 191.0255 , 466.0000 , 1.979933 ,	9P	2342
191.8784 , 466.0000 , -0.1207227E-05, 191.8773 ,	9P	2343
0,1, 11;	9P	2344
406,4,15HWINGCAP , , , ;	11P	2345
5001 2346 1 1 1 0	0D	13
5001 0 1 158 2	D	14
406 2504 1 1 1 0	0D	15
406 0 1 1 9	D	16
5001,2, 8, 26, 26, 26, 26, 26, 26, 26, 26, 0,	13P	2346
366.2000 , 49.00090 , 187.5047 , 366.6537 ,	13P	2347

Figure 8-b. IGES Example File (Cont.)

48.89798	,	189.7337	,	367.6191	,	49.12039	,	13P	2348
191.7314	,	370.5496	,	49.81583	,	195.0682	,	13P	2349
374.1895	,	50.42565	,	197.5946	,	378.2353	,	13P	2350
51.00906	,	199.4536	,	382.4832	,	51.54235	,	13P	2351
.		
432.0627	,	259.0803	,	204.4084	,	436.2645	,	13P	2500
259.1799	,	203.2315	,	446.4217	,	259.2695	,	13P	2501
201.9655	,	444.5984	,	259.3346	,	200.6224	,	13P	2502
0.1,		15;						13P	2503
406,4,15HWIN		TOP		,				15P	2504

Figure 8-c. IGES Example File (Cont.)

PERM •TEMP INPUT •DISPLAY		136 / U1ACD		TOP FILE	
<div style="border: 1px solid black; height: 600px; width: 100%;"></div>		•OPEN FILE TRANSFER SURF DELETE SURF RENOME SURF		PERM •TEMP 136S TO TEMP OUTPUT	
		•CENTER DSDLY CNTR/SCL DSDLY SCALE DSDLY RESET HELP ANALYSIS ON SHELL			
UP DOWN		Select a file option, then input a filename demo.tmp			

Figure 9. OPEN / FILE

136 / UINGO		TOP GRID SURFACE	
PERM *TOP INPUT *DISPLAY FUSE *NODE *SURFACE NEW *SURFACE *JING TOP *ZING TOP *ZING TOP		*PANEL *HIDE POINT *HIDE POINT *DELETE POINT *DELETE LINE *ADD LINE	
		*FIXED *VARIABLE	
		*CENTER DISPLAY *CENTER/SOL DISPLAY *SCALE DISPLAY *RESET *HELP *ANALYSIS ON *SHELL	
UP DOWN		Select gridding option, then input surface name	

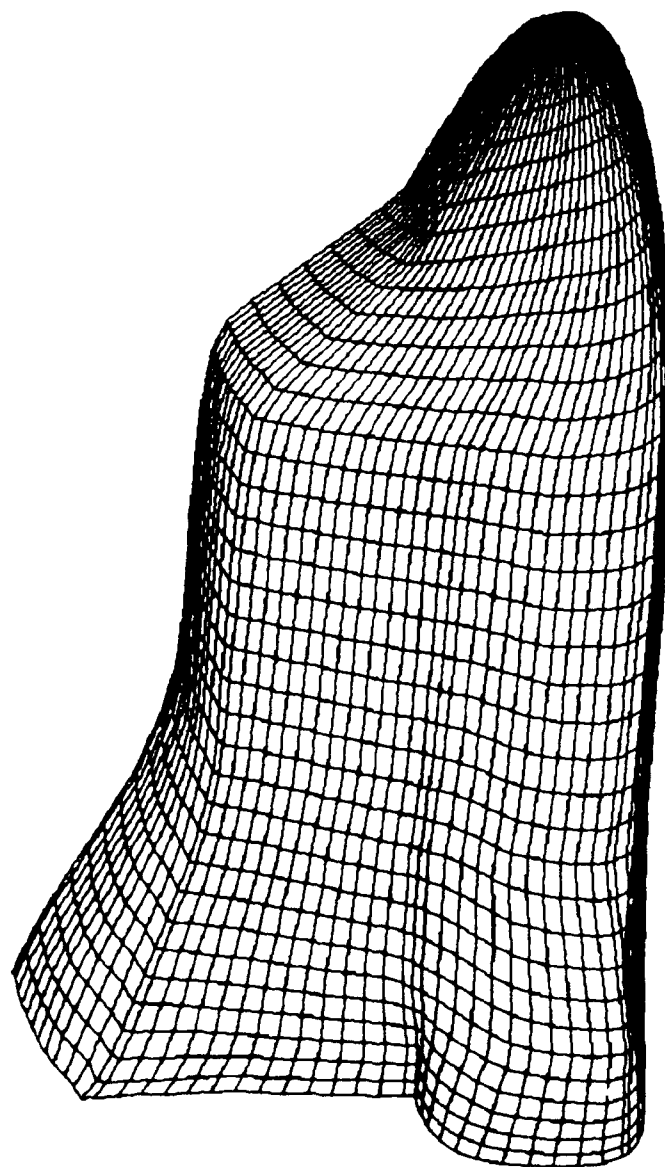


Figure 13. SURFACENEW

1 TOP

INTERACTIVE GRAPHICS for GEOMETRY GENERATION
and
VISUAL INTERACTIVE RAPID GRID GENERATION

Version 4.50 - WRDC/FIMM Mod (May 90)

- updated (Dec 90)

Version 4.75 - WRDC/FIMM Mod (May 91)

The Interactive Graphics for Geometry Generation (I3G) and the Visual Interactive Rapid Grid Generation (VIRGO) programs have been combined as (I3G/VIRGO), an interactive program that aids the engineer in generating geometric models for input to computational aerodynamics codes. I3G/VIRGO is capable of accepting input from a variety of geometry sources and can support the generation of models for many different computational codes. The geometric data within the program is stored in the system database and can be retrieved and displayed on graphics devices ranging from sophisticated systems to low-cost display-only terminals. The program's modular structure allows new modeling capabilities to be readily incorporated. The program works with 'surfaces', a surface being defined as a point, curve, or three-dimensional surface.

In the IRIS version of the code, depth-cuing and z-buffering are provided to aid the use in determining which surfaces are 'closer' to the viewer's eye. Surfaces behind the point on which you are centered will fade in intensity as you zoom in while surfaces in front of the centered point will remain at maximum intensity until they are clipped off. Z-buffering is always active in the program and displays the surfaces with the surface 'closest' to the user's eye overwriting the surfaces 'behind' it.

Seven main functions are provided : FILE, MANIPULATE, GEN(erate) SURFACE, GRID SURFACE, BLOCK BUILD, OUTPUT, and END. Each is described in detail in this HELP function.

New users are directed to read the GENERAL NOTES before reading the remaining documentation to get a "feel" for how the program runs.

1.1 TOP / FILE

This function provides all the file-related operations.

I3G/VIRGO uses two main model types, TEMP(orary) and PERM(anent). These types are, in fact, identical in format, but most operations use the TEMP model. Only TRANSFER_SURF and RENAME_SURF can operate on the PERM model.

A TEMP model must be opened for use at the start of each session. Models are opened to the program through the OPEN_FILE option under this function. Details of this procedure are available under OPEN_FILE help. New models may be opened at any time during the operation of the program. Geometry data is loaded into the program from Initial Graphics Exchange Specification (IGES) formatted files. Programs are provided to generate this format from various data sources. Opening an input (IGES format) file causes its contents to be loaded into the current TEMP model.

An OUTPUT file may be opened before code formatted data is output. Opening an output file allows the user to give a specific name to the output file. If an output file is not specifically opened, the output file will be I3G.OUT by default.

1.1.1 TOP / FILE / OPEN_FILE

The OPEN FILE function is used to open PERM(anent), TEMP(orary), IGES, and output files. The process of opening a file will cause previously opened files to be closed. This allows inputting from or outputting to several files. The file types are displayed as OPTIONS, and you will be prompted for a file name. Any file name syntax is allowed, including other directories. Finally, you will be prompted for a confirmation before completing the open operation.

1.1.2 TOP / FILE / TRANSFER_SURF

The TRANSFER SURF function is used to move surfaces between the PERM(anent) and TEMP(orary) files. The direction of transfer is chosen from the OPTION block, and the user is prompted for the surface name.

1.1.3 TOP / FILE / DELETE_SURF

The DELETE SURF function does just that - it deletes surfaces. It is non-recoverable, so be careful. Surfaces may only be deleted from temporary files. The user is prompted for the name of the surface to be deleted, and then for a confirmation before completing the delete operation.

1.1.4 TOP / FILE / RENAME_SURF

The RENAME SURF function allows surfaces in the TEMP(orary) or PERM(anent) files to be renamed. The model type is chosen in the OPTION block, and the user is prompted for the old and new surface names.

1.2 TOP / MANIPULATE

This function provides capability to create new surfaces by modifying existing ones. Most operations do not modify the original surface, but rather create a new one. Only CHANGE changes the original surface definition.

1.2.1 TOP / MANIPULATE / BREAK

The BREAK function is used to sub-divide a surface into two pieces or to extract a single curve from a surface. Surfaces may be broken along a displayed line, at an arbitrary point (along a line of constant parametric value), or at a previously computed intersection line. An individual curve may be extracted from a surface and if a surface is made up of a single curve it may be broken at up to 10 points, to form 11 new curves.

1.2.1.1 TOP / MANIPULATE / BREAK / SURF_AT_LINE

Break at a LINE prompts for the line at which to break a surface, and new surface names for the two generated surfaces. Point definition surfaces are simply broken. If any edge of a surface is indicated, it is duplicated under the first new surface name, and there is no second new surface.

1.2.1.2 TOP / MANIPULATE / BREAK / SURF_AT_POINT

Break at a POINT prompts for the surface name, the point at which to break, the break direction, and two new surface names. The break direction is chosen by indicating one displayed line with the cursor.

1.2.1.3 TOP / MANIPULATE / BREAK / SURF_AT_CURVE

Break a SURF AT CURVE allows a surface to be broken along any line lying on the surface. The curve must be available as a separate surface. Often, this curve could have been generated as the intersection of two surfaces, but this is not a requirement. The desired break curve may come from any source. Unusual results will occur if the curve does not lie in close proximity to the surface to be broken.

The break curve has the additional restriction that it must be monotonically increasing or decreasing in the direction of the requested break.

This operation prompts for the surface to be broken, the curve along which to break, and the direction of the break. The first two prompts are self-explanatory. The break direction is input by using the cursor to indicate a line on the surface to be broken that is roughly parallel to the break curve. Some curves will allow a break in either direction, but some will not, due to the requirement for the line to be monotonic.

1.2.1.4 TOP / MANIPULATE / BREAK / EXTRACT_CURVE

EXTRACT a CURVE prompts for the curve which is to be extracted from a surface and a new surface name for the extracted curve. Any curve, including edge curves, that exists as part of a surface may be extracted and isolated with this function.

1.2.1.5 TOP / MANIPULATE / BREAK / CURVE AT POINT

Break a CURVE AT A POINT requires the user to select from the options list whether to break at a NODE POINT or an ARBITRARY POINT. The user then selects the point at which to break a curve. If the point selected is a node point on a surface which consists of more than a single curve, the user will also be prompted for which curve to break of the two curves which intersect at that node point. The user continues to be prompted for additional points to break the curve at, up to a limit of 10 break points. Each of these points must lie on the original curve selected. The option of NODE POINT or ARBITRARY POINT may be changed for each consecutive point picked. The user types END to signify that the last break point has been entered, and the program prompts for a name for the new set of curves. The name for each of the up to 11 new curves consists of this "set" name appended by the numbers 1 up to 11. When the ARBITRARY POINT option is used, care should be taken when indicating (with the cursor) the break point since the program is working with the "curve" through the points and not the "straight line point-to-point" display of the curve. Note that a curve (or line) that is part of an existing surface can be broken with this function.

1.2.1.6 TOP / MANIPULATE / BREAK / EXTRACT SURFACE

EXTRACT a SURFACE allows a region of an existing surface to be extracted and made into a new surface. The user will be prompted for the name of the surface containing the desired region, and then will select the new region by inputting two diagonally opposite corners of the region. To input the corner points, the user has the option of typing in the two indices (n,m) or picking the point from the display. (see below) The '+' symbols indicates the white "side 1" vector that appears on the surface.

The diagram illustrates the geometry of a surface A and its interaction with a region B. It is divided into three main sections:

- Left Section (Surface A):** A grid of points labeled 1 to 6. The points are arranged in a 6x6 grid. The top row is labeled 1 to 6 from left to right. The left column is labeled 1 to 6 from top to bottom. The grid is bounded by dashed lines.
- Middle Section (Surface A with region B highlighted):** A grid of points labeled 1 to 6. The points are arranged in a 6x6 grid. The top row is labeled 1 to 6 from left to right. The left column is labeled 1 to 6 from top to bottom. The grid is bounded by dashed lines. A central region is highlighted with asterisks.
- Right Section (Surface B1 and Surface B2):** Two separate grids of points. The top grid is labeled 'surface B1' and the bottom grid is labeled 'surface B2'. Both grids are bounded by dashed lines.

Surface B1 can be extracted from surface A by inputting or picking pt (4,2) then pt (1,4) and surface B2 is extracted with pts (1,2) and (4,4). B1 and B2 are equivalent surfaces and differ only by storage sequence.

1.2. TOP / MANIPULATE / JOIN

The JOIN function is used to combine two or more surfaces into one new surface. Care should be taken with the order in which the surfaces to be joined are input. Surfaces will be combined based on this order.

1.2.2.1 TOP / MANIPULATE / JOIN / ABUT_SURFACES

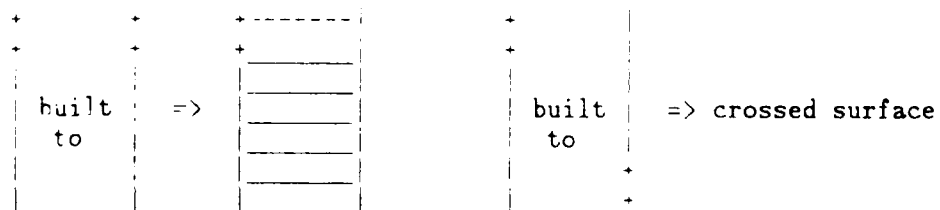
The ABUT SURFACES option allows the joining of two surfaces that touch to generate a new surface. The user is prompted for the two surfaces and the abutting edge. The coding contains the logic to properly order the points on the abutting surfaces, and the new surface will be ordered like the first surface input. Finally, the user is prompted for the name of the new surface.

1.2.2.2 TOP / MANIPULATE / JOIN / ABUT_CURVES

The ABUT CURVES option allows the joining of curves (two or more) to generate a new curve. The curves may be touching or not. If two curves that have been sequentially input touch, the first point on the second curve will be eliminated. If none of the curves are touching, all points will be kept. The user is first prompted for the first curve and then to indicate the end of this curve at which the new curve that is being generated by the join operation is to start. The user is then prompted for a succession of surfaces, up to a maximum of 40 surfaces, that are terminated by inputting an END. Finally, the user is prompted for the name of the new curve. The coding contains the logic to properly order the points on the curves being joined.

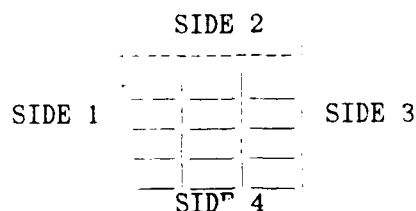
1.2.2.3 TOP / MANIPULATE / JOIN / BUILD

The BUILD option allows the joining of non-touching surfaces to generate a new surface. The user is prompted for a succession of surfaces up to a maximum of 40 surfaces. The coding does not contain the logic to properly order the points on the surfaces, therefore, if the surface from a BUILD operation is "twisted," the ordering of the N and M lines on the surfaces needs to be checked (see below). Any reordering of the lines that is necessary can be done with the CHANGE function. The '+' symbol represents the white "side 1" vector that shows how the line is stored in memory.



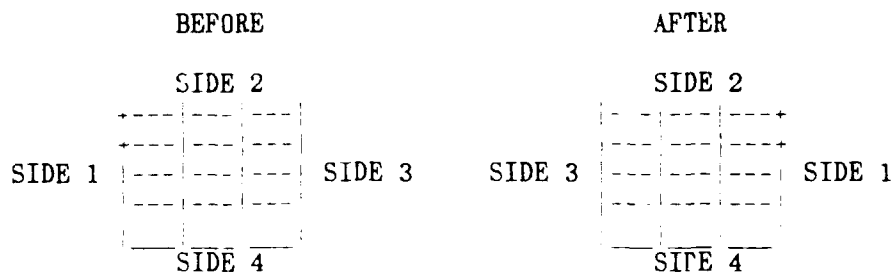
1.2.3 TOP / MANIPULATE / CHANGE

The CHANGE function provides the ability to change the order of the N and M lines and also to switch N and M lines. M lines connect sides 1 and 3, N lines connect sides 2 and 4. The '+' symbols represents the white "side 1" vector that appears on the surface.



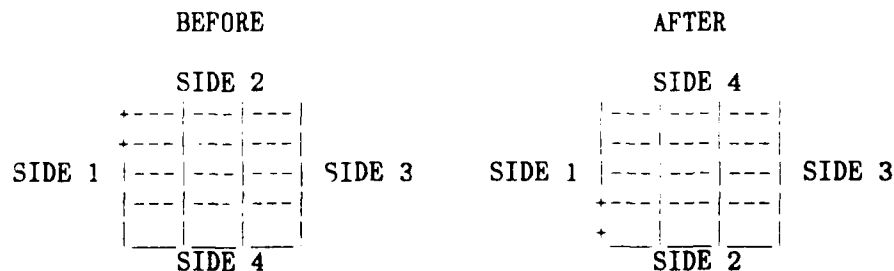
1.2.3.1 TOP / MANIPULATE / CHANGE / REORDR_N_LINES

The order of N lines is reversed by this function. The surface is the only input required. The surface name remains the same. On a surface this has the effect of making side 1 into side 3, and making side 3 into side 1. The '+' symbols represents the white "side 1" vectors that appear on the surfaces.



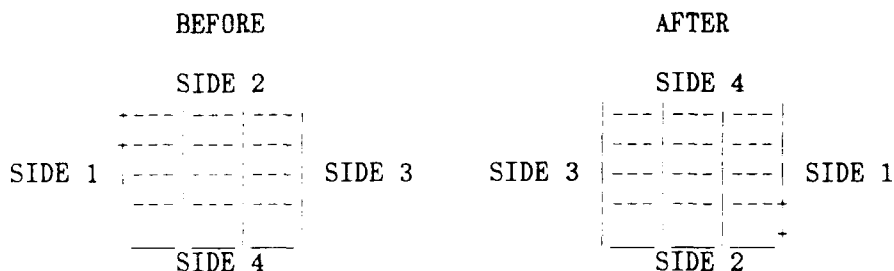
1.2.3.2 TOP / MANIPULATE / CHANGE / REORDR_M_LINES

The order of M lines is reversed by this function. The surface is the only input required. The surface name remains the same. On a surface this has the effect of making side 2 into side 4, and making side 4 into side 2. The '+' symbols represents the white "side 1" vectors that appear on the surfaces.



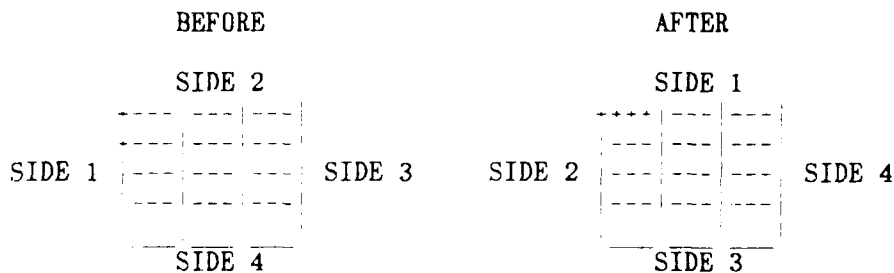
1.2.3.3 TOP / MANIPULATE / CHANGE / REORDER BOTH

The order of both N and M lines is reversed by this function. The surface is the only input required. The surface name remains the same. On a surface this has the effect of making side 3 into side 1 and making side 4 into side 2. The '+' symbols represents the white "side 1" vectors that appear on the surfaces.



1.2.3.4 TOP / MANIPULATE / CHANGE / SWITCH N AND M

The N and M lines are switched by this function. The new N lines will run in the old M line direction and vice versa. The surface is the only input required. The surface name remains the same. On a surface this has the effect of making side 2 into side 1 and making side 3 into side 4. The '+' symbols represents the white "side 1" vectors that appear on the surfaces.



1.2.4 TOP / MANIPULATE / INTERSECT

The INTERSECT function creates a new 'surface' which is the intersection curve of two surfaces. The user is first prompted to indicate (using the cursor) the lines on one surface that will intersect another surface. Next, the user will input the surface that is being intersected by the intersecting lines. Surface names may be input by picking the surface with the cursor, selecting the name from the list with the cursor, or by inputting the name using the keyboard if a Tektronix 4115 terminal is being utilized. Note that if the surface name is being input either through the surface list or the keyboard, the INPUT mode under INPUT/DISPLAY must be invoked. Once the

user has input the name of the surface to be intersected, the user is prompted for a spacing option and a number of intersection points to be applied to the calculated intersection curve. If the user selects the EXISTING spacing option, an additional input of the line or curve to match the spacing on is required. Finally, the user is prompted to input the new name for the intersection curve. The intersection function can perform partial intersections, however intersection curves that are discontinuous can be a problem.

The computed intersection curve can be used to break a surface using the BREAK SURF AT CURVE function.

1.2.5 TOP / MANIPULATE / TRANSFORM

The TRANSFORM function provides the ability to scale, translate, and rotate any surface.

1.2.5.1 TOP / MANIPULATE / TRANSFORM / SCALE

A surface can be scaled using this function. The X, Y, and Z coordinates are multiplied by X, Y, and Z scaling factors input by the user. The user has the option of retaining the old surface name or inputting a new one.

1.2.5.2 TOP / MANIPULATE / TRANSFORM / TRANSLATE

A surface can be translated using this function. Two different methods are available to the user: PICK LOCATION and X,Y,Z TRANS. Under each option the user has the choice of retaining the old surface name or inputting a new one.

1.2.5.2.1 TOP / MANIPULATE / TRANSFORM / TRANSLATE / PICK_LOCATION

A surface can be translated to a specific location, which can be picked from the displayed or input with the keyboard. A reference point on the translating surface and the final location point are chosen by the user, and the surface is translated such that the two points are coincident. When picking the reference and final points, NEAREST NODE or ARBITRARY PT allow the user the flexibility to pick the nodes or points in between the nodes as input. Positive and negative values are permitted.

1.2.5.2.2 TOP / MANIPULATE / TRANSFORM / TRANSLATE / X,Y,Z_TRANS

A surface can be translated using this option. X, Y, and Z translation distances are input by the user and the surface is shifted the input values. Positive and negative values are permitted.

1.2.5.3 TOP / MANIPULATE / TRANSFORM / ROTATE

A surface can be rotated using this function. The surface is rotated

about a rotation point through a rotation angle, both of which are input by the user. If the user selects the rotate ABOUT LINE option, the user is prompted for either a line or the endpoints of a line. The user has the option of retaining the old surface name or inputting a new one.

1.2.6 TOP / MANIPULATE / SMOOTH

The SMOOTH function allows the user to smooth the points defining either a curve or a surface. To smooth a curve, the user selects the CURVE option and picks any displayed curve. This may be an independent curve or an N-line or M-line on a surface. End points on the smoothed curve are always the same as on the original curve. Entire surfaces may be smoothed by selecting the SURFACE option. The user picks a displayed rectangular surface. Smoothing of a surface is accomplished by individually smoothing the defining curves that make up the surface. Edge curves are not modified in this process. Edge curves can be smoothed only by choosing the CURVE option and specifically picking the edge curve to be smoothed. Finally, the user is prompted for a new name for the smoothed surface or curve.

1.2.7 TOP / MANIPULATE / CUT_PLANE

The CUT PLANE function allows the user to intersect surfaces with planes and generate the intersection curves. When this function is selected the user is given the option of cutting all of the surfaces that are currently being displayed or a list of surfaces that is specified by the user and is ended by the user keying in an END. The user then selects one of four possible plane options. These are YZ Plane, XZ Plane, XY Plane or Arbitrary Plane. If the user chooses either the YZ, XZ, or XY plane option, the user inputs either the X, Y, or Z value, respectively, of the cutting plane. If the arbitrary plane option is chosen, the user is required to input three points that define the cutting plane. When the user is inputting the numerical values to define the plane location, the user can either use the keyboard or use the cursor and pick points off of the screen. After the user has specified the cutting planes the user is then prompted for a segment ordering tolerance. Segments generated by cutting several surfaces, that are within the tolerance, will be connected together. If multiple segments are produced, or multiple cutting plane curves are generated the program will treat them as the individual defining curves of a single surface. In general, the intersection curve consists of the points where the cutting plane intersects N- or M-lines, as well as one additional point on the surface in between consecutive N- or M-lines. Within one execution of this function the user can specify up to 100 cutting planes and they can be in any combination of the various options. Surfaces that are being cut must be multi-curved, rectangular, point definition surfaces. Also, if more than one surface is being cut, the surface points should be ordered such that they are consistent from surface to surface.

1.2.8 TOP / MANIPULATE / SURFACE_OFFSET

The SURFACE OFFSET function allows the user to generate a new surface by offsetting an existing surface. The user is prompted for the existing surface from which to offset and then an offset distance. The new surface is generated by offsetting the points from the existing surface in a local normal direction (the normal direction is defined in GENERAL NOTES). Care must be taken when offsetting highly curved surfaces.

1.3 TOP / GEN_SURFACE

The GEN(erate) SURFACE function contains a limited set of surface generation capabilities. This function can be used where an "area fill" capability is needed - where no surface definition exists between other defined surfaces. This function is also useful when defining "surfaces" that are in the flowfield and do not lie on the configuration geometry. These surfaces may be defined by existing curves or by new curves such as circles, ellipses or space curves that can be generated using this function. Finally, surfaces consisting of points with no specified connectivity can be created. The GEN SURFACE capabilities are divided into three categories : generate POINT, generate CURVE, and generate SURFACE.

1.3.1 TOP / GEN_SURFACE / POINT

The generate POINT menu option allows the user to create point definition surfaces, where the surface created consists of an array of random points, with no connectivity specified. This surface may be created in one of three ways. These are MANUAL INPUT, OFFSET, and SURFACE POINTS.

1.3.1.1 TOP / GEN_SURFACE / POINT / MANUAL_INPUT

The MANUAL INPUT menu item under the generate POINT menu allows the user to simply enter, one by one, the coordinates of the points that are to comprise the surface. The user first enters a name for the new surface that is being generated, and then begins entering points one at a time. When entering points, the user may either enter the X, Y, and Z coordinates at the keyboard or pick an existing node point from the display area. If the display area pick is not on a node point, the program will use the coordinates of the node point closest to the point picked.

1.3.1.2 TOP / GEN_SURFACE / POINT / OFFSET

The OFFSET menu item under the generate POINT menu allows the user to generate a surface consisting of a single point by offsetting by specified X, Y, and Z distances from any other point. The user is first prompted for the name for the new surface (point) being generated. Next, the user is prompted for the point from which to

offset. This may be entered at the keyboard or picked from the display area. Finally, the user must key in the X, Y, and Z distance by which to offset from this point.

1.3.1.3 TOP / GEN_SURFACE / POINT / SURFACE_POINTS

The SURFACE POINTS function is available to generate a representation of a surface which contains only the defining points, i.e., there is no assumed connectivity of the points and the lines connecting the points are removed. This function is useful if new curves or surfaces need to be generated that can be derived from the defining points of existing surfaces. The user is prompted for the new surface (points only) name, and then an existing surface name from which these points are to be obtained.

1.3.2 TOP / GEN_SURFACE / CURVE

The generate CURVE menu option allows the user to create curves, where the surface created consists of a single string of connected points. A curve can be created in four ways. These are MANUAL INPUT, OFFSET, BEZIER, and CONIC.

1.3.2.1 TOP / GEN_SURFACE / CURVE / MANUAL_INPUT

The MANUAL INPUT menu item under the generate CURVE menu allows the user to simply enter, one by one, the coordinates of the points that are to comprise the curve. The user first enters a name for the new curve that is being generated, and then begins entering points one at a time. When entering points, the user may either enter the X, Y, and Z coordinates at the keyboard or pick an existing node point from the display area. If the display area pick is not on a node point, the program will use the coordinates of the node point closest to the point picked. The points on the curve will be ordered in the same order in which they are input.

1.3.2.2 TOP / GEN_SURFACE / CURVE / OFFSET

The OFFSET menu item under the generate CURVE menu allows the user to generate a new curve by offsetting by specified X, Y, and Z distances from any other curve. The curve offset from may be an individual curve or any grid line within a surface definition. The user is first prompted for the name for the new curve being generated. Next, the user is prompted for the curve from which to offset. This must be picked from the display area even if it is an individual curve rather than an interior grid line in a surface. (That is, the name of the curve being offset from cannot be entered from the surface list while in INPUT mode - it MUST be picked from the display area.) Finally, the user must key in the X, Y, and Z distances by which to offset from this curve.

1.3.2.3 TOP / GEN_SURFACE / CURVE / BEZIER

The BEZIER menu item under generate CURVE is used to generate a Bezier curve between any two endpoints, using an additional two points to define the tangents at the endpoints as well as the shape in the interior of the curve. The user is first prompted for a new surface name. Next, the user is prompted for the first endpoint on the curve, the second endpoint on the curve, the control point at the first endpoint, and the control point at the second endpoint, in that order. The Bezier curve computed approaches the first endpoint from the direction of the first control point, becoming exactly tangent to the line between these two points at the endpoint (and likewise for the second endpoint and control point). The interior shape of the curve depends on the distance along the tangent line at which the control point is placed relative to the endpoint. In general, the entire curve will lie within the quadrilateral defined by the four points. Finally, the user is prompted for a spacing option and number of points to use to create the curve. The allowed spacing options are RELATIVE, BLENDED, and EXISTING.

1.3.2.3.1 TOP / GEN_SURFACE / CURVE / BEZIER / RELATIVE

The point spacing using the RELATIVE option is specified as a fraction of total arc length of the curve. The user must first enter the number of points to be spaced along the curve. Next, the user must choose from the option list one of the three types of spacings that are available under this option. These are GEOMETRIC PROG(ression), HYPERBOLIC SIN(e), and HYPERBOLIC TAN(gent). After choosing the desired spacing, and hitting a RETURN, the user must select from the option list the end at which he is going to specify the spacing. The choices are FIRST END, LAST END, and BOTH ENDS if the user has selected GEOMETRIC PROG or HYPERBOLIC TAN, and FIRST END, LAST END, and INTERIOR POINT if the user has selected HYPERBOLIC SIN. After selecting the end option, the user must enter at the keyboard the value for the relative spacing to be used at this end. If BOTH ENDS has been selected, the user may enter both spacings on one line (first end first, last end second), or may enter just the first end spacing, and be re-prompted for the last end spacing. If INTERIOR POINT has been selected, the user will receive another prompt to input the relative location of the interior point at which to use the relative spacing just entered. This parametric value may be typed at the keyboard, or, alternatively, the user may use the cursor to pick the point along the side of the displayed surface at which the specified spacing is to be used.

1.3.2.3.2 TOP / GEN_SURFACE / CURVE / BEZIER / BLENDED

Point spacing with this option is interpolated from a set of equal arc, sine, and cosine spacing functions. First, the user will be prompted for the number of points desired. Then the user will be prompted to input a value between -3 and +4 for the blending. This value specifies a spacing interpolated from a set defined by :

Blending Value	Spacing
-3	Equal Arc Length
-2	Sine Spacing Packed at End
-1	Cosine
0	Equal Arc Length
1	Cosine
2	Sine Spacing Packed at Start
3	Equal Arc Length
4	Cosine Packed at Center

Any real value between -3 and +4 may be input, with the final spacing being interpolated from the above set.

1.3.2.3.3 TOP / GEN_SURFACE / CURVE / BEZIER / EXISTING

This option allows the user to specify the spacing to be the same (parametrically) as an existing curve. The user is prompted to use the cursor to choose the curve whose spacing is to be matched. The number of points on the chosen curve is then augmented (if specified by the user) to the number of panels requested by the user.

1.3.2.4 TOP / GEN_SURFACE / CURVE / CONIC

The CONIC menu item under generate CURVE allows the user to generate curves which are described by conic sections. The choices under this menu item are CIRCLE, ELLIPSE, and SPACE ELLIPSE.

1.3.2.4.1 TOP / GEN_SURFACE / CURVE / CONIC / CIRCLE

The CIRCLE option allows the user to define a circle (or any part of one) in any one of the three coordinate planes (YZ, XZ or XY). The user must first enter a name for the circle to be generated. Next, the user selects the plane option and then inputs the circle center point (either input from the keyboard or selected on a displayed surface). The user is then prompted for a radius (again either input from the keyboard or selected). Finally, the user is prompted for starting and ending angles for the circular arc (this provides the capability for defining parts of circles), and for the number of points to use to define the circular arc. The points defining the circular arc are equally spaced around the arc.

1.3.2.4.2 TOP / GEN_SURFACE / CURVE / CONIC / ELLIPSE

The ELLIPSE option allows the user to define an ellipse (or any part of one) in any one of the three coordinate planes (YZ, XZ or XY). The user must first enter a name for the ellipse to be generated. Next, the user selects the plane option and then inputs the ellipse center point (either input from the keyboard or selected on a displayed surface). The user is then prompted for an A axis and a B axis (again these are input either from the keyboard or selected). The A and B axes are defined in the same order as the coordinate plane in which the

user is working (that is, the plane selected is the AB plane). Finally, the user is prompted for starting and ending angles for the elliptical arc (this provides the capability for defining parts of ellipses), and for the number of points to use to define the elliptical arc. The points defining the elliptical arc are automatically clustered in areas of highest curvature.

1.3.2.4.3 TOP / GEN_SURFACE / CURVE / CONIC / SPACE_ELLIPSE

The SPACE ELLIPSE option allows the user to define a part of an ellipse in any orientation in space. After a name for the new surface has been input, the user selects the ellipse center point (either input from the keyboard or selected on a displayed surface). The user is then prompted for a point to define one of the axes (again this is input either from the keyboard or selected). Finally, the user is prompted for another point on the ellipse and for the number of points to use to define the arc. The section of the ellipse that will be generated will extend from the axis point to the second point on the ellipse. The three input points must not be colinear.

1.3.3 TOP / GEN_SURFACE / SURFACE

The generate SURFACE menu option allows the user to create new surfaces, where the surface created consists of lines in two parametric directions. A surface can be created in three ways. These are MANUAL INPUT, SURFACE TFI, and COONS PATCH.

1.3.3.1 TOP / GEN_SURFACE / SURFACE / MANUAL_INPUT

The MANUAL INPUT option allows the user to define a point definition surface by entering the points describing the surface one at a time. The user first enters a name for the new surface being generated. Next, the user indicates via the options list whether the point about to be input is the NEXT POINT on a curve currently being described, or the beginning of the NEXT CURVE on the surface. The user then inputs the point, either from the keyboard or from the display area. After entering the first point after a NEXT CURVE option, the program automatically shifts the option to NEXT POINT, so the user need only re-select from the options list when moving ahead to define the next curve on a surface (by picking NEXT CURVE). After the last point on the last curve has been input, the user should type END at the keyboard to complete the surface definition.

1.3.3.2 TOP / GEN_SURFACE / SURFACE / SURFACE_TFI

A SURFACE TransFinite Interpolation function is available to generate a 3-D surface from four space curves. The user is prompted to indicate the desired four curves and a name for the new surface to be generated. The user must indicate these curves by using the cursor and picking in the display area. Note that the curves should be input in order (side 1, side 2, side 3, side 4) and the resulting surface will have its

sides similarly ordered. The ordering of the points on side 1 will be preserved, although the ordering on sides 2, 3, and 4 will be modified as needed and is not critical. The curves do not necessarily need to be isolated curves or even at the edge of existing surfaces. There must be the same numbers of points along opposite edges. The surface that is generated as a result of the transfinite interpolation function will preserve the defining edge points.

1.3.3.3 TOP / GEN_SURFACE / SURFACE / COONS_PATCH

A COONS PATCH function is available to generate a 3-D surface from four space curves. The user is prompted to indicate the desired four curves and a name for the new surface to be generated. The user must indicate these curves by using the cursor and picking in the display area. The curves may be input in any order and do not necessarily need to be isolated curves or even at the edge of existing surfaces. The surface that is generated as a result of the Coons patch operation will have eleven equally spaced node points along each side.

1.3.3.4 TOP / GEN_SURFACE / SURFACE / BODY_OF_REV

A BODY OF REVolution may be generated from any displayed curve. The curve may be isolated or a particular grid line in a surface. The body of revolution is generated by selecting the curve to rotate, and choosing from the options list whether to rotate parallel to the X axis, Y axis, or Z axis, or about some arbitrary line segment. If rotating parallel to one of the axes, the user is prompted for a point to define the rotation axis. If rotating about an arbitrary line, the user must pick a displayed line segment or type in the coordinates of the two endpoints of the line. Next, the user enters the total angle to rotate the curve through and the number of curves to generate on the final surface. Finally, a new name is input for the body of revolution.

1.4 TOP / GRID_SURFACE

The GRID SURFACE function is the heart of I3G and contains the capability to generate a grid of surface points. This function includes the following sub-functions: PANEL, MOVE POINT, ABUT POINT, DELETE POINT, DELETE LINE, and ADD LINE.

1.4.1 TOP / GRID_SURFACE / PANEL

This function controls the generation of a grid of surface points on any surface within the TEMP model. The user must first select from the options list whether to use FIXED or VARIABLE spacing on opposite sides of the surface, and then input the surface name to be respaced. If the surface chosen is not a single curve, the user must next select from the options list whether to respace ALONG N LINES, ALONG M LINES, or ALONG BOTH. If the surface chosen was non-rectangular, the ALONG M LINES option does not appear since M lines do not exist on non-

rectangular surfaces. If the FIXED option was selected, the user is prompted for the spacing option to use on side 1 for ALONG N LINES, or on side 2 for ALONG M LINES, or on both sides 1 and 2 for ALONG BOTH. The program will then use these spacings on opposite sides as well (if the surface is not a single curve). If the VARIABLE option was selected, the program prompts for spacing options on all four sides if the respacing is ALONG BOTH, or for the two opposite sides - 1 and 3 if ALONG N LINES or 2 and 4 if ALONG M LINES. The number of panels on each side is input for sides 1 and 2 only.

Remember that side 1 is identified by a high-lighted vector.

Eight spacing options are provided. These are RELATIVE, ABSOLUTE, BLENDED, CURVATURE, USER SPECIFIED, EXISTING, EQUAL_X_Y_OR_Z, and LIKE OPPOSITE, and each is described in detail below.

1.4.1.1 TOP / GRID_SURFACE / PANEL / RELATIVE

The point spacing using the RELATIVE option is specified as a fraction of total arc length along a side of the surface being paneled. The user must first enter the number of panels to be spaced along the side. Next, the user must choose from the option list one of the three types of spacings that are available under this option. These are GEOMETRIC PROG(ression), HYPERBOLIC SIN(e), and HYPERBOLIC TAN(gent). After choosing the desired spacing, and hitting a RETURN, the user must select from the option list the end at which he is going to specify the spacing. The choices are FIRST END, LAST END, and BOTH ENDS if the user has selected GEOMETRIC PROG or HYPERBOLIC TAN, and FIRST END, LAST END, and INTERIOR POINT if the user has selected HYPERBOLIC SIN. After selecting the end option, the user must enter at the keyboard the value for the relative spacing to be used at this end. If BOTH ENDS has been selected, the user may enter both spacings on one line (first end first, last end second), or may enter just the first end spacing, and be re-prompted for the last end spacing. If INTERIOR POINT has been selected, the user will receive another prompt to input the relative location of the interior point at which to use the relative spacing just entered. This parametric value may be typed at the keyboard, or, alternatively, the user may use the cursor to pick the point along the side of the displayed surface at which the specified spacing is to be used.

1.4.1.2 TOP / GRID_SURFACE / PANEL / ABSOLUTE

The point spacing using the ABSOLUTE option is specified as a physical arc length to be used along a side of the surface being paneled. The user must first enter the number of panels to be spaced along the side. Next, the user must choose from the option list one of the three types of spacings that are available under this option. These are GEOMETRIC PROG(ression), HYPERBOLIC SIN(e), and HYPERBOLIC TAN(gent). After choosing the desired spacing, and hitting a RETURN, the user must select from the option list the end at which the spacing will be

specified. The choices are FIRST END, LAST END, and BOTH ENDS if the user has selected GEOMETRIC PROG or HYPERBOLIC TAN, and FIRST END, LAST END, and INTERIOR POINT if the user has selected HYPERBOLIC SIN. After selecting the end option, the user must enter the value for the absolute spacing to be used at this end. This value may be typed at the keyboard, or any displayed line segment may be picked with the cursor, and the absolute length of that segment will be used as the value of the absolute spacing at the end selected. If BOTH ENDS has been selected, the user may enter both spacings at the keyboard on one line (first end first, last end second), or may enter just the first end spacing either at the keyboard or with a cursor pick, and be re-prompted for the last end spacing which may then be typed at the keyboard or picked with the cursor. If INTERIOR POINT has been selected, the user will receive another prompt to input the relative location of the interior point at which to use the absolute spacing just entered. This parametric value may be typed at the keyboard, or, alternatively, the user may use the cursor to pick the point along the side of the displayed surface at which the specified spacing is to be used.

1.4.1.3 TOP / GRID_SURFACE / PANEL / BLENDED

Panel spacing with this option is interpolated from a set of equal arc, sine, and cosine spacing functions. First, the user will be prompted for the number of panels desired. Then the user will be prompted to input a value between -3 and +4 for the blending. This value specifies a spacing interpolated from a set defined by :

Blending Value	Spacing
-3	Equal Arc Length
-2	Sine Spacing Packed at End
-1	Cosine
0	Equal Arc Length
1	Cosine
2	Sine Spacing Packed at Start
3	Equal Arc Length
4	Cosine Packed at Center

Any real value between -3 and +4 may be input, with the final spacing being interpolated from the above set.

1.4.1.4 TOP / GRID_SURFACE / PANEL / CURVATURE

This option allows the user to obtain a spacing that is based on the curvature along the specified edge. The spacing is determined using the 1st and 2nd derivatives of the arc length with respect to each of the three coordinates.

1.4.1.5 TOP / GRID_SURFACE / PANEL / USER_SPECIFIED

User-specified panel spacing allows the user to request exactly the

panel edge spacing that he desires. This is done by picking (using the screen cursor) a number of points equal to the number of panels plus 1. The screen prompt shows both the total number of points expected, and the current point number. The chosen points will typically come from the side being paneled, but this is not a necessity. The array of input points is used to compute an array of non-dimensional positions along the line, and it is these percentages that are used to determine the final panel edge locations.

1.4.1.6 TOP / GRID_SURFACE / PANEL / EXISTING

This option allows the user to specify the spacing to be the same (parametrically) as an existing curve. The user is prompted to use the cursor to choose the curve whose spacing is to be matched. The number of points on the chosen curve is then augmented (if specified by the user) to the number of panels requested by the user.

1.4.1.7 TOP / GRID_SURFACE / PANEL / EQUAL_X_Y_OR_Z

This option allows the user to generate spacing lines that are along either the X, Y or Z directions. The user is first prompted for the number of points to space along the edge. Next, the user is prompted for the spacing line direction (X, Y or Z) and must then choose from the option list whether to use variable incrementing (VARIABLE INC) or equal incrementing (EQUAL INCREMENT) in that direction. If the VARIABLE INC option is selected, the user is prompted for the specific X, Y or Z values at which to place each of the points along the edge. The user can either input the X, Y or Z values from the keyboard or use the cursor to choose the desired values. If the EQUAL INCREMENT option is selected, the X, Y, or Z difference between the endpoints of the edge will be divided equally such that the delta-X, Y or Z is constant from point to point along the edge.

1.4.1.8 TOP / GRID_SURFACE / PANEL / LIKE_OPPOSITE

This option is self-explanatory and simplifies the prompt sequence in the VARIABLE paneling function.

1.4.2 TOP / GRID_SURFACE / MOVE_POINT

The MOVE POINT function provides the capability to interactively move any point. The new coordinates may be input via the keyboard, by indicating another displayed point, or by indicating a point on any surface.

1.4.2.1 TOP / GRID_SURFACE / MOVE_POINT / NODE_POINT

The NODE POINT function allows moving any node point on a point-definition surface to a point on a displayed line, or to another node point location. The procedure is to choose the surface, and then to pick the point to be moved. The user then picks a location on a displayed line or point to where the first point will be moved.

1.4.2.2 TOP / GRID_SURFACE / MOVE_POINT / KEYBOARD

The KEYBOARD function allows changing the coordinates of any node point by manual entry from the keyboard. The procedure is to identify the surface, and then to pick the point to be moved. The user then is prompted to key in new coordinates (X,Y,Z) for the point.

1.4.2.3 TOP / GRID_SURFACE / MOVE_POINT / ARBITRARY_PNT

The ARBITRARY PNT function allows moving any node point on a point-definition surface to a location anywhere on the same surface or on another surface. The four-step procedure is as follows:

- 1) Choose the surface where the desired point to be moved is located.
- 2) Pick the node point to be moved.
- 3) Choose the surface where the point will be moved to.
- 4) Use the cursor to indicate a location anywhere on the desired surface. The node point picked (Step 2) will be moved to that location.

1.4.3 TOP / GRID_SURFACE / ABUT_POINT

This function provides the capability to automatically abut two surfaces. The "new" points are either the points from surface 1, surface 2 or the average as specified by the user. The user is prompted for surface 1, surface 2, a point on the abutting edge, and an abutment tolerance. If the distance between a point on the abutting edge of surface 1 and a point on the abutting edge of surface 2 is less than the abutment tolerance, then the two points will be abutted (made to match). The tolerance must be a positive number. If the user desires that all of the points on the abutting edges be abutted (and there are the same numbers of points of the abutting edges), the user should not input an abutment tolerance and just input a RETURN. New surface names are input for those surfaces that will be changed.

1.4.4 TOP / GRID_SURFACE / DELETE_POINT

This function provides the capability to delete points from a surface grid. The user is first prompted to input the surface to delete points from. If the surface is anything other than a non-connected point (type 106) surface, the user is next presented with a list of options for how he will identify the points to be deleted. If the surface is rectangular, the options are SPECIFIC PTS, RANGE OF M, and RANGE OF N. For non-rectangular surfaces, the RANGE OF N option is omitted since there is no implied connection in that direction. For type 106 surfaces, no options are presented since there are no N or M lines, and the SPECIFIC PTS option is used with no user input. If the SPECIFIC PTS option is selected (or whenever deleting from a type 106 surface), the user simply inputs one by one the points to be deleted. For the RANGE OF M or RANGE OF N options, the user first enters the first point to be deleted on a given N or M line, and then the last point to be deleted on the same N or M line. All points between these two points,

and including these two points, will be deleted. Points may be input by either picking the point from the display or by entering the N,M indices at the keyboard. If the surface is a curve (that is, it has only one N-line), a single index for the point may be entered. If the surface is a type 106, the points must be picked from the display since there is no order implied among the points making up the surface. The selected option may be switched at any time while deleting points from a surface, so that some N-ranges, some M-ranges, and some specific points may all be deleted from a single surface without generating intermediate surfaces. When all points and ranges of points to be deleted have been input, the user enters END at the keyboard and is prompted to enter a name for the new surface to be generated. Note that the new surface generated may be non-rectangular even though the original surface was rectangular, unless the same number of points are deleted from each N-line on the surface.

1.4.5 TOP / GRID_SURFACE / DELETE_LINE

This function provides the capability to delete lines from a surface grid. The user first inputs the surface to delete lines from. Then the user successively selects lines to be deleted (up to a maximum of 40) and inputs an "END" to conclude the line selecting process. The user is then prompted for a new surface name.

1.4.6 TOP / GRID_SURFACE / ADD_LINE

This function provides the capability to add lines to a surface grid. The added line will be included in the surface grid parametrically. The user first selects the option (cursor or keyboard) describing the method of inputting the location of the line to be added, and then inputs the surface name. The point at which the line will be added is then input by either cursor or keyboard (dependent on the previously selected option). The user then indicates the direction the new line by picking an existing line on the surface, which runs in the same direction. The last prompt requires a name for the new surface.

1.5 TOP / BLOCK_BUILD

The BLOCK_BUILD option attaches the user to the VIRGO subroutines. The options available to the user are: POINTS, EDGES, SPLINE, TFI, ELLIPTIC_GRID, METRIC_ANALYSIS, BEZIER and SURFACE_MAPPING.

1.5.1 TOP / BLOCK_BUILD / POINTS

The POINTS function provides the capability to generate planes in space displayed as either a set of points or surfaces. The user should view the input coordinates as the diagonally opposite corners of a three-dimensional cube and the number of planes in each direction as the subdivision of the faces of the cube.

1.5.2 TOP / BLOCK_BUILD / EDGES

The EDGES option provides the capability to generate a three-dimensional curve from existing points, surface nodes or keyboard inputs. In this process the user will be prompted to input points in order and END when the curve has been defined. REDO can be entered if the user is unsatisfied with the edge being defined and wishes to start over. Before a point is input, the user should set the option list to the appropriate menu option, and then pick a point or input the x,y,z coordinates of a new point (see below for options). As the edge is created, the displayed line and surface name will cycle through the colors as new points are added, and the total number of points on the edge will be displayed at the bottom of the prompt window.

- (1) POINT SURF should be set when picking a point surface or when inputting x,y,z coordinates with the keyboard.
- (2) GRID SURF should be set when the input point is a node on an existing surface.
- (3) ERASE POINT deletes a previously input point on the curve, if used before the curve is finalized with END.
- (4) MOVE POINT allows the user to move a previously defined point in the curve to a different location, if used before the surface is finalized with END.
- (5) CLUSTER is a special option which allows the user to generate a circular grid or field of points around a selected location without including the picked location in the curve. It is used to generate points that can be picked with options (1) and (2). The user is required to enter the plane in which the cluster should lie, the radius of the circle, the number of points along each ray, and the number of rays in the circle.

1.5.3 TOP / BLOCK_BUILD / SPLINE

The SPLINE function provides the capability to increase and/or redistribute the points on a line. The user is provided a number of curve fitting options, clustering of points at internal and/or endpoint locations, and a choice of inokur, cubic, or equal spacing. In the six step procedure that is outlined below, the user is queried for the spline information.

- (1) INPUT THE SURFACE NAME. This prompt requires the user to key in the name of an existing line, pick a line from the display, or select a line from the surface list while in input mode. It is important to note that a line chosen from the display is not necessarily an independent surface, it can be an existing - or m- line on a displayed surface.
- (2) SELECT INTERNAL BREAK POINTS ON CHOSEN LINE. A break point is an existing internal point about which the user specifies the spacing. By specifying break points, the line chosen above is subdivided into (# break pts +1) sections in which the spacing, number of points and splining is handled independently. If the user wants

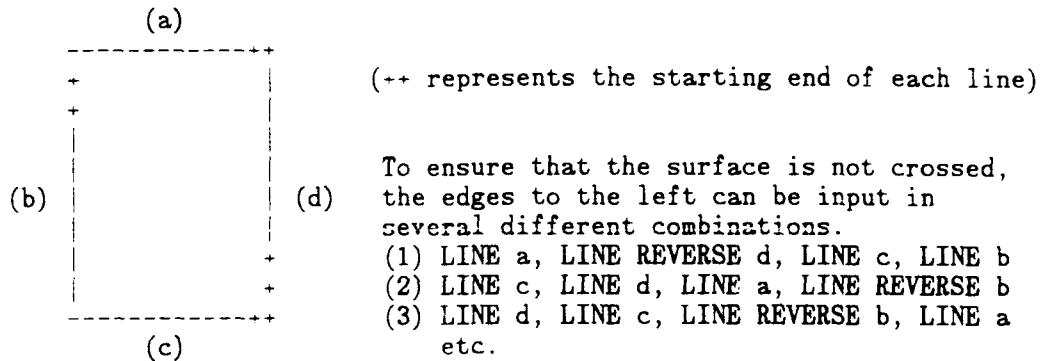
break points in the line, the location of each point will be requested. The keyboard or mouse inputs of each break location is used to find the closest existing node point for the break. The end point of a line cannot be chosen as a break point, and therefore any line which is defined by only two points cannot have any break points.

- (3) CHOOSING THE SPLINE BASIS AND NUMBER OF POINTS/INTERVAL. For most applications the default basis (ARCLENGTH) is suggested, however the X,Y,Z DIRECTIONS for splining have been retained to allow flexibility. The user should be aware that when using the latter options, the line needs to be single valued in the direction selected. Once an option is selected the user is required to input the number of points in the interval, including both endpoints in the count. Equal spacing of the points can be chosen by inputting a negative number of points.
- (4) SELECTING THE ENDPOINT SPACING. If equal spacing is not chosen in the previous step, the user will be required to input the spacing at each end of the line or section in question. The index number in the prompt indicates at which point on the existing line the spacing is being set. If the user is interested in specifying the spacing at only one end, autospacing should be selected at the free end. Autospacing will calculate an appropriate 'large' or 'small' spacing for the free end based on the number of points and the fixed end spacing. Autospacing should not be selected at both ends.
- (5) CHOOSING WEIGHTING FACTOR AND STRETCHING PARAMETERS. In this step, the user will select the stretching method used to distribute the points along the line (Vinokur or Cubic), and the type of curve fit preferred (TENSION, CUBIC, or LINEAR). LINEAR will maintain the initial shape of the curve with a straight line fit through the initial points, TENSION will slightly round the sharp slope discontinuities at the initial points, and CUBIC will fit a cubic spline through the points.
- (6) INPUT NEW SURFACE NAME. The last step requires that the user input a new name or SAME if the current name should be reused. CAUTION!! When using SAME, the current surface is not retained, therefore if the original line selected was an n- or m- line of an existing surface a new name should be chosen.

1.5.4 TOP / BLOCK_BUILD / TFI

The TFI function provides the capability to: generate a surface from four independently defined sides or regenerate a surface from the definition of its own four sides. In both cases a two-dimensional transfinite interpolation scheme is used. To accomplish the latter the user should set the menu option to SURFACE and then select the surface to be regenerated. When generating a surface from four independently defined sides the user will be prompted to input the sides one at a time while indicating its surface type. It is important to note that I3G/VIRGO's ordering scheme must be followed, or the surfaces will be crossed (see example below). LINE indicates that the input side is a

defined line, POINT indicates that the input side is a collapsed edge or point, and LINE REVERSE is used when the input side is a line but it is defined in the opposite direction required.



1.5.5 TOP / BLOCK_BUILD / ELLIPTIC_GRID

The ELLIPTIC GRID function provides the capability to smooth the grid on a surface with the use of LAPLACE, POISSON or POIS/CURV (Poisson with curvature) control functions. These control functions allow the user to manipulate the spacing and orthogonality of a surface grid in an interactive setting.

LAPLACE sets the control functions to zero and the grid will tend towards equal spacing everywhere.

POISSON linearly interpolates Thomas-Middlecoff control functions across the mesh to control the spacing. These functions are based on the arclength on the boundaries.

POIS/CURV uses Thomas-Middlecoff control functions as a start and then uses the curvatures of the adjacent boundaries to modify the control functions.

Selection of any of the control functions is independent of the angle and/or spacing control associated with each side of the mesh. Sorenson's GRAPE Method control functions are used to enforce orthogonality and/or spacing. Their use requires the input of exponential decay rates to calculate the P & Q control functions. Default values and additional options have been made available. These options represent values that enforce the orthogonality and spacing in a WEAK, MEDIUM or STRONG fashion. USER SPECIFY will allow the user to input P & Q control functions independently. When this control function is selected the user has the capability to control the angle of the grid lines and their spacing with respect to each edge independently.

ANGLE CONTROL OPTIONS

- (1) NO CONTROL - no angle control requested, control function set to zero (La²lace).
- (2) ORTHOGONAL - requires the gridlines to be orthogonal to the side in question.
- (3) LOCAL ANGLE - the angles between the gridlines and the side will be the same as the initial angles.
- (4) INTERPOLATE - interpolates the angles of the corners to get angles for the interior gridlines.

SPACING OPTIONS

- (1) INTERPOLATE - uses the arclength spacing on the adjacent edges to interpolate for the interior spacing of the gridlines.
- (2) LOCAL SPACING - the arclength distance between the side and the first gridline or a multiple of the distance will be maintained on the new grid (fractional multiples are acceptable, default is 1.).
- (3) USER SPECIFY - user is able to specify the spacing that is required. This is an important option when a surface has two collapsed edges. Under this option, the user will be asked to provide a spacing distance at one or more locations along the side.

After selecting the ELLIPTIC GRID function, the user chooses the smoothing mode (PLANE, or PARAMETRIC) and inputs the name of the surface to be smoothed (the old SURFACE option has been removed, and the user is directed to the SURFACE MAPPING options to map a surface to a database and then PARAMETRIC may be used to smooth the surface).

The user is also provided the option provides to smooth the WHOLE GRID or a SUBSET GRID of the surface. When the later, option is picked, two diagonally opposite points are required to define the region to be smoothed. These points are chosen either by picking them off the display or by inputting their computational coordinates (M,N). The defined region is displayed for the user's inspection and can be reselected if desired.

Now that a region is selected for smoothing (either a whole surface or a subset of a chosen surface) the user picks the desired control function. All three modes use the control functions defined above. In each mode the user will be requested to input the following information (additional input may be requested):

- (1) Number of smoothing iterations that should be run if convergence is not reached (Default = 100)
- (2) Convergence value, which is based on the physical distance that the points are moved per iteration (Default = 0.001).
- (3) Relaxation number, which controls speed of convergence. Higher numbers provide quicker convergence but may cause solution divergence (Default = 0.3). Values input should be between 0 and 2. An under-relaxed scheme will be used for values between 0 and 1 while an over-relaxed scheme will be used for values between 1

- and 2.
- (4) Feedback frequency, which controls the number of iterations between display update (Default = 5).
 - (5) New surface name. CAUTION!! If SAME is used the initial surface will not be maintained.

Defaults are given where possible and may require only a carriage return to accept them.

Once the smoother has been started, the user can interrupt the solution at any time by hitting the F1 key. After the smoother pauses, the user can stop the solution at the current iteration, redefine any parameters previously selected except for the number of iterations, or resume the current smoothing. (HINT: the user should initially choose more iterations than are deemed necessary, allowing extra time to redefine parameters.

1.5.5.1 TOP / BLOCK_BUILD / ELLIPTIC_GRID / SURFACE

 This option has been removed and the user is directed to the SURFACE MAPPING options to map a surface to a database. PARAMETRIC smoothing can then be used to elliptically smooth the grid.

The SURFACE mode provides the capability to interpolate a surface such that it lies on a 'database' of surfaces of the user's choice (common edges are not required between the database and the chosen surface, however, the surface must fall completely within the selected database). It is necessary that the 'database' is single valued in the direction of the interpolation and therefore temporary rotations to ensure this condition are made. The user is prompted to provide a rotation angle which will produce monotonicity in one direction. Both the surface and 'database' are rotated by the given angle and then rotated back after the interpolations are completed. This option can be used to recover an accurate surface definition, which could be lost through manipulating and re-paneling with a sparse number of grid points.

When the surface is ready for interpolation, the user is prompted to select a fit method for the point mapping and then input the edges that will be mapped to the database. The fit methods available are, TFI and BI-CUBIC. TFI uses a straight line fit between existing points of the database while BI-CUBIC uses bi-cubic piecewise splines to locate the new points. The edges of a surface are only mapped to the database if the user requests it. By default the edges of the original surface will become the new edges unless the user specifies they should be mapped.

WARNING! This option is very slow by nature and it is only recommended for recovery of an accurate surface definition (1 iteration). Once the

correct shape is produced, further smoothing should be done in the PARAMETRIC mode which will maintain the initial shape. The user should also consider using the SURFACE_MAPPING options to interpolate a surface to a given shape.

1.5.5.2 TOP / BLOCK_BUILD / ELLIPTIC_GRID / PLANE

The PLANE mode should only be selected when the surface being smoothed does not require the internal mesh to lie on the initial surface. The final mesh produced at the end of the smoothing will only maintain the shape of the edges and will interpolate to get the internal surface. The user will be prompted to input a plane in which the surface is not double valued.

1.5.5.3 TOP / BLOCK_BUILD / ELLIPTIC_GRID / PARAMETRIC

The PARAMETRIC mode should be used in most instances where the shape of the surface needs to be maintained. All surface points are put into uv space therefore monotonicity of the surface is not required in any direction. The final smoothed surface points will lie on the initial surface.

1.5.6 TOP / BLOCK_BUILD / HYPERBOLIC_GRID

The HYPERBOLIC GRID function provides the capability to generate O, C and H-type hyperbolic planar grids around arbitrary bodies or starting from a given edge. The methods used are documented in AFWAL-TM-84-191.

All methods require the user to pick or input:

- (1) the starting line ('N-line' = 1)
- (2) the plane in which the grid will be created
- (3) the number of points away from the starting line (# of 'N-lines')
- (4) the distance of the first spacing (between N=1 and N=2)
- (5) the distance to the outer boundary (between N=1 and N = last)
- (6) a value for ESCAL
- (7) implicit and explicit smoothing parameters
- (8) a value for ALPHA

Defaults are included where possible. These values can be accepted by returning after the prompt or can be changed by inputting a different value where required. Make sure that two values are input where necessary. Inputs can be separated either by a comma or a space.

ESCAL defines the rate at which the grid distribution around the body will transition from the input coordinate distribution to a grid with equal cell areas for a given radial location. Large values can cause awkward spacing and if this happens reduce the value.

The Explicit smoothing parameter defines the amount of fourth order dissipation to be used to damp numerical oscillations. It has an

adverse effect on orthogonality and too large a value can lead to numerical instabilities. For O-type grids a range of this smoothing parameter allows the amount to vary from the minimum value at the surface to the maximum value at the outer boundary. By using a range, orthogonality can be maintained at the surface but the grid can be highly smoothed away from the surface.

The Implicit smoothing parameter is not as effective as the explicit smoothing but it will not cause numerical instability problems. Implicit smoothing has the dual effect of adding higher order smoothing plus increasing the amount of explicit smoothing that can be added before the procedure becomes unstable. There is no stability limit however practical limits can be reached by adversely effecting orthogonality and spacing. As discussed above a range is also required for this smoothing parameter when generating an O-type grid.

ALPHA controls the nature of the finite difference marching algorithm used to march the grid from the body to the outer boundary. Values greater than 1 tend to weigh the procedure in favor of the implicit method and has the result of improving the smoothness of the grid. Values less than 1 will result in improved orthogonality, however on complex bodies, this can cause awkward spacing. For the O-type grid a range of values will be requested.

Attention!! If the grid is generated in the wrong direction or inside the body (you will know when it happens) the points on the starting line have to be reordered before generating the grid. This can be done by using the `MANIPULATE / CHANGE / REORDR_M_LINES` option.

1.5.7 TOP / BLOCK_BUILD / METRIC_ANALYSIS

The Metric Analysis option allows the user to analyze the derivatives which transform the surface from the physical domain to the computational domain. This capability is very useful when attempting to evaluate the "quality" of the grided surface (grid spacing, etc.).

In order for a grid to perform well in a flow solver, the metrics of each surface must be "smooth"; i.e. the metrics, when plotted, should form a fairly smooth surface with no discontinuities or "spikes". A point of discontinuity in the metric plot points out an area of poor spacing in the surface grid, and a potential disaster area for the flow solver.

Before the metric plot appears, you will be asked to input a scaling factor for the metric values (the z-axis values). A good procedure is to accept the default scaling value of 10.0, and examine the plot for the locations of maximum and minimum values. Once you have determined the areas of interest on the plot, you might recalculate and replot the same metric, but with a scale factor of 1.0. This plot may then be used with the Analysis function to determine the values of the maximums and minimums.

The plot of the selected metric will be given a name on the list which is a combination of the first 12 characters of the surface name, followed by a space and then an abbreviation for the metric which is being plotted ('XE' for 'DX/D(XI)', 'YN' for 'DY/D(ETA)', etc.).

1.5.7.1 TOP / BLOCK_BUILD / METRIC_ANALYSIS / DX/D(XI)

Choosing this metric will result in the calculation and plotting of the derivative of the X physical plane variable with respect to XI (the first coordinate, or abscissa of the computational domain).

1.5.7.2 TOP / BLOCK_BUILD / METRIC_ANALYSIS / DY/D(XI)

Choosing this metric will result in the calculation and plotting of the derivative of the Y physical plane variable with respect to XI (the first coordinate, or abscissa of the computational domain).

1.5.7.3 TOP / BLOCK_BUILD / METRIC_ANALYSIS / DZ/D(XI)

Choosing this metric will result in the calculation and plotting of the derivative of the Z physical plane variable with respect to XI (the first coordinate, or abscissa of the computational domain).

1.5.7.4 TOP / BLOCK_BUILD / METRIC_ANALYSIS / DX/D(ETA)

Choosing this metric will result in the calculation and plotting of the derivative of the X physical plane variable with respect to ETA (the second coordinate, or ordinate of the computational domain).

1.5.7.5 TOP / BLOCK_BUILD / METRIC_ANALYSIS / DY/D(ETA)

Choosing this metric will result in the calculation and plotting of the derivative of the Y physical plane variable with respect to ETA (the second coordinate, or ordinate of the computational domain).

1.5.7.6 TOP / BLOCK_BUILD / METRIC_ANALYSIS / DZ/D(ETA)

Choosing this metric will result in the calculation and plotting of the derivative of the Z physical plane variable with respect to ETA (the second coordinate, or ordinate of the computational domain).

1.5.8 TOP / BLOCK_BUILD / SURFACE_MAPPING

The SURFACE MAPPING option provides the capabilities to collapse a surface onto any x, y, or z plane with CONSTANT PLANE, or to INTERPOLATE a surface to a given database of points. Options available to the user are: CONSTANT_PLANE, or INTERPOLATE.

1.5.8.1 TOP / BLOCK_BUILD / SURFACE_MAPPING / CONSTANT_PLANE

The CONSTANT_PLANE function provides the capability to map a surface

onto an constant x,y or z plane. (e.g. mapping a surface such that all y=0)

1.5.8.2 TOP / BLOCK_BUILD / SURFACE_MAPPING / INTERPOLATE

The INTERPOLATE function provides the capability to map a surface onto an existing database of points. This database (DB) can be made up by as many as ten different surfaces. To accurately interpolate the surface, it is important that the surface and DB are not double-valued in at least one direction. Under this option, both the surface and DB can be temporarily rotated and then interpolated in the most appropriate direction. The user is given the option to hold any or all of the original edges fixed or to map them to the database.

When the surface is ready for interpolation, the user is prompted to select a fit method for the point mapping and then input the edges that will be mapped to the database. The fit methods available are, TFI and BI-CUBIC. TFI uses a straight line fit between existing points of the database while BI-CUBIC uses bi-cubic piecewise splines to locate the new points. The edges of a surface are only mapped to the database if the user requests it. By default the edges of the original surface will become the new edges unless the user specifies the sides should be mapped.

When a surface is unable to be mapped to the DB (may be due to several factors, but most often, there is no DB present for the mapping of all the points desired), the surface will be displayed in white, while unmapped points will be displayed in red. The user may save this new surface if desired (to manipulate the problem areas with other options) by a return when prompted, or may get rid of it by leaving INTERPOLATE through the pick of a new option.

1.5.9 TOP / BLOCK_BUILD / BEZIER

The BEZIER options that are available here: (1) provide the capability to generate a control point curve and the corresponding bezier curve, (2) generate a bezier curve from a predefined control point curve and (3) allows the degree of an existing control point curve to be raised.

1.5.9.1 TOP / BLOCK_BUILD / BEZIER / CREATE_BEZIER

The CREATE BEZIER option provides the capability to interactively generate a set of control points and the corresponding Bezier curve. The control points can be picked from the display or input with the keyboard. The Bezier curve is automatically displayed as the control points are input. After a control point has been added, it can be moved or erased if the user desires. The curve will be written to the file when END is input. If the user wants to start over, REDO should be input at the prompt.

1.5.9.2 TOP / BLOCK_BUILD / BEZIER / CURVE_FIT

The CURVE FIT option provides the capability to generate a Bezier curve with the de Casteljau algorithm. The user will be prompted to:

- (1) supply a curve which contains the control points
- (2) input the desired number of points on the Bezier curve
- (3) input a name for the curve

1.5.9.3 TOP / BLOCK_BUILD / BEZIER / DEG_ELEVATION

The DEGREE ELEVATION option provides the capability to increase the number of vertices on the control point curve without changing the shape of the Bezier curve that can be generated. The user is prompted to input:

- (1) the name of the control point curve
- (2) the degree of elevation (number of additional vertices)
- (3) a name for the new control point curve

The user will find that the initial and final control point curves will generate identical Bezier curves with the CURVE_FIT option. As the degree of the control curve goes to infinity the closer it will represent the Bezier curve that can be generated from its control points.

1.6 TOP / OUTPUT

The OUTPUT function provides the capability to generate an output file of surface geometry in the format of an input dataset of a particular code. Once an output file is opened, the user may direct the generation of a partial input dataset consisting of any number of surfaces that are in the current temporary model. The procedure is 1) Choose the output function in the menu, 2) Choose the output code option (if the output is for a panel code or grid generator, the PANEL CODES or GRID GENERATORS option is selected on the OUTPUT menu and then the specific panel code or grid generator option is selected on the subsequent menu), and 3) Select the surface to be output. For some output formats additional information about each surface is prompted for. Users should be aware of what information the code they will be running requires, and therefore should be somewhat familiar with the code. Once a surface has been written to the output file, the display of that surface will be turned off if it had been displayed.

1.6.1 TOP / OUTPUT / PANEL_CODES

The PANEL CODES option provides the capability to generate an output file of surface geometry in the format of an input dataset of a particular panel code. The following panel codes are supported through this option: HESS, MCAERO, PANAIR, VSAERO, S-HABP, and QUADPAN.

1.6.1.1 TOP / OUTPUT / PANEL_CODES / HESS

This output format option will cause the surface geometry to be output in the form of a HESS geometry input dataset. The procedure is to choose the code option, and then select the surface to be output. The user is then prompted for the lifting/non-lifting flag for the surface. Only rectangular surfaces may be output. Note that this data format is particularly good for manual operations on the geometry.

1.6.1.2 TOP / OUTPUT / PANEL_CODES / MCAERO

This output format option will cause the surface geometry to be output in the form of a MCAERO geometry input dataset. The procedure is to choose the code option, select the surface type, and then select the surface to be output. The user is then prompted for the point retraction factor for the surface. The user can select the default point retraction factor of 0.01 by entering a RETURN. If the code option selected was WAKE/NON-CONST, the user is prompted for the wake normal calculation factor and the X, Y and Z components of the wake normal. Only rectangular surfaces may be output.

1.6.1.3 TOP / OUTPUT / PANEL_CODES / PANAIR

This output format option will cause the surface geometry to be output in the form of a PANAIR geometry input dataset. The procedure is to choose the code option, select the surface option, and then select the surface to be output. Only rectangular surfaces may be output.

1.6.1.4 TOP / OUTPUT / PANEL_CODES / VSAERO

This output format option will cause the surface geometry to be output in the form of a VSAERO geometry input dataset. The procedure is to choose the code option, select the patch option, and then select the surface to be output. The user then selects the patch type option and inputs the patch assembly number. The user is then prompted for the spanwise node point numbers and whether the slope is continuous or discontinuous at the node points. The format for inputting the node points is as follows - a string of point indices (separated by a space or comma) followed by a space or comma and then a string of C's and D's (without delimiters) that indicate a continuous or discontinuous slope, respectively, assuming a one-to-one match between the point indices and slope specifications. An example of the input for two node points is 2 5 CD. A maximum of 15 node points may be specified. If no node point specification is necessary for the surface, a RETURN is all that is needed. Similar input (node point numbers and slope continuity) is then requested for the points in the chordwise direction. The input format and limit of the chordwise node points is the same as the spanwise points. It is assumed that the specification of chordwise node points is the same for all sections of the surface. Only rectangular surfaces may be output.

1.6.1.5 TOP / OUTPUT / PANEL_CODES / S-HABP

This output format option will cause the surface geometry to be output in the form of a S-HABP geometry input dataset. The procedure is to choose the code option and then to input Y for "yes" if the surface coordinates are to be transformed. The transformation will change the X and Y coordinates to -X and -Y, respectively. The user then selects the option and inputs either the surface name or types in END if all the surfaces for the current panel have been entered. If the surface input is the first surface of a new panel, the user will fill out a form to specify the information for the Panel Identification Card and also a form for the Scale Factor Card if the appropriate Scale Factor Flag was specified on the Panel Identification Card form. The user successively inputs all of the surfaces for one panel until an END has been entered. This procedure is then repeated for subsequent panels until all panels have been output. Only rectangular surfaces may be output.

1.6.1.6 TOP / OUTPUT / PANEL_CODES / QUADPAN

This output format will cause the surface geometry to be output in the form of the 'panel' and 'section' part of the QUADPAN V3.2 input dataset. The procedure is to choose the code option, select the surface type and then select the surface to be output. The user is then prompted for the wet, force and image flags for the surface. Only rectangular surfaces may be output.

1.6.2 TOP / OUTPUT / GRID_GENERATRS

The GRID GENERAT(o)RS option provides the capability to generate an output file of surface geometry in the format of an input dataset of a particular grid generator. The following grid generators are supported through this option: EAGLE, PGRID, PRE-SCRAMG, SCRAMG, WGRID, and ZGRID.

1.6.2.1 TOP / OUTPUT / GRID_GENERATRS / EAGLE

This output format option will cause the surface geometry to be output in the form of an EAGLE geometry input dataset. The procedure is to choose the code option, and then to input the starting index number or enter RETURN for a starting index number of one. The user is then asked to input the name of the surface corresponding to the starting index number. Additional surfaces corresponding to subsequent index numbers will be prompted for and may be input if appropriate. Only rectangular surfaces may be output.

1.6.2.2 TOP / OUTPUT / GRID_GENERATRS / PGRID

This output format option will cause the surface geometry to be output in the form of a PGRID geometry input dataset. The procedure is to choose the code option, and then to input first the wing upper surface

and then either the wing lower surface or the keyword SYM to have the program create a symmetric lower surface. The program then checks to ensure that the wing upper and lower surfaces have the same number of span stations. However, the user is responsible for ensuring that the span station cuts are at the same location for the upper and lower surfaces and that they are constant span station cuts. After the user has input the wing surface(s), the user is prompted for the fuselage surface. Note that a sting will be added as the last "fuselage" station and will be located ten fuselage lengths downstream of the last actual fuselage station. Lastly, the user is presented with a series of forms (one form for each span station) on which to enter additional wing definition data, including incidence angle (in degrees), trailing edge included angle (in degrees), trailing edge bisector slope, and singularity point coordinates. The user has the option of inputting the incidence angle, included angle, and bisector slope or having these values calculated. Only rectangular surfaces may be output.

1.6.2.3 TOP / OUTPUT / GRID_GENERATORS / PRE-SCRAM

This output format option will cause the surface geometry to be output in the form of a SCRAM pre-processor geometry input dataset. The procedure is to choose the code option, then to choose the data coordinate system (the coordinate system defining the I3G surface data) option, and then input first the wing upper surface and then either the wing lower surface or the keyword SYM to have the program create a symmetric lower surface. The program then checks to ensure that the wing upper and lower surfaces have the same number of span stations. However, the user is responsible for ensuring that the span station cuts are at the same location for the upper and lower surfaces and that they are constant span station cuts. If the surface data is defined in the I3G coordinate system, the Y and Z coordinates will be switched when they are written to the output file. This switching will not occur if the data is defined in the pre-processor coordinate system. After the user has input the wing surface(s), the user is prompted for the fuselage surface. The user is then presented with a form on which to enter SCRAM Pre-processor parameters, including a title, the number of points per cut for input to SCRAM, the number of grid points around the cross-section and from the configuration to the far field, the aspect ratio, the distance from the center plane to the far field, the fraction of the grid on the airfoil, and the wing leading and trailing edge slopes. Only rectangular surfaces may be output.

1.6.2.4 TOP / OUTPUT / GRID_GENERATORS / SCRAM

This output format option will cause the surface geometry to be output in the form of a SCRAM geometry input dataset. The procedure is to choose the code option, then to choose the configuration nose type (sharp or wedge) option, and then input the surface or surfaces (up to a maximum of 20) to be output, and then to type in END to indicate that all of the surfaces have been entered. These surfaces should be made up of YZ plane cuts through the configuration, which can be generated

with the CUT PLANE function under MANIPULATE. The surfaces should also have the cuts ordered starting from the front of the configuration and proceeding aft (i.e., increasing X). The first surface input should contain the nose of the configuration and subsequent surfaces should proceed in order to the rear of the configuration. It is assumed that if more than one surface is input the surfaces touch at their edges and therefore, the first cross section of surfaces two through N-surfaces is eliminated from the output to prevent duplication of a cross section. After the user has typed in END to indicate the last surface has been input, the user is presented with a form on which to enter grid generation data, including a title, the number of grid points around the cross-section and from the configuration to the far field, grid cluster parameters, a grid scale, interpolation parameters, and the grid distance from the configuration to far field parameters. Only rectangular surfaces may be output.

1.6.2.5 TOP / OUTPUT / GRID_GENERATRS / WGRID

This output format option will cause the surface geometry to be output in the form of a WGRID geometry input dataset. The procedure is to choose the code option, and then to input first the wing upper surface and then either the wing lower surface or the keyword SYM to have the program create a symmetric lower surface. The program then checks to ensure that the wing upper and lower surfaces have the same number of span stations. However, the user is responsible for ensuring that the span station cuts are at the same location for the upper and lower surfaces and that they are constant span station cuts. Lastly, the user is presented with a form on which to enter grid generation data, including a title, the number of grid cells, surface grid generation parameters, and 3-D grid generation parameters. Only rectangular surfaces may be output.

1.6.2.6 TOP / OUTPUT / GRID_GENERATRS / ZGRID

This output format option will cause the surface geometry to be output in the form of a ZGRID geometry input dataset. The procedure is to choose the code option, and then the user is presented with a form on which to enter ZGRID input data, including a title, a geometric scale factor, the types of surfaces (i.e., configuration components) to be output, and symmetry codes. After leaving the form, the user will be prompted for the surfaces that correspond to all of the surface types that were specified on the form. Note that for a forebody surface type, the first point will be assumed to be the nose point and will be output accordingly. If additional points are included in the first curve of the forebody surface, they will not be output. Only rectangular surfaces may be output.

1.6.3 TOP / OUTPUT / FLO59

This output format option will cause the surface geometry to be output in the form of a FLO59 geometry input dataset. The output interface

requires that the user specify a wing and may optionally specify a fuselage and horizontal and vertical tails. The procedure is to choose the code option, and then to input first the wing upper surface and then either the wing lower surface or the keyword SYM to have the program create a symmetric lower surface. The program then checks to ensure that the wing upper and lower surfaces have the same number of span stations. However, the user is responsible for ensuring that the span station cuts are at the same location for the upper and lower surfaces and that they are constant span station cuts. The user is then asked to input the wing dihedral angle (in degrees) or enter RETURN if the angle to be output is zero. Next, the user is presented with a series of forms (one form for each span station) on which to enter additional wing definition data, including incidence angle (in degrees), trailing edge included angle (in degrees), trailing edge bisector slope, and singularity point coordinates. The user has the option of inputting the incidence angle, included angle, and bisector slope or having these values calculated. After the user has input the wing information, the user is prompted for the fuselage surface or enter RETURN if no fuselage is to be output. If the user has input a fuselage surface, the user is asked if a sting is to be added as the last fuselage station. The user responds with a Y(es) or N(o). Note that if a sting is requested it will be located ten fuselage lengths downstream of the last actual fuselage station. After the user has input the fuselage, if appropriate, the user is prompted for the horizontal tail upper surface. As in the case of the fuselage, the user will respond with a RETURN if no horizontal tail is to be output. If the user has input a horizontal tail upper surface, the user is presented with a series of prompts and forms, similar to the wing, for the lower surface, dihedral angle and additional horizontal tail definition data. As with the wing, the program checks to ensure that the horizontal tail upper and lower surfaces have the same number of span stations and the user is responsible for ensuring that the span station cuts are at the same location for the upper and lower surfaces and that they are constant span station cuts. Finally, the user may input vertical tail information. The user is prompted for a vertical tail surface or enter RETURN if no vertical tail is to be output. Since FL059 assumes that the vertical tail is located on the centerline, the vertical tail consists of only one surface representing half of the actual tail. As with the wing and horizontal tail, if the user has input a vertical tail surface the user is then presented with a prompt and a series of forms for the dihedral angle and additional vertical tail definition data. Only rectangular surfaces may be output.

1.6.4 TOP / OUTPUT / FL067

This output format option will cause the surface geometry to be output in the form of a FL067 geometry input dataset. The procedure is to choose the code option, and then to input first the wing upper surface and then either the wing lower surface or the keyword SYM to have the program create a symmetric lower surface. The program then checks to

ensure that the wing upper and lower surfaces have the same number of span stations. However, the user is responsible for ensuring that the span station cuts are at the same location for the upper and lower surfaces and that they are constant span station cuts. The user is then asked to input the wing dihedral angle (in degrees) or enter RETURN if the angle to be output is zero. Lastly, the user is presented with a series of forms (one form for each span station) on which to enter additional wing definition data, including incidence angle (in degrees), trailing edge included angle (in degrees), trailing edge bisector slope, and singularity point coordinates. The user has the option of inputting the incidence angle, included angle, and bisector slope or having these values calculated. Only rectangular surfaces may be output.

1.6.5 TOP / OUTPUT / WIBCO

This output format option will cause the surface geometry to be output in the form of a WIBCO geometry input dataset. The procedure is to input the code option, and then input either the wing upper surface or the keyword NONE, as prompted by the program. If a wing upper surface is input, the user next inputs the wing lower surface, and is then prompted one by one for the values of ANOSW (sharp/blunt nose wing sections), XMOM (X-position for calculating moments), REFAR (wing reference area), and WS (wing Cp distribution plot scale factor). Next, or if no wing upper surface was selected, the program prompts to input the inboard side of pylon number 1 or the keyword NONE. If a pylon inboard side is input, the user next inputs the outboard side of pylon number 1, and enters one by one the values for TITL PY (pylon title), and PNOS (sharp/blunt nose pylon sections). Next, or if no pylon inboard side was selected, the program prompts to input the winglet inboard surface, or the keyword NONE. If a winglet inboard surface is input, the user next inputs the winglet outboard surface and enters one by one the values for TITL W (winglet title), and VNOS (sharp/blunt nose winglet sections). Only rectangular surfaces may be output.

1.6.6 TOP / OUTPUT / AERHET(DSGEOM)

This output format option will cause the surface geometry to be output in the form of an AERHET geometry input (more specifically, DSGEOM input) dataset. The procedure is to choose the code option and then input the surface or surfaces (up to a maximum of 20) to be output, and then to type in END to indicate that all of the surfaces have been entered. These surfaces should be made up of YZ plane cuts (can be generated with the CUT PLANE function under MANIPULATE) through the configuration. The surfaces should also have the cuts ordered starting from the front of the configuration and proceeding aft (i.e., increasing X). The first surface input should contain the nose of the configuration and subsequent surfaces should proceed in order to the rear of the configuration. It is assumed that if more than one surface is input the surfaces touch at their edges and therefore, the first

cross section cut of surfaces two through N-surfaces is eliminated from the output to prevent duplication of a cross section. After inputting the surfaces that make up the configuration, the user will be presented with two forms to fill out. The first form is used to specify the information on the Title and Geometry Control Options cards. The second form, which will be displayed for each cross section, allows the user to input the Cross Section Coordinate Data Control cards information. The user is required to input the control point (NC) information, and may optionally input the slope (straight line, right-handed, or left-handed) information.

1.6.7 TOP / OUTPUT / PNS(QUIKII)

This output format option will cause the surface geometry to be output in the form of an PNS geometry input (more specifically, QUIKII input) dataset. The procedure is to choose the code option and then input the surface or surfaces (up to a maximum of 20) to be output, and then to type in END to indicate that all of the surfaces have been entered. These surfaces should be made up of YZ plane cuts through the configuration, which can be generated with the CUT PLANE function under MANIPULATE. A maximum of 120 cross section cuts, i.e., total of all of the surfaces input, is allowed for input to the QUIKII program. There is also a maximum of 30 points in the pitch-plane half cross section. The surfaces should also have the cuts ordered starting from the front of the configuration and proceeding aft (..e., increasing X). The first surface input should contain the nose of the configuration and subsequent surfaces should proceed in order to the rear of the configuration. It is assumed that if more than one surface is input the surfaces touch at their edges and therefore, the first cross section cut of surfaces two through N-surfaces is eliminated from the output to prevent duplication of a cross section.

1.6.8 TOP / OUTPUT / BLOCK

This output format option will allow the surfaces to be output in a block format for a three-dimensional grid generator (PLUTO) and allow the user to create the boundary condition file for the MERCURY Euler Code (AFWAL-TM-88-217). Under this option the user will be prompted for:

- (1) the number of blocks to be output
- (2) the six faces of each block
- (3) the boundary conditions on each face

1.6.9 TOP / OUTPUT / GRIDGEN

This output format option will output the surfaces in the database format required by the GRIDGEN software.

1.6.10 TOP / OUTPUT / IGES

This output format option will cause the surface geometry to be output

in an IGES format. The specific IGES format is determined by the entity type of the surface. For example, a point definition surface entity type 5001 will be written out in a type 5001 format and a point surface entity type 106 will be written out in a type 106 format. Entity types 5001 and 106 are currently retained within I3G. The procedure is to choose the code option and then to input Y for "yes" if all the surfaces in the model are to be output. If the user has indicated that all the surfaces will be output, no more input is required. If the user has indicated that not all the surfaces will be output, the user must then select the surface to be output. This data format is good for transferring geometric information to other systems.

1.7 TOP / END

This function ends the current I3G session. During every I3G session an I3G.LOG file containing every I3G input is generated. After exiting the program the user is asked if the I3G.LOG file is to be deleted.

2 PERMANENT_FUNCTIONS

The PERMANENT_FUNCTIONS (lower right corner) control the current view.

2.1 PERMANENT_FUNCTIONS / CENTER_DSPLY

The CENTER_DSPLY function allows any displayed point to be centered on the screen. The cursor is used to specify the desired point. Once a point is chosen, all rotations occur about that point until a translation or another centering function is done.

2.2 PERMANENT_FUNCTIONS / CNTR_SCL_DSPLY

The CNTR/SCL_DSPLY function adds a view scaling operation to the CENTER_DSPLY operation. You first indicate the point to be centered, then indicate with the cursor another location (it need not be a displayed point) to define a radius. The display area within defined circle will be scaled to full screen.

2.3 PERMANENT_FUNCTIONS / SCALE_DSPLY

The SCALE_DSPLY function provides a cursor controlled scaling operation. Only the vertical cursor position is important. The vertical center of the screen is a scale factor of 1.0, the top of the screen applies a scale factor of 10. to the view, and the bottom of the screen generates a scale factor of about 0.1.

3 IMMEDIATE_FUNCTIONS

The IMMEDIATE_FUNCTIONS (lower right corner) are functions for which the user wants an immediate response.

3.1 IMMEDIATE_FUNCTIONS / RESET

The RESET function returns the view to the original settings of zero rotation and translations, and a scale factor of 1.0.

3.2 IMMEDIATE_FUNCTIONS / HELP

The HELP function got you here. It may be chosen at any time. You are put into the VAX-like syntax HELP routine at a point determined by the Menu Trail and any Current Menu Selection. You may return to the TOP of the HELP information by typing TOP.

3.3 IMMEDIATE_FUNCTIONS / ANALYSIS

The ANALYSIS ON function puts the program into a mode in which the user is able to analyze the coordinates and other relevant data of a displayed point. When the ANALYSIS ON function is chosen, the surface list on the left of the screen is erased and replaced with the phrase 'ANALYSIS RESULTS'. At this point, the user may use the cursor to pick any displayed point and receive information concerning this point. All other screen menus are disabled after picking the ANALYSIS ON function, except for the other IMMEDIATE functions (such as HELP and RESET). The information returned for any point picked includes the name of the surface which the point lies on, the index of the defining curve which the point lies on (N), the maximum number of defining curves (MAX N), the index of the point on this defining curve (M), the maximum number of points on this defining curve (MAX M), the total number of points defining the surface, the X, Y, and Z values at the point, the total arc lengths in both the N and M directions at the point (SN and SM), the relative location of the point along both of these arcs (REL N and REL M), and the point spacing from the chosen node to its neighbors in the N and M directions. This information is displayed on the left side of the screen in the area normally used for the surface list. After analyzing a point, the program stays in ANALYSIS mode so additional points may be analyzed. To return to the normal operating mode, the ANALYSIS OFF function (which replaced the ANALYSIS ON menu item in the immediate functions menu after entering analysis mode) must be picked. This causes the surface list to replace the analysis data, and re-enables all of the menu blocks, putting the program back in normal operating mode.

3.4 IMMEDIATE_FUNCTIONS / SHELL

The SHELL function allows the opening of fully functional windows. This capability allows the user to operate outside of the I3G/VIRGO

program without stopping the current session. The window can perform any tasks allowed under 3.2, and can be killed at any time or stowed as an icon for later use.

4 GENERAL NOTES

4.1 GENERAL NOTES / PROGRAM DIMENSIONS

Every attempt has been made to trap errors caused by exceeding program dimensions. The user should be aware of the following limits. A maximum of 300 surfaces may be loaded in one TEMP or PERM model of the program. A surface can have no more than 300 curves and any curve can have no more than 300 points. The total number of points on a surface can not exceed 9000 3-D points. (NOTE: Some machines have enough memory to increase the above mentioned values (up to 30000 points have been used at WL/FIMM) however the user should be prepared for memory overwriting if the values are set too high. Experiment. All parameters are located in the "i3g.prm" file.)

4.2 GENERAL NOTES / SCREEN DISPLAY

The right column of the screen presents the Menu Trail, Current Menu, Options, and Permanent Functions available to control the program. Choices are made with the tablet-controlled screen cursor on the PS300, by keyboard entry or thumbwheel-controlled cursor on the Tektronix 4115, by the left mouse button and mouse-controlled cursor on the IRIS terminals, and by keyboard entry or arrow-controlled/mouse-controlled cursor on Tektronix 4014 emulators. An asterisk marks the current choices. You may select any displayed choice at any time.

The left column presents a list of surfaces in the current model. The model type (PERM/TEMP) is displayed and can be changed. The list use (INPUT/DISPLAY) allows the list to be used for surface name inputs or to toggle a surface display on or off. The PERM model list is used for INPUT only. The list contains 300 surface names, so the UP/DOWN commands are used to scroll the list. When the program is in ANALYSIS mode, the left column presents the analysis data for the desired point rather than the list of surfaces.

The axis system displayed in the lower left of the screen is used to show view orientation. I3G employs a right-handed coordinate system.

The lower box is for Prompts, Keyboard Input, Warnings and Errors.

The current view settings, Rotation, Scale, Screen Center Point can be shown optionally above the lower box. This display may be toggled on or off with Function Key 6 on the PS300 and Tektronix 4115 (shifted Function Key 6 toggles off the display on the Tektronix 4115), and with the F4 key on the IRIS terminals. This information is always displayed on Tektronix 4014 emulators (including NDS, Codonics, and Modgraph), in the lower right corner of the screen.

All display windows can be toggled on or off on the PS300 only using Function Keys 1-4. Key 5 toggles the main display from full screen to one inside the menu blocks.

4.3 GENERAL_NOTES / SIDE_NUMBERING

You are occasionally required to know the order of the sides of a surface. The sides are numbered from 1 to 4 clockwise (if the surface normal is positive when side 1 is crossed into side 2). Side 1 is identified on every surface displayed by highlighting the first vector on side 1. The side adjacent to the highlighted vector is side 2; side 3 is opposite side 1; side 4 is opposite side 2. On the PS300 this highlighting is accomplished by drawing the vector at double intensity. On the Tektronix 4115, it is done by drawing the vector in red. On the IRIS terminals, it is done by drawing the vector in white. On the Tektronix 4014 emulators, it is done by drawing the vector with a dashed line.

The user will occasionally see prompts referring to M and N lines. To understand this notation, one should view a surface as a group of N lines with M points per line. M lines are therefore the lines which join the N lines together. N lines run between sides 2 and 4 and M lines run between sides 1 and 3.

4.4 GENERAL_NOTES / SURFACE_NORMAL

The surface normal direction is defined as being positive in the direction resulting from Side 1 (vector from the first point to the last point) is crossed into Side 2 (vector from the first point to the last point).

4.5 GENERAL_NOTES / SURFACE_NAMING

Surface names must obey the following naming convention. The first character must be a letter. Succeeding characters can be either letters, numbers, or blanks. A maximum of 15 characters are allowed. If an invalid name is input the invalid characters are changed to blanks and a message is displayed to the user.

A number of operations in the program generate two new surfaces as a result of the operation. The user may input the special name 'JUNK' to eliminate the generation of that particular surface. This feature is provided to eliminate the need to delete extraneous surfaces.

4.6 GENERAL_NOTES / TEKTRONIX_4115

The cursor on the Tektronix 4115 is a red square which first appears in the approximate center of the screen. The cursor is positioned using the thumbwheels and the pick is accomplished by hitting the space bar. The cursor remains at its current position after the pick and is relocated only through user movement of the thumbwheels.

Since there is no dial box for rotations of the geometry on the Tektronix 4115, different views of the displayed surface are obtained by entering VIEW under the immediate functions list (lower right of

screen), and entering elevation and azimuth angles for the desired viewpoint. These angles are absolute angles, with the original display angles set to 0,0. On the axis displayed in the lower left of the screen, axes pointing into the screen are displayed with a smaller X, Y, or Z than axes pointing out from the screen.

4.7 GENERAL NOTES / IRIS TERMINALS

There are several capabilities associated with the IRIS terminals that are not available on other supported graphics devices. I3G uses different colors to display the wire frame surfaces. A surface and its name are assigned a color when it is first displayed. When a surface is turned off, its name will return to white, but its color is retained in memory and reused if the surface is redisplayed. Colors are assigned in the order in which the surfaces are initially selected, based on a color cycle of the six base colors (blue, cyan, green, yellow, red, and magenta). Variation of each of the base colors provides additional colors. When a surface is deleted, its color is made available again at the bottom of the list of colors, so that once the original 300 surfaces have been exhausted, new surfaces are given colors according to the order in which surfaces were deleted.

The colormap which is generated for use on the IRIS is a double ramp scale which allows for depth-cuing and anti-aliasing (or single ramped when operating on a G series terminal, due to the absence of anti-aliasing).

The I3G/VIRGO display screen can be pushed, popped or stowed to view windows behind the display. This can be accomplished by placing the cursor along the top of the display. A bullseye cursor will appear, and use of the right mouse button will provide a list of options. WARNING: The move and resize options should not be used. These options will confused the mouse location due to the pixel mouse relationship that is used in the program.

Rotations and translations on the IRIS are controlled with either the dials box or keys F3, and F5 thru F12 (SEE BELOW). The F4 key is used to toggle the display of the x,y,z coordinates of the center, the x,y,z rotations, and the scale factor of the display. Key F1 is used when elliptically smoothing a surface (see section 1.5.5). By hitting F1, the smoother pauses at the current iteration and allows the user to change the input parameters, stop the smoother early or restart the iterations.

F1	F2	F3	F4	F5	F6	F7	F8	F9	F10	F11	F12
				X	Y	Z	(-)	X	Y	Z	(-)
PAUSE		90 deg rotation		TRANSLATIONS				ROTATIONS			

If the keys are used, the reverse directions for keys F5-F7 and F9-F11 are performed by pressing F8 or F12 in conjunction with one of the other keys. 90 degree rotations are performed by pressing F3 in conjunction with the rotation keys.

	****	****	
	* * *	* * *	SCALE
	* * *	* * *	
	****	****	
X ROTATE	****	****	X TRANSLATE
	* * *	* * *	
	* * *	* * *	
	****	****	
Y ROTATE	****	****	Y TRANSLATE
	* * *	* * *	
	* * *	* * *	
	****	****	
Z ROTATE	****	****	Z TRANSLATE
	* * *	* * *	
	* * *	* * *	
	****	****	

The three button mouse is used for picking, scaling and the center/scale options as follows:

LEFT BUTTON: controls the picking of surfaces from the display or choices from the menus

MIDDLE BUTTON: controls the scaling of displayed surfaces

RIGHT BUTTON: controls the center and center/scale options on displayed surfaces

L	M	R
LLL	MMM	RRR
LLL	MMM	RRR
LLL	MMM	RRR
LLL	MMM	RRR
L	M	R

The rates of the dials, buttons and mouse controlling rotations, translations, and scaling can be adjusted on IRIS terminals by using the up-arrow and down-arrow keys. All dial rates are reduced to 80 percent of their previous values by hitting the down-arrow key, or increased to 125 percent of their previous value by hitting the up-

arrow key. ALL rates are modified each time an arrow key is hit. An additional capability of the IRIS terminal may be the ability to take photographs directly from the screen using a hardwired camera. The Dunn Instruments camera, in particular, requires the image on the screen to be displayed at a different frequency than the monitor normally operates. In order to give the user the ability to make such photographs, on IRIS terminals the PF4 key toggles the frequency of the monitor display. Pressing the PF4 key will cause static on the IRIS monitor, but a clear display on the camera monitor, making it possible to take photographs. Once a photograph has been taken, pushing the PF4 key again toggles the frequency back so that program operation can continue.

Floating point errors are not detected on the IRIS terminals, by default. Thus, no error-handling mechanism is incorporated on the IRIS. This may cause unexpected results, since no flag is raised to indicate that an error has occurred, as would happen on other operating systems. The result of divide-by-zero operations is typically for the IRIS to set the result of the operation equal to zero, although according to IRIS documentation, the result of this and other floating point errors is "indeterminate".

INSTALLATION

To install I3G/VIRGO on your iris workstation, use the 'makefile' that is included with the software. This file will ensure that all necessary parts of the code are compiled and linked appropriately. Modifications to this file may be required due to differences in directory pathnames.

On line help is available, however, several steps are required to make it operational. **Helpgen.f** and **i3ghelp.hlp.may91** are the two help files that are provided. The first file should be compiled to receive a **helpgen** executable and the latter of the two files contains the text help information. Run **helpgen** and input '**i3ghelp.hlp.may91**' to the first prompt and '**i3ghelp.hel**' to the second prompt. The **i3ghelp.hel** file that is created is the keyed file that the I3G/VIRGO executable will be looking for. To ensure that it is located, edit **i3g.f**, search for '**/usr/local/i3g/i3ghelp.hel**' and modify the pathname to locate the help file.

Now you are ready to compile the program. Run your makefile to compile and link the software. The executable '**i3gvir**' should be created.

The following routines should be present for successful compilation:

add.f	- VIRGO subroutines
dummysubs.f	- dummy hooks for additional terminals and database
formsubs.f	- form subroutines
hyper.f	- hyperbolic grid generator routines
i3g.f	- I3G/VIRGO main body
i3g.prm	- parameter file
i3ghelp.hel	direct access help file
i3gnodbfiles.f	- data management and access routines
iaasutil.f	- general utility routines
irissubs.f	- IRIS specific routines
irissh.fc	- shell routine
makefile	- compilation and linking file
outputsubs.f	- output routines
proconst.for	- dummy file

REFERENCES

1. Smith, B. and Wellington, Jr., "Initial Graphics Exchange Specification (IGES), Version 3.0," NBSIR 86-3359, April 1986.