GMAN User's Guide*

The NPARC Alliance

NASA Glenn Research Center
Cleveland, Ohio

USAF Arnold Engineering Development Center
Tullahoma, Tennessee

*This is an unnumbered version of this document, created September 13, 2018. It is essentially identical in content to the 8 Jul 1997 version of the GMAN User's Guide, by M. S. Fisher and T. D. Gatzke of the Boeing Corporation. Please send corrections, additions, ideas, etc., to Mark Fisher at *mark.s.fisher@boeing.com* or Dennis Yoder at *Dennis.A.Yoder@nasa.gov*.

Contents

1 Introduction

GMAN (Grid MANager) is an interactive, menu-driven program which provides capability to generate and manipulate 3-D volume grids, and specify boundary conditions and connectivity between zones in multi-zone grids. It also provides important quality checking, display, and reporting functions. GMAN is one module of the McDonnell Aircraft Computational Grid System (MACGS). MACGS is an integrated three-dimensional grid system which includes two other modules, ZONI3G and *gpro*. ZONI3G is used for boundary grid generation, and *gpro* is used to manipulate grids and perform format conversions.

GMAN has two user interface modes; a *command line mode*, and a graphical *menu-driven mode*. In the command line mode, the user types commands in a text window or runs a script file containing the commands to be performed. The commands are processed as they are entered. In the menu-driven mode, the user operates in a graphical environment, and controls operations and display functions using menu picks. This mode provides interactive display of a computational grid. This allows the user to see the grid that is being generated. Modifications made to the grid are displayed immediately. This also greatly enhances boundary condition specification by allowing the user to see the boudaries being specified. The user can switch between command line and graphic menu modes at any time. All operations except display control are saved to a journal file in a readable format which can be easily edited and played back to repeat a set of operations.

GMAN is compatible with a wide range of operating systems and terminals. Binaries for GMAN are distributed with the Wind-US application release for Hewlett Packard (9000-7xx) and SGI (IRIX 5.3 and 6.2) workstations. Non-graphics mode can be run from virtually any terminal, while graphics menu mode requires an appropriate terminal type, or an X-terminal connection.

2 File Types

GMAN supports several formats for the grid file, described in the following list.

Common File — *Common files* use a flexible binary direct access file architecture. This allows grid data for multizone structured grids and their associated boundary condition data to be stored in the same file. It also supports overlapping grids and unstructured grids. Common Grid files, which use the *.cgd* extension, along with Common Flow files (extension *.cfl*), are used directly by the Wind-US Navier-Stokes flow solver, and by the common file post-processing utility CF-POST. Many other tools are also built around the common file format. See the *Common File User's Guide* for details about the structure of common files.

MDA/ZDA — *Mesh Direct Access* (MDA) files are binary files associated with earlier versions of NASTD, containing the coordinate information for each zone of the grid. These files were typically generated using the McDonnell Douglas zonal batch grid generator ZGRID. A *Zone coupling Direct Access* (ZDA) file is associated with the MDA file to provide boundary condition and zone interface coupling data.

ZDF — A *Zone Definition File* (ZDF) is a common file that contains only the boundary grid information for each zone. This makes for a smaller file for transporting between machines and also makes access quicker.

A *Zone Definition File* (ZDF) is a formatted ascii file from the McDonnell Douglas surface grid generator ZONI3G.

PDA — *Plot Direct Access* (PDA) files are binary files containing the computational grid and some commonly plotted flow variables.

FANSI FDA — A *FANSI Direct Access* (FANSI FDA) file is a binary direct access file containing the grid and conservation variables for a case run with the FANSI two-dimensional Navier-Stokes code.

HESS

3 How To Use This Manual

Section 4 through Section 9 of this manual describe the command line mode syntax and each of the available commands. These commands are grouped by function into the following sections:

- *Command Line Interface Mode* (Section 4) — Describes syntax of commands and general interface issues.
- *Basic Commands* (Section 5) — Covers file I/O and global grid parameters such as scaling, units, etc.
- *Grid Generation and Modification* (Section 6) — Commands that generate the interior grid by interpolation, smooth the grid using elliptic PDE solvers, or adapt the grid based on an existing solution.
- *Grid Listing Listing and Quality* (Section 7) — Commands that assess grid quality, list detail information for any point in the grid, and produce reports.
- *Boundary Conditions* (Section 8) — Commands that assign boundary conditions or calculate interpolation factors for information transfer at zonal interfaces.
- *Boundary Condition Listing and Checking* (Section 9) — Commands that check for boundary condition and coupling errors.

The final two sections describe the graphical menu-driven mode available in GMAN. Section 10 introduces the menu-driven mode, with information on display control, picking, and menu organization. Section 11 is a reference section, listing all the various menu items available.

4 Command Line Interface Mode

GMAN starts out in command line mode with input read from standard input and output written to standard output. All commands or menu picks that cause file modification are written to *gman.jou*. This file can be used to repeat a session via use of the `@` command. Note that *gman.jou* is appended to if it already exists at the beginning of a GMAN session. By typing `SWITCH` or `GRAPHICS` at the prompt, the program will switch to the menu-driven, graphics mode of operation. The user can get back to command line mode by picking `SWITCH TO TEXT` from the Main Menu (Section 10.4) in graphics mode.

Commands are entered freeform with all lowercase characters not contained within quotes converted to uppercase. Words are separated by blanks or tab characters. The start of a comment is signalled by the `!` character. If a comment appears on a line with a command then the remainder of the line is considered to be a comment and is ignored. Only one command may appear on a line and a command may not span lines. Blank lines are ignored.

In the command descriptions, parameters within `[...]` are optional. (The `[` and `]` are not to be entered as part of the command.) The `|` specifies that only one of the values should be selected.

Most words used in a command string can be abbreviated to as few as three characters, for example `SWI` for `SWITCH` or `GRA` for `GRAPHICS`. However, if more than one command has the same first three characters, enough characters must be used to make the command unique. For example, `SUBREGION` and `SUBAREA` would require four characters to avoid ambiguity.

5 Basic Commands

`ADD` — Add a zone

```
ADD [ZONE] zone [FROM] FILE file [UNITS units]
```

The parameter *units* is one of the following: `METERS(M)` (default), `INCHES(IN)`, `FEET(FT)`, `CENTIMETERS(CM)`, or `MILLIMETERS(MM)`.

`CANCEL` — Cancel pending changes

```
CANCEL
```

Causes any pending grid or boundary condition changes that have not yet been written to the files to be discarded. Changes are normally written whenever a `ZONE` or `BOUNDARY` command with a non-null zone or boundary is issued.

`DO/ENDDO` — Perform commands for a range of zones

```
DO {[ZONE] zone1 [TO] [ZONE] zone2|MAX | ALL} [[STEP] inc]
    commands
ENDDO
```

The `DO ...ENDDO` structure allows the user to perform a series of commands simultaneously for a range of zones from *zone1* to *zone2*, incrementing as specified (default increment is 1). Any valid commands — except `FILE`, `FLOWFILE`, `ADD`, `REPLACE`, or `SWITCH|GRAPHICS` — may be entered through a command file or in text mode. GMAN will loop through the specified zones and perform the given commands.

`EXIT | QUIT` — Terminate session

```
EXIT | QUIT
```

Terminates the GMAN session. Any pending changes are written to the file.

`FILE` — Specify file(s) to process

```
FILE file [CGD|MDA|FDA|ZDF|Z2M|Z2C|HESS] [coord] [index] [UNITS units]
```

The `FILE` command specifies the name of the grid file(s) to process. With the exception of Common Grid files, the file name *file* must be entered *without* a suffix. The format of the input file(s) may be specified by using one of the following parameters:

`CGD` Common Grid (suffix *.cgd*) file.

`MDA` Mesh Direct Access (MDA) file, with suffix *.mda*. If the corresponding Zone coupling Direct Access (ZDA) file does not exist, any command that would reference boundary condition data will cause the ZDA file to be created.

`FDA` FANSI Direct Access (FDA) file, with suffix *.fda*.

`ZDF` Zone Definition File (ZDF), with suffix *.zdf*. The file will first be converted to a Common Grid file, then processed.

`Z2C` Same as `ZDF`.

`Z2M` Zone Definition File (ZDF), with suffix *.zdf*. The file will first be converted to a Mesh Direct Access (MDA) file, then processed. If the corresponding Zone coupling Direct Access (ZDA) file does not exist, any command that would reference boundary condition data will cause the ZDA file to be created.

`HESS` ??, with suffix *.hes*. The file will first be converted to an FANSI Direct Access (FDA) file, then processed.

If a file type is not specified using one of the above parameters, the following process is used, in the order listed, to determine the file to use.

1. If a Common Grid file exists, it will be used.
2. If a Mesh Direct Access (MDA) file exists, it will be used.
3. If a Zone Definition File (ZDF) exists, it will be converted to a Common Grid file and the Common Grid file will be used.
4. If a FANSI Direct Access (FDA) file exists it will be used.
5. If a HESS file exists, it will be converted to a FANSI Direct Access (FDA) file and the FDA file will be used.

If a Zone Definition File (ZDF) is being processed, the input parameters *coord*, *index*, and *units* may be used to remap the computational coordinates and/or indices, and to assign units.

coord `[+|-]X|Y|Z [+|-]X|Y|Z [+|-]X|Y|Z`

index `[+|-]I|J|K [+|-]I|J|K [+|-]I|J|K`

units `METERS|M | INCHES|IN | FEET|FT | CENTIMETERS|CM | MILLIMETERS|MM`

No checks are made for left-handedness or uniqueness of specification, however. (I.e., specifying the indices as `J J K` does not generate an error message.)

<u>*Example*</u>

If the user is reading a ZDF file, and wants to change the sign of the X-coordinate and exchange the Y and Z coordinates in the resulting Common Grid file, the `FILE` command would be

`FILE` *file* `Z2C -X Z Y`

The same usage applies to the coordinate indices. Thus,

`FILE` *file* `Z2C -I K J`

Both operations may be performed at once, as in

`FILE` *file* `Z2C -X Z Y -I K J`

`REFERENCE` — Set values for y^+

```
REFERENCE [CONDITION] [LENGTH [val] | MACH [val] | PRESSURE [val] |
   REYNOLDS [NUMBER] [val] | TEMPERATURE [val]]
```

The `REFERENCE` command sets values which are used for calculating y^+ when checking grid quality.

`REMAP` — Remap grid indices

```
REMAP [+|-]I|J|K [+|-]I|J|K [+|-]I|J|K
```

The `REMAP` command allows the user to change the relationship between the grid indices and the physical coordinates, essentially remapping the indices. The specified index directions represent the existing mapping (i.e., the old coordinate directions), and their order determines the new mapping (i.e., the new coordinate directions). The syntax is thus

```
REMAP "which old index is new i" "which old index is new j" \
      "which old index is new k"
```

Note - in earlier versions of GMAN this command was called `INDEX`.

Example

Suppose we wish to remap the indices such that the new i direction is the same as the old j direction, the new j direction is the same as the old k direction, and the new k direction is the same as the old $-i$ direction. I.e.,

New	Old
i	j
j	k
k	$-i$

The appropriate command would be

```
REMAP J K -I
```

`REPLACE` — Overwrite current zone with one from another file

```
REPLACE [WITH] [ZONE] zone [FROM] FILE file [UNITS units]
```

The `REPLACE` command lets the user overwrite the current zone with a zone from another Common Grid file. If the new zone is larger than the current zone, it will be necessary to compress the Common Grid file using *cfcnvt* to recover the space of the original zone.

`STATUS` — Display status information

```
STATUS
```

The `STATUS` command displays information about the currently selected file, zone, and boundary.

`SUBAREA` — Set active range of current face

`SUBAREA` *2d-range*

The `SUBAREA` command sets the active range of the current face for boundary coupling.

For a constant *i* face, *2d-range* is specified as

`J`*m1* `K`*m2* `J`*n1* `K`*n2* `| ALL`

where *m1* and *m2* are the minimum values, and *n1* and *n2* are the maximum values, of the *j* and *k* indices defining the desired subarea. The parameters may be specified in any order. `J`*n1* and/or `K`*n2* may also be specified as `MAX`, meaning the maximum value in that direction. If `ALL` is specified, the subarea is defined as the entire face.

Similarly, for a constant *j* face, *2d-range* is specified as

`I`*m1* `K`*m2* `I`*n1* `K`*n2* `| ALL`

and for a constant *k* face, *2d-range* is specified as

`I`*m1* `J`*m2* `I`*n1* `J`*n2* `| ALL`

As an example, to set the subarea on an *j* face to the range from $i = 1$ to 20 and $k = 5$ to 30, you could write

```
SUBAREA I1 K5 I20 K30
```

`SUBREGION` — Set active range of current zone

`SUBREGION` *3d-range*

The `SUBREGION` command sets the active range of the current zone.

The parameter *3d-range* is specified as

`I`*m1* `J`*m2* `K`*m3* `I`*n1* `J`*n2* `K`*n3* `| ALL`

where *m1*, *m2*, and *m3* are the minimum values, and *n1*, *n2*, and *n3* are the maximum values, of the *i*, *j*, and *k* indices defining the desired subregion. The parameters may be specified in any order. `I`*n1*, `J`*n2*, and/or `K`*n3* may also be specified as `MAX`, meaning the maximum value in that direction. If `ALL` is specified, the subregion is defined as the entire zone.

As an example, to set the subregion to the range from $i = 1$ to 10, $y = 1$ to 20, and $k = 5$ to 30, you could write

```
SUBREGION I1 J1 K5 I10 J20 K30
```

`SWITCH|GRAPHICS` — Switch to graphics mode

`SWITCH|GRAPHICS`

The `SWITCH|GRAPHICS` command switches from command line mode to graphics mode.

`TITLE` — Change file or zone title

`TITLE [TLMAIN|TLSECONDARY] [TLGLOBAL|TLZONAL] [`*new_title*`]`

The `TITLE` command allows the user to change either the file title or a zone title. If a `ZONE` command has not been issued then the file title will be modified. If a `ZONE` command has been issued then the current zone title will be modified.

`UPDATE` — Update pending changes

`UPDATE`

Force an immediate update of any grid or boundary condition changes that have not yet been written to the files.

`ZONE` — Specify zone

`ZONE` *zone*`|NONE`

The `ZONE` command specifies the zone number to which subsequent commands will apply. Any grid or boundary condition changes that have not yet been written are applied before selecting the new zone. If a zone number is not specified then information about the currently selected zone will be displayed.

If `NONE` is specified then the current zone selection will be cleared. This is useful if one wishes to issue a `TITLE` command to change the global title, but has already issued a `ZONE` command.

6 Grid Generation and Modification

Field grid generation is the first main function of GMAN. This function is used to generate and/or modify the field (i.e., interior) points of a three-dimensional zone. Prior to field grid generation, the grid on each face of the zone must be defined. The field point locations are then determined from the location of the grid points on the six bounding faces. The field points can be determined using one of two basic methods: (1) algebraic grid generation (see Section 6.2), or (2) elliptic grid generation (see Section 6.3). Field grid generation affects only the interior points of a selected range, leaving the boundary points unchanged.

GMAN can be used to generate the field grid in conjunction with the boundary grid generation module ZONI3G, or to modify or regenerate the field grids for a grid from any other source that can be converted to an acceptable format for GMAN. To assist in obtaining acceptable grids, the program contains software which will check for grid flaws and print the locations of such errors. The user may also physically rotate, translate or scale an entire zone's mesh. Changes are not made in the grid file until the user performs a save operation, so that the original grid can be restored from the file if changes are not acceptable to the user.

6.1 General Commands

`AXISYM AXIS` — Set axisymmetric flag

`AXISYM AXIS [SHOW|NONE] XY|YZ|XZ BASE` *x y z* `SLOPE` *m* `ANGLE` *a*

The `AXISYM AXIS` command sets the axisymmetric flag in the Common Grid file. The rotation axis lies in the `XY`, `YZ`, or `XZ` plane and passes through point *x y z* with a slope of *m*. *a* is the angle through which the 2D plane is rotated about the specified axis.

`GRID SINGULAR` — Make the grid singular

`GRID [`*range*`] SINGULAR [IN] I|J|K`

The `GRID SINGULAR` command resets grid point values in a given direction to make the grid singular. The command makes every point in the selected range equal to the first point in the specified direction. The parameter *range* has the same syntax as the *3d-range* parameter in the `SUBREGION` command (see p. 12).

`REDISTRIBUTE` — Redistribute grid points along grid lines

`REDISTRIBUTE [`*range*`] [IN] I|J|K [`*method* `{INDEX | ARC LENGTH | SEMI ARC LENGTH}]`

The `REDISTRIBUTE` command redistributes grid points along existing grid lines in the specified computational direction, using the selected method. The parameter *range* has the same syntax as the *3d-range* parameter in the SUBREGION command (see p. 12). This command is useful when the elliptic grid generator has destroyed boundary layer grid packing.

The choices for *method* are:

`INDEX`	Perform index interpolation, producing a highly smoothed grid. Ignore boundary grid distributions.
`ARC LENGTH`	Use all appropriate boundary grid distributions to determine the interior grid distribution.
`SEMI ARC LENGTH`	Use boundary grid distributions and index interpolation to determine the interior grid distribution.

`ROTATE` — Rotate the grid

`ROTATE` *angle* `[DEGREES] [ABOUT] X|Y|Z [AXIS] [[CENTER]` *pt1 pt2*`]`

The `ROTATE` command rotates the grid in the current zone through *angle* degrees about an axis parallel to the one specified and passing through the point (*pt1*,*pt2*). The sign of rotation is determined from the right hand rule, with the thumb pointing along the rotation axis in the direction of increasing coordinate. (*pt1*,*pt2*) is (y, z), (x, z) or (x, y) for x-, y-, and z-axis rotations, respectively.

`SCALE` — Scale the grid

`SCALE {`*xscale yscale zscale* `| [X` *xscale*`] [Y` *yscale*`] [Z` *zscale*`]}`

The `SCALE` command scales the grid in the current zone by the amount specified in each of the three coordinate directions. The x-, y-, or z-coordinate is simply multiplied by the corresponding scale factor.

`TRANSLATE` — Translate the grid

`TRANSLATE [X` *dx*`] [Y` *dy*`] [Z` *dz*`]`

The `TRANSLATE` command translates the grid in the current zone by the amount specified in each of the three coordinate directions.

`UNITS` — Display or set grid units

`UNITS [`*units*`]`

The `UNITS` command displays or set the grid units in a Common Grid file. The parameter *units* is one of the following: `METERS(M)`, `INCHES(IN)`, `FEET(FT)`, `CENTIMETERS(CM)`, or `MILLIMETERS(MM)`.

6.2 Algebraic Grid Generation

Algebraic grid generation is interpolation of the field grid from the boundary grid points. Several interpolation options are avaliable. The recommended interpolation technique is transfinite interpolation in all three directions. Each field point is interpolated from points on each of the six faces. Alternate interpolation techniques allow the user to interpolate between opposite boundaries along one of the coordinate directions, or to interpolate in two directions for "planes" marching successively in the third direction. For many applications, the algebraic grid is suitable. Algebraic techniques provide good control of the spacing near each of the faces (controlled by the surface grid generation process) and are the fastest grid generation tools available. However, slope discontinuities on the boundary are propagated into the interior and grid lines may be highly skewed or even crossed. In order to determine whether the algebraic grid is suitable, it is necessary to do some sort of grid quality checking.

The `ALGEBRAIC INTRP` item of the `INTERIOR GRID` menu (see the `GRID` command) presents several choices to the user for computing a grid based on an algebraic interpolation scheme. `LINEAR IN I`, `LINEAR IN J`, and `LINEAR IN K` interpolate the grid along straight lines connecting boundary planes of constant i, j, or k, respectively. `JK TRANSFINITE`, `KI TRANSFINITE`, and `JI TRANSFINITE` interpolate the grid in jk, ki, and ji planes, respectively, from values on the four edges of those planes. `IJK TRANSFINITE` interpolates the grid from values on the six boundary faces and is generally more useful than the other interpolation schemes.

The algebraic grid generation options in GMAN allow the user to create a grid with 3-D transfinite bilinear interpolation within a volume or on constant computational planes. This scheme requires a weighting function for the interpolations, which is implemented as a three-dimensional arc length array in the code. The default method for calculating this array computes the boundary distributions and interpolates those values to the interior using transfinite interpolation based on computational index (evenly distributed weights). In addition to weighting the interpolations, the distribution array is used to redistribute the grid along lines of varying computational index.

`STACK` — Treat 3-D grid as stack of i, j, or k planes when interpolating

```
STACK [MODE] [3D | I [PLANES] | J [PLANES] | K [PLANES]]
```

This command provides the ability to treat a 3-D volume grid as a number of i, j, or k planes when performing 2-D or 1-D algebraic interpolation. Thus if you are doing i plane stacking with JK interpolation the i direction is completely uncoupled in the calculation. If 3-D stacking with JK interpolation is used the i direction is still used in the calculation of the interpolation factors. The new i plane stacking is useful if you have a boundary layer beginning to spread out at some i plane, this spread will not influence the calculation in the wall region. Thus the viscous packing will be maintained.

`INITIALIZE` — Generate grid for range of zones

```
INITIALIZE [ZONES [zone1 [TO|THRU] zone2]] [grid_command]
```

This command generates the grid for every zone in the specified range (default is all zones) using the method described by *grid_command*. *grid_command* is any valid `GRID` command. If omitted, the default grid generation scheme is `3D INTERPOLATION`. Because this command may be used at any time in command line mode, it is not strictly for initial grids.

`GRID` — Generate or alter grid for a zone

```
GRID [range] [USING] scheme [method {INDEX | ARC LENGTH | SEMI ARC LENGTH}]
```

The `GRID` command is used to generate or alter the computational grid in a particular zone. The parameter *range* has the same syntax as the *3d-range* parameter in the `SUBREGION` command (see p. 12). The grid may be generated using algebraic interpolation or a 3D elliptic equation solver.

For algebraic grid generation, the choices for *method* are:

`INDEX` Perform index interpolation, producing a highly smoothed grid. Ignore boundary grid distributions.

`ARC LENGTH` Use all appropriate boundary grid distributions to determine the interior grid distribution.

`SEMI ARC LENGTH` Use boundary grid distributions and index interpolation to determine the interior grid distribution.

The parameter *scheme* must be one of the following:

`LINEAR INTERPOLATION [IN] {I|J|K}`
Performs interpolation along grid lines of varying i, j, or k index, as selected.

`2D INTERPOLATION [ON] {I|J|K} [PLANES]`
Performs 2D transfinite interpolation on i-, j-, or k-planes, as selected.

`3D INTERPOLATION` Performs 3D transfinite interpolation on the entire grid. This command also works for 2D grids.

6.3 Elliptic Grid Generation

Elliptic grid generation is the creation of field grid points by the solution of elliptic partial differential equations. Prior to elliptic grid generation, an algebraic grid must be generated to provide a starting point for the elliptic solution procedure. As with algebraic grid generation, the elliptic grid generation procedure can be performed on an entire zone or on a subset of a zone. Elliptic grid generation has the advantage that the resulting grid is generally smoother than an algebraic grid and, through proper choice of control functions, orthogonality can be obtained on the boundary. The disadvantage is that it is more computationally expensive and it does not always preserve the desired spacings at the boundaries.

Elliptic grid generation is a complicated process, which sometimes works and sometimes does not. The user should first seek to simply make the grid generator work on a grid which bears some resemblance to the grid that is wanted. The desired grid can then be approached by gradually adjusting the input parameters.

The elliptic PDE grid generator in GMAN may only be used for 3D grids. It solves the inverted Poisson's equation for the physical coordinates in terms of computational coordinates using a successive line over-relaxation method. Several different forcing functions may be selected, including no forcing (Laplace equation) or forcing calculated from an existing grid or interpolated from the boundaries, using arc length and curvature. In the event that the elliptic solver destroys viscous clustering at a wall, a redistribution option can be used to apply the original spacing along existing grid lines in any one of three directions.

The `ELLIPTIC SOLVER` option allows the user to smooth the grid by solving a set of elliptic partial differential equations. The inverted Poisson's (or Laplace) equation is used, treating the physical coordinates as the independent variables. After selecting a forcing function from the displayed list (`LAPLACE` is the default), selection of `RUN SOLVER` will prompt the user for a range of the grid on which to operate and the number of iterations desired. In the `DRAWING ON` mode (see Section 10.2), the displayed grid will be updated after each iteration, interactively showing the effect of the smoothing on the grid.

Note that solving a set of elliptic partial differential equations can be a time-consuming task. Even when solving for a relatively small grid ($40 \times 32 \times 24$, for example), run times can become significant and will vary depending on the type of forcing function selected, the machine being used and the current processing load.

Stack mode is used the same as method.

`ELLIPTIC CONVERGENCE` — Set convergence criterion

`ELLIPTIC CONVERGENCE [LIMIT]` *limit*

This command sets the value used to determine convergence of the elliptic grid generator.

`ELLIPTIC DECAY` — Set elliptic decay factor

`ELLIPTIC DECAY [FACTOR]` *factor*

This command sets the rate at which the elliptic forcing functions decay from the boundary to the interior.

`ELLIPTIC FORCING` — Set elliptic forcing functions

`ELLIPTIC FORCING [FUNCTION]` *function*

This command sets the elliptic forcing function. The parameter function must be one of the following:

`LAPLACE` — This option has a strong smoothing influence on the grid but does not take into account the spacing distribution on the boundary. This often leads to undesirable effects as the grid varies from the specified boundary clustering to uniform spacing in the field. This is the default forcing function.

`ARC LENGTH` — Takes into account the grid spacing when smoothing the grid. Does not provide any explicit orthogonality control.

`ARC LENGTH AND CURVATURE`	Takes into account both the grid spacing and curvature when smoothing the grid. Does not provide any explicit orthogonality control.
`INTERPOLATED ARC LENGTH`	Takes into account the grid spacing on the boundaries and interpolates the information to the interior when smoothing the grid. Does not provide any explicit orthogonality control.
`INTERPOLATED ARC LENGTH AND CURVATURE`	Takes into account both the grid spacing and curvature on the boundaries, and interpolates the information to the interior when smoothing the grid. Does not provide any explicit orthogonality control.
`GRAPE`	Calculates forcing functions based on boundary spacing and orthogonality constraints. Forcing functions decay toward zero in the interior of a zone. Forcing functions are recomputed in the iteration loop to provide orthogonality.
`SMOOTH`	??

`ELLIPTIC GRAPE BOUNDARY` — Specify boundaries for `GRAPE` forcing function

```
ELLIPTIC GRAPE BOUNDARY {I1|IMAX|J1|JMAX|K1|KMAX} {1|2} ...
```

The `ELLIPTIC GRAPE BOUNDARY` command specifies to which boundaries the forcing function will be applied when the `GRAPE` forcing function is used.

`ELLIPTIC ITERATION` — Set maximum number of iterations

`ELLIPTIC ITERATION [LIMIT]` *iteration_limit*

This command sets the maximum number of iterations for elliptic grid generation. This limit may be overwritten by the `GRID` command.

`ELLIPTIC RELAXATION` — Set elliptic relaxation factor

`ELLIPTIC RELAXATION [FACTOR]` *factor*

This command sets the factor used in relaxing the change between the current grid and the next iteration. A factor greater than 1.0 indicates over-relaxation, and a factor less than 1.0 indicates under-relaxation.

`ELLIPTIC SHOW` — Lists current elliptic parameters

```
ELLIPTIC SHOW [PARAMETERS]
```

This command lists the current set of elliptic parameters.

`ELLIPTIC SOLVER` — Select elliptic solver

```
ELLIPTIC SOLVER {POINT | X LINE | Y LINE | Z LINE | ADI LINE} {SOR | MULTIGRID}
```

This command selects the numerical scheme to be used by the elliptic grid generator.

`ELLIPTIC UPDATE INTERVAL` — Set elliptic update interval

`ELLIPTIC UPDATE INTERVAL` *interval*

This command sets the interval between updates of the elliptic forcing functions, and only applies to the `GRAPE` forcing function.

`GRID` — Generate or alter grid for a zone

`GRID [`*range*`] [USING]` *scheme* `[`*method* `{INDEX | ARC LENGTH | SEMI ARC LENGTH}]`

The `GRID` command is used to generate or alter the computational grid in a particular zone. The parameter *range* has the same syntax as the *3d-range* parameter in the `SUBREGION` command (see p. 12). The grid may be generated using algebraic interpolation or a 3D elliptic equation solver.

For the elliptic solver, the parameter *scheme* must be `ELLIPTIC [SOLVER]`. The solver will be run for 'n' iterations.

The choices for *method*, which set the method to be used in interpolating the forcing functions into the interior, are:

`INDEX` Perform index interpolation, producing a highly smoothed grid. Ignore boundary grid distributions.

`ARC LENGTH` Use all appropriate boundary grid distributions to determine the interior grid distribution.

`SEMI ARC LENGTH` Use boundary grid distributions and index interpolation to determine the interior grid distribution.

6.4 Adaptive Grid Generation

`ADAPTIVE BOUNDARY` — Enable/disable adaptive boundary

```
ADAPTIVE BOUNDARY I1|IMAX|J1|JMAX|K1|KMAX
```

This command enables or disables boundary grid adaption for the specified boundary.

ADAPTIVE FORCING — Select adaptive grid forcing function

```
ADAPTIVE FORCING [FUNCTION] {DENSITY | MACH | TOTAL PRESSURE | STATIC PRESSURE |
   TOTAL TEMPERATURE | STATIC TEMPERATURE}
```

This command selects the flowfield variable to be used in grid adaption.

ADAPTIVE ITERATION — Set max number of iterations

`ADAPTIVE ITERATION [LIMIT]` *iteration_limit*

This command sets the maximum number of iterations to be used when adapting the grid. This limit may be overwritten with the GRID command.

ADAPTIVE WEIGHTS — Set weighting factors

`ADAPTIVE WEIGHTS` *wt0 wt1 wt2*

This command sets weighting factors for the adaptive forcing function and its first and second derivatives, respectively.

FLOWFILE — Specify flowfield file

`FLOWFILE` *filename* `[CFL|CGF|PDA|FDA|FAN]`

This command opens the specified flowfield file for use with adaptive grid commands.

GRID — Adapt the grid

`GRID [`*range*`] [USING]` *scheme* `[`*method* `{INDEX | ARC LENGTH | SEMI ARC LENGTH}]`

This command adapts the grid. The parameter *range* has the same syntax as the *3d-range* parameter in the SUBREGION command (see p. 12). The parameter *scheme* must be one of the following:

`LINEAR ADAPTION [IN] {I|J|K}`
: Adapts the grid along existing grid lines in the i, j, or k direction.

`2D ADAPTION [ON] {I|J|K} [PLANES]`
: Adapts the grid on existing grid planes with i, j, or k constant.

`3D ADAPTION`
: Performs a full 3-D adaption.

The choices for *method* are:

```
INDEX
ARC LENGTH
SEMI ARC LENGTH
```

7 Grid Listing and Quality

Checking grid quality is an area that is being investigated and is very difficult to quantify. The ultimate goal of grid quality checking is a quick, accurate flowfield solution; however, running the solution is a very expensive way to check a grid. The traditional way of checking a grid is to look at it. This can be very time consuming for large grids containing hundreds of thousands to millions of points, and it is very possible that a severe grid flaw could be missed. The grid quality checks of the QBERT Grid Evaluation Code have been implemented in the field grid generator. These checks are performed using the grid quality menu option. The user can perform these checks on the algebraic grid or at any stage of elliptic grid generation and can operate on part or all of a zone.

These checks will indicate the location of any negative volume cells, zero volume cells, collapsed faces, or crossed sides. The user may then look at the grid in that region to identify corrective action. Of all the problems, negative volume (negative Jacobian) cells are the most severe and should be corrected. Zero volume cells are generally an indication of a severe problem unless the user has intentionally chosen a topology with zero volume cells and the flow solver can handle that condition. Collapsed faces are not necessarily a problem as they occur naturally along a singular axis; however, the user should make sure that all of the collapsed faces are accounted for by the singular axes. The check for crossed sides indicates sides that are crossed or just highly skewed. This is a more rigorous test than most flow solvers employ. The user must visually inspect these areas to see what is causing the problem and how it may be corrected. Some codes have run grids with a small number of "crossed" sides with no noticeable degradation of the solution.

7.1 Checking Grid Quality

`CHECK GRID` — Perform grid checking

```
CHECK GRID [SCREEN|FILE filename [NEW|APPEND|OVERWRITE]] [PAUSE|NOPAUSE]
   [LINES lines]
```

Currently any combination of eight checks are available and the results can be displayed in any one of six output modes. Note that the output mode is no longer included in the `CHECK GRID` command. Also four of the check modes now have user defined tolerances.

Several of the checks are *directional*, i.e., they are calculated in each of the three coordinate directions of the grid, and reported as three separate values.

The grid quality checks in GMAN have been revised to work on 2-D (planer) grids. All check modes are available with areas replacing volumes (etc.). A new graphics display mode is available which places a star on each currently displayed node point which failed the check.

A new tolerance for the new `SPACING` check was added. The syntax for setting the tolerances was modified to be more user friendly.

To select the grid checking and output format modes the `CKMODE` command with the `GRID` qualifier is used. All previous grid quality modes are available as well as four new ones. The syntax for specifying the status of a check or output mode is shown below followed by a list and description of valid modes.

See Section 7.3 for the definitions of the `SCREEN|FILE`, `NEW|APPEND|OVERWRITE`, `PAUSE|NOPAUSE`, and `LINES` parameters.

`CKMODE GRID` — Select grid checking and output format modes

`CKMODE GRID [[ON|OFF]` *mode1* `[`*mode2* `[...]]]`

Each *mode* parameter is one of the following:

`ALL`	Enables or disables all check modes.
`VOLUMES_SIDES`	Check for collapsed and crossed sides, and zero and negative volumes. Same as specifying `NEGATIVE_VOLUMES`, `ZERO_VOLUMS`, `COLLAPSED_SIDES`, and `CROSSED_SIDES`.
`NEGATIVE_VOLUMES`	Check for points with negative Jacobians.
`ZERO_VOLUMS`	Check for cells with zero volumes.
`COLLAPSED_SIDES`	Check for cells with collapsed sides. A collapsed side is a face of a cell with zero area, and most often occurs at a grid singularity. (Directional)
`CROSSED_SIDES`	Check for cells with crossed sides. Crossed sides are a by-product of the negative volume computation and may indicate a highly skewed or "nearly negative" volume. (Directional)
`DISCONTINUITIES`	Check for angles between cells which exceed the user tolerance. See the `TOLERANCE` command. (Directional)
`ORTHOGONALITY`	Check for cells whose angle between faces exceeds the user tolerance from 90°. See the `TOLERANCE` command. (Directional)
`STRETCHING`	Check for consecutive cells whose ratio of side lengths exceeds the user tolerance. See the `TOLERANCE` command. (Directional)
`SPACING`	Check for cells whose spacing normal to the boundary exceeds the user tolerance. See the `TOLERANCE` command. (Directional)
`YPLUS`	Check for cells whose y^+ value normal to a wall boundary exceeds the user tolerance. The `YPLUS` check returns the actual y^+ value based on the reference conditions. This check is only applied to boundary faces which have a wall boundary condition defined. To set the reference conditions see the `REFERENCE` command. See the `TOLERANCE` command. (Directional)
`NODE_BY_NODE`	Displays each node that fails the check criteria.
`PLANE_BY_PLANE`	Displays each plane that fails the check criteria.
`MAXIMUM_PLANES`	Displays the i, j, and k planes with the most failures.
`REGIONS`	Displays corner points of the volume that bounds each group of nodes that fail the check criteria.
`SUMMARY`	Displays number of failures, average and maximum values, and node location of maximum value. This is always displayed except when in interactive mode.
`SHOW_WORK`	Displays the calulated value for each selected mode at each node in the selected subregion. This data is not saved, and thus it is not available in interactive mode.

`INTERACTIVE` The user is prompted for the desired output mode after the check computations have been completed. The results are displayed and the user is again prompted for the desired output mode. This loop continues until "quit" is selected. This allows multiple and varied viewings of check results without having to repeat the check calculations.

`TOLERANCE GRID` — Specify tolerances

```
TOLERANCE GRID [DISCONTINUITES [tol] | ORTHOGONALITY [tol] | STRETCHING [tol] |
   YPLUS [tol] | SPACING [tol]]
```

Five new tolerances have been defined for orthogonality, discontinuities, stretching, y^+, and spacing checks.

`DISCONTINUITIES` Sets the threshhold angle for determining discontinuity.

`ORTHOGONALITY` Sets the threshhold angle, calculated between a normal to the surface and the actual grid line, for determining undesirable orthogonality.

`SPACING` Sets the threshhold for determining undesirable spacing.

`STRETCHING` Sets the threshhold for determining undesirable stretching (ratio of lengths of adjacent cells in a particular direction).

`YPLUS` Sets the threshhold for determining undesirable wall spacing in terms of the reference value y^+.

7.2 List Commands

`LIST GRID` — List grid

```
LIST GRID [SCREEN|FILE filename [NEW|APPEND|OVERWRITE]] [PAUSE|NOPAUSE]
   [LINES lines]
```

`LIST NUMBER` — List grid info

```
LIST NUMBER [OF POINTS] [PAUSE|NOPAUSE] [LINES lines]
   [SCREEN|FILE filename [NEW|APPEND|OVERWRITE]] [ZONE n1 [THRU n2]]
```

This command writes the zone name and size for each zone in the selected range, plus the total number of points in that range. Output may be directed to a file.

7.3 List and Check Input/Output Control

The `CHECK` and `LIST` commands are also used to check and list boundary conditions. (See Section 9.) These commands have common input and output modes. Once a list or check input mode (`GRID`, `BOUNDARY`, or `FRINGE`) has been selected, it will remain selected in all the checking and listing commands until changed. The number of lines per page, pause mode, output mode are also common between commands.

`SCREEN|FILE` If `FILE` is specified then all data will be dumped into the file *filename* until a new file name is entered or the mode is set to `SCREEN`. Changing the mode from `FILE` to `SCREEN` causes the file *filename* to be closed. When `FILE` is specified, the `NEW|APPEND|OVERWRITE` option may be used when to create a new file, append to an existing file, or overwrite an existing file. If `NEW` is specified, *filename* must not be an existing file.

`PAUSE|NOPAUSE` Toggle determining whether the output pauses and prompts the user when a page of output has been written.

`LINES` Number of lines defining a page of output.

8 Boundary Conditions

Boundary condition specification/zone coupling is the second main function of GMAN. This involves specification of boundary conditions for all boundary points of each zone of a given computational grid.

GMAN allows the user to display any boundary, identify and specify or alter the boundary conditions on it, and perform zone coupling. Zone coupling involves locating points on a particular boundary which are "flow-through" points to another zone and setting interpolation factors relative to the other boundary. GMAN makes use of a marching scheme and a "line of sight" check when coupling two boundaries, using data from surrounding regions to locate each point on the surface.

In the boundary condition mode of GMAN, picking a grid point on a boundary surface causes the code to display the computational index and boundary condition for that point. An `IDENTIFY PNTS` option allows the placing of symbols on the grid at the locations of certain boundary condition types. Boundary conditions can be changed on all or part of a boundary surface, allowing multiple conditions for a single boundary. After selection of an area to be changed, the user may select any one of 12 different boundary conditions to be applied to the selected area. Bleed regions and reference areas for diffuser and inlet grids can also be defined in GMAN.

8.1 Non-Overlapping Boundary Conditions

`BOUNDARY` — Select a boundary

```
BOUNDARY [I1|IMAX|J1|JMAX|K1|KMAX]
```

The `BOUNDARY` command specifies to which boundary subsequent boundary modification commands will apply. Any boundary condition changes that have not yet been written are applied before selecting the new boundary. If a boundary is not specified then information about the currently selected boundary will be displayed.

Set a Boundary Condition

[*range* `[IS|ARE]`] *boundary_condition*

This command sets a boundary condition on the currently selected boundary. The optional *range* allows you to set a boundary condition over a specified region of the boundary. If *range* is not specified, the boundary condition is set at all the points on the boundary.

The *range* for an `I1` or `IMAX` boundary is specified as

`J`*m1* `K`*m2* `J`*n1* `K`*n2* `| ALL`

where *m1* and *m2* are the minimum values, and *n1* and *n2* are the maximum values, of the *j* and *k* indices defining the desired range. The parameters may be specified in any order. `J`*n1* and/or `K`*n2* may also be specified as `MAX`, meaning the maximum value in that direction. If `ALL` is specified, the range is defined as the entire boundary (i.e., the same as not specifying *range*).

Similarly, for a `J1` or `JMAX` boundary, *range* is specified as

```
Im1 Km2 In1 Kn2 | ALL
```

and for a `K1` or `KMAX` boundary, *range* is specified as

```
Im1 Jm2 In1 Jn2 | ALL
```

As an example, to set an arbitrary inflow boundary condition at the $i = 1$ boundary, over the range from $j = 1$ to 20 and $k = 5$ to 30, you could write

```
BOUNDARY I1
J1 K5 J20 K30 ARBITRARY INFLOW
```

The *boundary_condition* may be any one of the following:

`UNDEFINED`	Boundary conditions are generally initialized as undefined. They may occasionally need to be reset to undefined as part of the coupling process.
`REFLECTION`	Used for symmetry plane. Otherwise similar to inviscid wall.
`FREESTREAM`	Used for freestream outer boundaries.
`VISCOUS [WALL]`	Used for viscous (no-slip) walls.
`ARBITRARY [INFLOW]`	Used for flow entering the computational region. User specified by zone.
`OUTFLOW`	Used for specification of flow leaving the computational domain. User specified exit mass flow or constant static pressure.
`INVISCID [WALL]`	Used for inviscid (slip) wall.
`SELF [CLOSING]`	Used for periodic boundaries.
`SINGULAR AXIS`	Used for a singular axis on the x, y, or $z = 0$ plane.
`MIXED AXIS WALL`	Used for a zone face containing both singular axis and wall boundary conditions.
`COUPLED`	Used to specify that the currently selected boundary (or region of the boundary) is coupled to another boundary in a different zone. See below for the complete syntax, and for additional information about boundary coupling.
`BLEED [REGION]` *n*	Used for specifying a "bleed" region on the currently selected boundary. The input value n is a number that will be used with various Wind-US keywords in the input data (*.dat*) file to identify the region. Note that this applies not only to the `BLEED` keyword, but also to the `BLOW` and `MOVING WALL` keywords, which operate on regions identified as "bleed" regions in the common grid (*.cgd*) file.
`PINWHEEL [AXIS]`	Used for a singular axis that does not lie on the x, y, or $z = 0$ plane.
`FROZEN`	Used to freeze existing flow conditions on a boundary for use in subsequent runs.

For a coupled boundary, the complete syntax for *boundary_condition* is:

```
COUPLED [TO] ZONE izone BOUNDARY [I1|IMAX|J1|JMAX|K1|KMAX]
```

where *izone* is the number of the zone containing the coupled-to boundary. GMAN will only couple points on the currently selected boundary (i.e., the one specified using the `BOUNDARY` command) whose boundary condition is `UNDEFINED`. Thus, if a different boundary condition has been set in the coupled region, it must be changed to `UNDEFINED` before using the above command. GMAN will also only couple points that fall within the coupling tolerances. Other points will be left as `UNDEFINED`.

Note that the `COUPLED` boundary condition is not automatically reciprocal. You must also specify the inverse coupling. I.e., if the $j = 1$ boundary in zone 1 is coupled to the $j = j_{max}$ boundary in zone 2, the coupling would be specified as follows:

```
ZONE 1
BOUNDARY J1
COUPLED TO ZONE 2 BOUNDARY JMAX

ZONE 2
BOUNDARY JMAX
COUPLED TO ZONE 1 BOUNDARY J1
```

Instead of using the `COUPLED` command to manually couple all the coupled boundaries in the problem, the separate command

```
AUTO COUPLE
```

may be used to search the entire grid for points whose boundary condition is `UNDEFINED`, and automatically couple them to the appropriate boundary. Again, GMAN will only couple points that satisfy the coupling tolerances. The results should be checked to ensure that the boundaries were coupled correctly.

For periodically coupled boundaries, the `CONNECTED` command may be used to specify a displacement of the boundary, which will be applied by GMAN before the actual coupling is done. The syntax is:

```
CONNECTED [TO] ZONE izone BOUNDARY [I1|IMAX|J1|JMAX|K1|KMAX]
   {ROTATION [CENTER] xc yc zc [ANGLES] a1 a2 a3 | TRANSLATION [DELTAS] dx dy dz}
```

where *izone* is the number of the zone containing the boundary to which the currently selected boundary is periodically coupled. For boundaries connected via rotation, the center of rotation is specified by *xc*, *yc*, and *zc*, and the rotation vector about the x, y, and z directions is specified by the angles *a1*, *a2*, and *a3*. For boundaries connected via translation, the displacement is specified by *dx*, *dy*, and *dz*. The coordinate and displacement values are in the same units as the grid, and the angles are in degrees.

The capability supplied by the `CONNECTED` command is only applicable to entire boundaries. I.e., part of a boundary may not have "connect" coupling while the rest of the boundary has some other boundary condition (including "regular" coupling).

After all the boundary condition setting and coupling has been done, the commands described in Section 9 should be used to verify that the intended coupling was successful, and that the boundary conditions have been set correctly.

Change a Boundary Condition

```
boundary_condition_1 [POINTS] [IN range] [IS|ARE] boundary_condition_2
```

This command may be used to change a boundary condition on the currently selected boundary. The boundary condition at points having *boundary_condition_1* is changed to *boundary_condition_2*. The optional "`IN` *range*" (note that the word `IN` is required when a range is specified) allows you to apply the change only over a specified region of the boundary. If *range* is not specified, the change is applied over the entire boundary. The syntax for *range* is the same as described above, under Set a Boundary Condition.

Any of the boundary conditions listed above under Set a Boundary Condition may be specified for *boundary_condition_1* and *boundary_condition_2*, except for `COUPLED`. This command does not apply to coupled boundaries. To change an existing boundary condition to `COUPLED`, you must first change it to `UNDEFINED`, then use the syntax described under Set a Boundary Condition to set it to `COUPLED`.

`CPMODE` — Set coupling mode

```
CPMODE [BILINEAR FIXED|BILINEAR BEST|POINT MATCH]
```

`BILINEAR FIXED`	Couples only undefined points using bilinear interpolation mode.
`BILINEAR BEST`	Couples undefined points or previously-coupled points if the new coupling is closer than the original coupling. Bilinear interpolation mode is used for the coupling.
`POINT MATCH`	Assumes that the faces are point matched and will report the maximum deviation from an exact point match interface.

`DEBUG` — Print coupling diagnostics

```
DEBUG {ON|OFF}
```

The `DEBUG` command enables, disables, or displays the status of diagnostic printing during zone coupling operations. If neither `ON` nor `OFF` is specified, the current status of the debug flag is displayed. This command is helpful when zone coupling problems are encountered. *Warning: this command can generate a lot of output.*

`SHIFT` — Toggle boundary layer coupling mode

```
SHIFT {ON|OFF} {ALL|I1|IMAX|J1|JMAX|K1|KMAX}
```

This command turns boundary layer coupling mode on or off. Boundary layer coupling mode shifts the coupling to account for variations between the conceptual location to which a point near a wall should couple, and the physical relation of the point to the line segment or panel representing the adjacent wall.

`TOLERANCE` — Set coupling tolerances

The `TOLERANCE` commands establish the tolerances to be used when performing zone coupling operations. If the *tolerance* parameter is not specificed, the current value for the tolerance is displayed. The specified tolerance applies until the next `TOLERANCE` command that specifies a new value.

There are four coupling tolerances that may be set, as follows:

`ANGULAR [TOLERANCE] [`*tolerance*`|DEFAULT]`

Sets the tolerance for the angle between the normals of the faces which are being coupled.

`CONTAINMENT [TOLERANCE] [`*tolerance*`|DEFAULT]`

Sets the tolerance for coupling outside the strict bilinear interpolation of the cell face.

`NORMAL [TOLERANCE] [`*tolerance*`|DEFAULT]`

Sets the tolerance for coupling in the normal direction.

`MATCH [TOLERANCE] [`*tolerance*`|DEFAULT]`

Sets the tolerance for point match coupling.

`AREA` — Area commands

There are three area commands, as follows:

`DEFINE AREA` *n*

Define a numbered area (associated with a bleed region).

`CAPTURE AREA` *capture_area*

The `CAPTURE AREA` command specifies a new capture area for the zone most recently specified by a `ZONE` command. The area must be in square inches. If *capture_area* is not specified then the current capture area is displayed.

`AREA [IS]` *boundary_condition*

`ZDF` — ZDF coupling file commands

There are two commands related to the ZDF coupling file, as follows:

`ZDF [COUPLING] FILE` *filename*

Set the name of the ZDF coupling file.

`SAVE ZDF FILE` *filename*

Save the ZDF coupling file.

8.2 Overlapping Boundary Conditions

All overlapping boundaries and holes are defined by a label.

The overlapping grid capabilities have been improved and expanded. The surface hole cutting routine has been improved by using a new method which intersects a constant coordinate line from the test point with the surface. The user may specify any combinations of directions ($\pm x, \pm y, \pm z$) to use in the test, with the majority result used to determine the outcome.

8.2.1 Generating Holes

`CUTTER` — Define surfaces

`CUTTER {SURFACE NEW | [IN] ZONE` *zone_number ijkrange*`}`

Defines one or more surfaces to form a closed region for cutting holes. The parameter *ijkrange* has the same syntax as the *3d-range* parameter in the `SUBREGION` command (see p. 12).

`CUTTER INDEX` — Define index range

`CUTTER INDEX {NEW|`*ijkrange*`}`

Define an index range of the current grid to be converted to a hole. The parameter *ijkrange* has the same syntax as the *3d-range* parameter in the `SUBREGION` command (see p. 12).

`CUTTER PUNCH` — Define predetermined shape

`CUTTER PUNCH {2D|3D} {RECTANGLE|CIRCLE|ELLIPSE}` *coord_data*

Define parameters for a predetermined shape to use for cutting holes. An XYZ minmax box and a sphere/circle punch are available.

`SHOW CUTTER` — Show cutter definition

`SHOW CUTTER [DEFINITION]`

This command displays the current selections for either the hole or fringe generation mode. For hole generation the type of cutter and the cutter definition is displayed. For fringe generation the type is displayed, and if the type is INDEX or OUTER then the current selections are displayed. The extrapolation mode and cell mode are also displayed.

`GENERATE` — Generate a hole

```
GENERATE [range] HOLE LABEL {label_number|CURRENT|NEXT}
```

The parameter *range* has the same syntax as the *3d-range* parameter in the SUBREGION command (see p. 12).

`ERASE HOLE LABEL` — Erase a hole

```
ERASE HOLE LABEL {label_number|ALL}
```

8.2.2 Generating Fringe Boundaries

`FRINGE-MODE` — Set fringe type

```
FRINGE-MODE [SINGLE|DOUBLE]
```

Set the type of fringe to be generated.

`FRINGE-MODE HOLE` — Generate fringe

```
FRINGE-MODE HOLE
```

Used to generate a fringe that is the boundary of a hole.

`FRINGE-MODE INDEX` — Generate fringe

```
FRINGE-MODE INDEX [NEW|ijkrange]
```

Used to generate a fringe on an interior grid plane/subset. The parameter *ijkrange* has the same syntax as the *3d-range* parameter in the SUBREGION command (see p. 12).

`FRINGE-MODE OUTER` — Generate fringe

```
FRINGE-MODE OUTER [BOUNDARY] [ON|OFF] [I1] [IMAX] [J1] [JMAX] [K1] [KMAX]
```

Used to generate a fringe on boundary faces of a zone.

`SHOW FRINGE` — Display selections for fringe generation

```
SHOW FRINGE [MODES]
```

A `SHOW` command has been added to display the current selections for either the hole or fringe generation mode. For hole generation the type of cutter and the cutter definition is displayed. For fringe generation the type is displayed, if the type is `INDEX` or `OUTER` then the current selections are displayed. The extrapolation mode and cell mode are also displayed.

`GENERATE FRINGE` — Create a fringe

`GENERATE [`*range*`] FRINGE [BOUNDARY] LABEL {`*label_number*`|CURRENT|NEXT}`

This command creates a fringe with the designated label. The parameter *range* has the same syntax as the *3d-range* parameter in the `SUBREGION` command (see p. 12).

`ERASE FRINGE` — Erase a fringe

`ERASE FRINGE LABEL {`*label_number*`|ALL}`

This command erases the fringe with the designated label.

8.2.3 Selecting the Fringe Boundary

`BOUNDARY OLAP` — Select overlapping mode

```
BOUNDARY OLAP
```

This command selects the overlapping boundary condition mode.

8.2.4 Setting Fringe Boundary Conditions

`SET FRINGE` — Couple fringe to boundary face

`SET [RANGE] FRINGE [BOUNDARY] [CONDITION] [FOR] LABEL {`*label_number*`|CURRENT|NEXT}`
`[TYPE` *bnd_con_type*`] [TO]` *bnd_cond_type*

A new face coupling mode has been added for fringe boundaries which are part of a boundary face (fringe boundaries are identified by having the same label). The new mode couples the fringe boundary using the regular boundary coupling routines - thus all regular modes and methods are available. To invoke the mode you specify the boundary face you wish to couple to, if no face is specified then overlapping coupling mode is used.

The various boundary condition types are described in the Set a Boundary Condition section, starting on page 29.

8.2.5 Overlapping Modes

`CELL` — Location of coupling interpolation factors

```
CELL [VERTEX|CENTER]
```

The `CELL` command specifies whether the coupling interpolation factors are calculated at cell vertices or centers.

`CUT TEST DIRECTIONS` — Set hole test directions

```
CUT TEST DIRECTIONS [[[ON|OFF] {+-X +-Y +-Z|ALL}] ...]
```

This command sets the directions in which a point is tested to see if it lies inside a hole cutter.

`EXTRAPOLATE` — Allow/disallow extrapolation

```
EXTRAPOLATE [ON|OFF]
```

This command determines whether extrapolation outside a cell is permitted.

9 Boundary Condition Checking and Listing

9.1 Checking Boundary Conditions

CHECK BOUNDARY|CHECK FRINGE — Check a boundary or fringe

The CHECK command searches the entire file, identifying undefined boundary conditions and non-symmetric coupling, and prints the results on the screen or to a file. There are two possible formats. For regular boundaries, the format is:

```
CHECK BOUNDARY [SCREEN|FILE filename [NEW|APPEND|OVERWRITE]] [PAUSE|NOPAUSE]
   [LINES lines] [ZONE n1 [THRU n2] [bndy]]
```

and for fringe boundaries, the format is:

```
CHECK FRINGE [SCREEN|FILE filename [NEW|APPEND|OVERWRITE]] [PAUSE|NOPAUSE]
   [LINES lines] [ZONE n1 [THRU n2] [bndy]] [LABEL label_number|CURRENT|NEXT|ALL]
```

CKMODE BOUNDARY — Select boundary condition check modes

```
CKMODE BOUNDARY [ON|OFF] mode1 [mode2 [...]]
```

The fringe boundary condition check modes have been modified to have the same capabilities as the regular boundary checks. The wall check was modified to also check for coupling to a hole. The singular axis check will run only on fringe boundaries which are index planer. The fringe and regular boundary condition check output format was modified to conform to the new check syntax.

Each *mode* parameter is one of the following:

- `UNDEFINED` Checks for points with undefined boundary conditions.
- `BOUNDARY` Checks to make sure that all boundary points have valid boundary condition types specified.
- `COUPLED` Checks to make sure that all coupling pointers reference valid zones, faces, and index ranges.
- `TRIFACTORS` Checks the distance between a point and its coupled point (calculated by applying the interpolation factors to the adjacent zone).
- `WALLS` Checks for wall points that may be coupled to another zone face which is not set to wall. (Usually not recommended.)
- `SINGULAR` Checks to make sure that any points specfies as singular really are singular in at least one direction.
- `RATIOS` Compares the average area of cells around a point to the area of the cell face to which the point couples. This check looks for potentially unreasonable changes in resolution from zone to zone.

`SELF-CLOSING` Checks to make sure that zones with periodic boundary conditions match point-to-point on two opposite (periodic) boundaries.

`BL-COUPLED` Looks for wall points of one zone that couple to the interior of a cell in an adjacent zone. Wall points should normally couple to the wall of an adjacent zone unless there is a physical step in the geometry, or the wall ends.

`TOLERANCE BOUNDARY` — Specify tolerances

`TOLERANCE BOUNDARY [ALL] | CELL [`*tol*`] | NORMAL [`*tol*`] | XYZ [`*tol*`]]`

`ALL` List the current values of all tolerances.
`CELL`
`NORMAL`
`XYZ`

9.2 Listing Boundary Conditions

`LIST BOUNDARY|LIST FRINGE` — List boundary conditions

The `LIST BOUNDARY` command allows one to list the boundary conditions. It remembers the values for all of the parameters except *bnd_con_type* from previous `LIST BOUNDARY` commands.

There are two possible formats. For regular boundaries, the format is:

`LIST BOUNDARY [SCREEN|FILE` *filename* `[NEW|APPEND|OVERWRITE]] [PAUSE|NOPAUSE]`
`[LINES` *lines*`] [ZONE` *n1* `[THRU` *n2*`] [`*bndy*`]] [`*bnd_con_type*`]`

and for fringe boundaries, the format is:

`LIST FRINGE [SCREEN|FILE` *filename* `[NEW|APPEND|OVERWRITE]] [PAUSE|NOPAUSE]`
`[LINES` *lines*`] [LABEL` *label_number*`|CURRENT|NEXT|ALL]`
`[ZONE` *n1* `[THRU` *n2*`] [`*bndy*`]] [`*bnd_con_type*`]`

The *bnd_con_type* parameter must be one of the following:

range All points within the specified range are displayed.

boundary_condition All points of the specified type are displayed.

boundary_condition `[POINTS] [IN]` *range*
Only points of the selected type within the specified range are displayed.

The *boundary_condition* and *range* parameters are the same as described for the `BOUNDARY` command on page 29.

`LIST BOUNDARY REPORT` — List a boundary report

```
LIST BOUNDARY REPORT FILE filename [NEW|APPEND|OVERWRITE] [LINES lines]
```

9.3 List and Check Input/Output Control

The `CHECK` and `LIST` commands are also used to check and list grid information. (See Section 7.) These commands have common input and output modes.

Once a list or check input mode (`GRID`, `BOUNDARY`, or `FRINGE`) has been selected, it will remain selected in all the checking and listing commands until changed. The number of lines per page, pause mode, and output mode are also common between commands. A range of zones to list or check may be specified for `BOUNDARY` and `FRINGE` input modes.

`SCREEN|FILE` — If `FILE` is specified then all data will be dumped into the file *filename* until a new file name is entered or the mode is set to `SCREEN`. Changing the mode from `FILE` to `SCREEN` causes the file *filename* to be closed. When `FILE` is specified, the `NEW|APPEND|OVERWRITE` option may be used when to create a new file, append to an existing file, or overwrite an existing file. If `NEW` is specified, *filename* must not be an existing file.

`PAUSE|NOPAUSE` — Toggle determining whether the output pauses and prompts the user when a page of output has been written.

`LINES` — Number of lines defining a page of output.

`ZONE` — Selects the zone or range of zones to list or check.

10 Graphical User Interface Basics

This section describes the use of GMAN through its graphical user interface. The command line mode, which is useful on terminals without graphics capabilities, is described in Section 4.

10.1 Overview

GMAN's general screen layout is shown in Figure 1. The material in the various areas of the screen will change, depending on the operation being performed, but the general screen layout will remain the same.

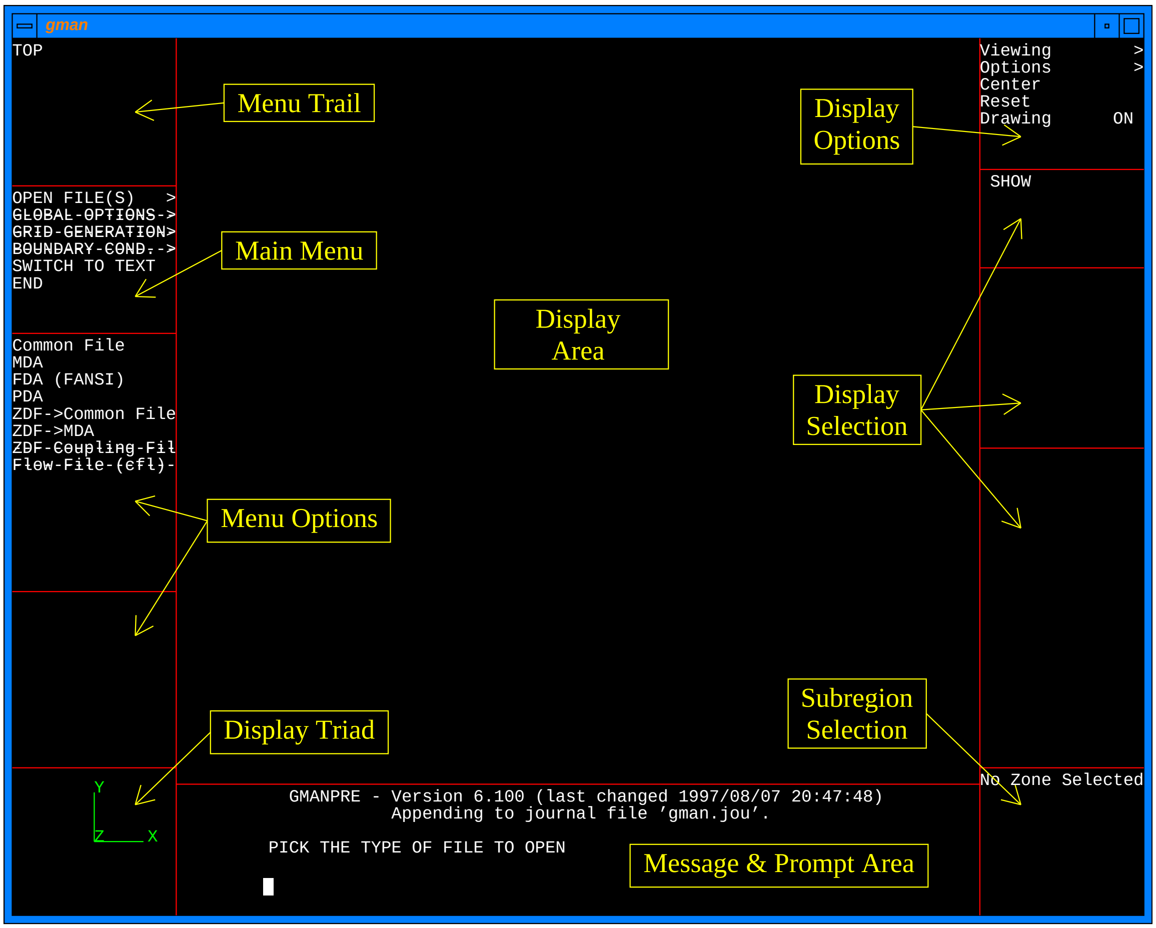

Figure 1: GMAN's screen layout.

The various areas of the screen, and their function are:

Display Area	Used to display the computational grid and/or boundary conditions.
Message and Prompt Area	Used to display GMAN messages and prompts, and for typing user input.

Menu Trail	History list of GMAN menus and sub-menus.
Main Menu	Current menu of GMAN operations to choose from.
Menu Options	Options for the currently selected GMAN menu.
Display Triad	Indicates the current orientation of the view in the Display Area.
Display Options	Menus and options affecting the viewpoint and appearance of the Display Area.
Display Selection	Menus and options relating to the elements of the grid to be shown in the Display Area, and/or written to a file.
Subregion Selection	Indicates the subregion selection within the current zone.

Input to GMAN is generated by either typing commands and/or responses into the Message and Prompt Area, or by picking "items" from the screen. An "item" might be an operation from a menu, a menu or display option, or a point or boundary in the Display Area.

Picking items from the screen involves pointing to the item using either a cursor or crosshairs, depending on the terminal type. Crosshairs will appear as two intersecting lines, one vertical and the other horizontal. The cursor or crosshair is controlled using the arrow keys on the keyboard or with the use of a mouse. The "pick" takes place when the user depresses the space bar or pushes the appropriate mouse button. (On certain terminals the mouse button must be programmed as a "space").

Menu items can also be picked by typing in a character string with enough characters to uniquely identify the item. For this reason, very short file names may cause problems. For example, if the user tries to open a file named *D* or *d*, the code cannot distinguish between the file name and, say, the `Drawing` item in the Display Options.

When the code requires input of an index range for grid generation or some other function, the entire grid can be selected by entering `ALL` at the prompt. The maximum index values can be referred to using `MAX` in place of the actual integer. Grid indices should be entered as integers (or `MAX`), separated by commas.

When a menu or option item is picked, the action corresponding to that item takes place. When an item in the Display Area is picked, GMAN will use the item picked as a response to the current prompt, if appropriate, or else will display information relevant to that point (i.e., the computational indices, xyz coordinates, boundary conditions, etc.).

In the various menu and option areas of the screen, the entry `==MORE==` at the top or bottom indicates that the list of choices is longer than the space available. Picking `==MORE==` will scroll the list, allowing the remaining choices to be seen.

The `>` symbol to the right of a menu item indicates that when this item is picked, the program will request additional input, either through another menu or list of options, or by prompting for input in the Message and Prompt Area.

Some menu items are flagged with an `*` at the left to indicate that they are the default or currently selected item.

Any menu item that appears with a line through it is not currently a valid selection, and cannot be selected. This helps guide the user by preventing selection of inappropriate menu items. Several items may not be selected until the user has performed other operations.

10.2 Display Options

The upper rightmost menu is used to control the Display Area. In the `Viewing` submenu (see Section 11.1.1, the user may set various `Axial views` or `X,Y,Z views`; `Rotate`, `Translate`, or `Scale` the display; `Reset view`, which resets the display to the default viewpoint; `Save view`, which saves the current viewpoint to a text file; or `Restore view`, which restores a previously-saved viewpoint.

Under the `Options` submenu (see Section 11.1.2, the user may choose the `Color` mode for the GMAN screen, define the type of `Coord. System` used, and choose the `Projection mode`.

The region of the grid shown in the Display Area may be set using the `Center` option, which allows the user to pick the center and outer boundary of the desired region, effectively zooming in or out on the selected region. `Reset` sets the viewpoint back to the default, which is based on the minimum and maximum coordinates of the first grid displayed.

The `Drawing` option toggles display redrawing on and off, with the current state displayed next to the menu item itself. This option may be helpful when using slower graphics devices. With `Drawing ON`, each time the user selects a menu item that affects the display, the Display Area is redrawn. This can be a slow process on slower graphics devices, especially if there is a lot to be drawn. By setting `Drawing OFF`, making changes to the display, and then setting `Drawing ON`, the user may save a lot of time in a GMAN session.

The first item in all the Display Options sub-menus is a menu title (e.g., `===VIEWING===`). Picking the title will return to the previously-chosen menu.

10.3 Display Selection

GMAN's most basic function is the interactive display of a computational mesh. This is accomplished using the `SHOW` selection in the Display Selection menu (see Section 11.3.1), located immediately below the Display Options area on the right side of the screen. Picking `SHOW`, then `SHOW SURFACES` from the next sub-menu, allows portions of the mesh to be displayed by selecting various zones and constant-index planes within each zone. In this way, a user may create a representative picture of the mesh used for a particular flowfield problem.

Once the grid is displayed, the user may place the cursor (or crosshairs) over the grid and pick a grid point, causing GMAN to display the zone number, computational indices and x-, y- and z-coordinates of the selected point. `NEXT PLANE` and `PREVIOUS PLANE` options allow the user to step through planes of a particular index, which is helpful in visualizing changes in the grid throughout the flowfield.

10.4 Main Menu

GMAN is structured such that the user can begin with the top level Main Menu and step down through various functions in the code in a logical manner. The Main Menu lists the major tasks that may be performed by GMAN. Picking the `OPEN FILE(S)` menu selection (see Section 11.2.1) produces a sub-menu in the Menu Options area listing the different file types recognized by GMAN. At the beginning of a GMAN session, the code automatically places the user in the `OPEN FILE(S)` sub-menu, since a file must be opened before the user can proceed.

Note that when reading a Zone Definition File (ZDF), only boundary points will be stored in the Mesh Direct Access (MDA) file that is created. The `GRID GENERATION / INTERIOR GRID / ALGEBRAIC` option (see Section 11.2.3) should be used to generate an initial algebraic interior grid.

After a file has been opened, and after displaying the grid using the SHOW selection from the Display Selection menu (see Section 10.3), the user will normally select GLOBAL OPTIONS, GRID GENERATION, or BOUNDARY COND. from the Main Menu, all of which will produce sub-menus containing more specific options. GLOBAL OPTIONS (Section 11.2.2) may be used to scale the grid, set the grid units, or read in a zone from another file. GRID GENERATION (Section 11.2.3) may be used to modify an existing zone, to generate or redistribute the grid points in a zone (given the boundary grid), or to adapt the grid to an existing flowfield. BOUNDARY COND. (Section 11.2.4) is used to set boundary condition types, and perform zonal coupling.

Selecting SWITCH TO TEXT from the Main Menu will switch the user back to command line mode. Selecting END from the Main Menu will produce a confirmation sub-menu, and a YES-TERMINATE selection from this sub-menu will end the current GMAN session.

10.5 Menu Trail

When an item from the Main Menu (or one of its sub-menus) is selected that brings up a sub-menu, the initial menu item is added to the Menu Trail, located just above the Main Menu. The Menu Trail thus becomes an ordered history list of selected menus. TOP is the first entry, followed by all the sub-menu selections in the order that they were picked. At any time, the user may jump to a particular menu in the path simply by selecting it from the Menu Trail. In some cases, the user may be prompted as to whether he/she wants to update any active files, particularly when boundary condition and/or grid generation changes have been made.

10.6 SGI-Specific Features

On SGI workstations, translation, rotation, and zooming of the Display Area can be done using the mouse in the manner of PLOT3D. Moving the mouse while holding the left mouse button down controls rotation. Moving the mouse while holding the center mouse button down controls zooming. Moving the mouse while holding the right mouse button down controls translation.

Note that when the mouse is moving, mouse buttons are associated with translation, rotation, and zooming. Therefore, when picking menu items or screen points, the mouse must be stationary.

11 Graphical User Interface Reference

11.1 Display Options

11.1.1 The Viewing Menu

`Viewing`	
`=== Viewing ===`	
`Axial views`	
`= AXIAL VIEWS =`	These views are oriented along a selected axis.
`+X AXIS`	
`-X AXIS`	
`+Y AXIS`	
`-Y AXIS`	
`+Z AXIS`	
`-Z AXIS`	
`X,Y,Z views`	
`= X,Y,Z VIEWS =`	These are trimetric views for each of eight quadrants.
`+X +Y +Z`	
`+X +Y -Z`	
`+X -Y +Z`	
`+X -Y -Z`	
`-X +Y +Z`	
`-X +Y -Z`	
`-X -Y +Z`	
`-X -Y -Z`	
`Rotate`	
`==== Rotate ====`	After selecting the axis to rotate about the user will be prompted for the angle to rotate about this axis.
`X axis`	
`Y axis`	
`Z axis`	
`Body based`	Rotations will be about the body X,Y,Z axes.
`Screen based`	Rotations will be about the screen horixontal and vertical axes.
`Translate`	
`== Translate ===`	After selecting the direction to translate, the user will be prompted for the distance to translate in that direction.
`X axis`	
`Y axis`	
`Z axis`	
`Scale`	The user will be prompted for a scale factor relative to the display size of the first grid plane displayed.
`Reset view`	Resets to the default view point and scale.

`Save view` — Allows the user to save the current view point information in a file.

`Restore view` — Allows the user to restore the view point information previously saved in a file.

11.1.2 The Options Menu

`Options`

```
=== OPTIONS ===

Color
   == Color Mode ==
   Full color
   White on black
   Black on white
```

- `Full color` — The default mode for GMAN operations.
- `White on black` — White drawing on black background, primarily for black and white plots.
- `Black on white` — Black drawing on white background, primarily for black and white plots.

```
Coord.  System
   == COORD SYS. ==
   STD. (+X +Y +Z)
   AERO (-Y +Z -X)
   CADD (+X +Z -Y)
   PROP (+Z +Y -X)
   USER DEFINED
```

`== COORD SYS. ==` — Sets the coordinate system to one of several common choices or a user defined system.

```
Projection mode
   == Projection ==
   Orthographic
   Perspective
   Viewing angle
```

`== Projection ==` — Determines whether or not perspective be used and if so, what viewing angle will be used for the perspective.

11.1.3 Miscellaneous

`Center` — Toggles between center/scale mode and standard picking mode.

`Reset` — Resets to the default view.

`Drawing ON/OFF`

11.2 Main Menu

11.2.1 The Open File(s) Menu

OPEN FILE(S)	A file must be opened before any other operation and the program will initially come up in this menu. The file can have been created on any machine, but must have the appropriate extension.

```
Common File
MDA
FDA (FANSI)
PDA
ZDF->Common File
ZDF->MDA
ZDF Coupling Fil
SLP Trace File
FLOW file (cfl)
```

11.2.2 The Global Options Menu

GLOBAL OPTIONS

SET GLOBL SCALE	
SET GRID UNITS	
ADD ZN FRM FILE	Appends a zone from another file to the current file.

11.2.3 The Grid Generation Menu

GRID GENERATION

PICK ZONE	The program will display a list of zones in the functional menu area.
MODIFY ZONE	
SAVE ZONE	Save the current zone at any time during the grid generation process. This writes over the corresponding zone in the file. It is not necessary for the user to remember to save the zone because if the user tries to exit from the grid generation section or read in another zone, the program prompts for whether the current zone should be saved.
RESTORE ZONE	Restore the last version of the zone that has been saved in the file. This overwrites the current version of the zone in memory.
LOAD FROM FILE	Replaces the current zone with a zone from another file.
TRANSLATE ZONE	Translate the physical coordinates of the entire zone. The user is prompted for DX, DY, and DZ.
ROTATE ZONE	Rotate the physical coordinates of the entire zone. The user is prompted for rotation axis and angle.

`SCALE ZONE`	Scale the physical coordinates of the entire zone. The user is prompted for individual scale factors for each coordinate direction, SX, SY, and SZ.
`RE-MAP ZONE`	Changes the I,J,K and/or X,Y,Z mapping of a zone.
`AXI-SYM SETUP`	
`NO AXI-SYM`	
`X-Y PLANE`	
`Y-Z PLANE`	
`X-Z PLANE`	
`CURRENT SETTING`	
`INTERIOR GRID`	The interior grid option operates only on interior points. The boundaries are not changed. In addition, the interior grid generation functions operate on the currently defined subset. The default subset is the entire zone, but can be changed by picking on the `Subregion` definition in the lower-right corner of the screen.
`ALGEBRAIC`	Algebraic interpolation may be performed over the entire zone or on a user-defined subregion of the zone.
`LINEAR IN I`	Perform straight line interpolation between opposite boundaries with spacing distributions interpolated from the boundaries.
`LINEAR IN J`	
`LINEAR IN K`	
`JK TRANSFINITE`	Perform transfinite interpolation in the designated plane for each such plane in selected region.
`IK TRANSFINITE`	
`IJ TRANSFINITE`	
`IJK TRANSFINITE`	Perform 3D transfinite interpolation from the boundaries of the specified region.
`= STACK MODE =`	
`FULL 3D`	
`I PLANES`	
`J PLANES`	
`K PLANES`	
`ELLIPTIC`	
`=FORCE FUN MEN=`	
`== 2D AND 3D ==`	
`LAPLACE ( = 0 )`	No forcing function.
`SMOOTH`	
`== 3D ONLY!! ==`	
`ARC LENGTH`	Forcing based on the arc length distribution of the initial grid.
`ARC + CURVATURE`	Forcing based on arc length and curvature of the initial grid.
`INTERP, ARC`	Forcing interpolated from the boundaries based on boundary arc length distributions.
`INTERP, ARC+CURV`	Forcing interpolated from the boundaries based on boundary arc length and curvature.
`GRAPE`	
`== 2D ONLY!! ==`	
`THOMAS MIDDLECF`	

```
    SORENSON

    RUN ELLIPTIC          Begin the elliptic equation smoothing.
    SHOW PARMETERS
    SET OMEGA
    SET SOLVER
       =SOLVER MENU=
       POINT SOR
       X LINE SOR
       Y LINE SOR
       Z LINE SOR
       ADI LINE SOR
       MULTIGRID POINT
       X LINE MULTI
       Y LINE MULTI
       Z LINE MULTI
       ADI LINE MULTI
    SET INTRP TYP
       =INTRP TYP MEN=
       INDEX
       SEMI ARC LENGTH
       ARC LENGTH
       = STACK MODE =
       FULL 3D
       I PLANES
       J PLANES
       K PLANES

 DISTRIBUTE               Redistribute the grid along lines of varying I, J, or K coordinate,
                          using the initial grid distribution.
    REDISTRIBUTE I
    REDISTRIBUTE J
    REDISTRIBUTE K
    = STACK MODE =
    FULL 3D
    I PLANES
    J PLANES
    K PLANES

ADAPT GRID

 RUN ADAPTATION
 OPEN FLOWFILE
    Common File
    FDA (NASTD)
    FDA (FANSI)
    PDA
 FORCING FUNC
    DENSITY
    MACH NUMBER
```

```
      TOTAL PRESSURE
      TOTAL TEMP.
      STATIC PRESSURE
      STATIC TEMP.
   METHOD
      FULL 3D
      3D ON I-PLANES
      3D ON J-PLANES
      3D ON K-PLANES
      1D IN I DIREC.
      1D IN J DIREC.
      1D IN K DIREC.
      Y-PLUS
   WEIGHTS
      CONSTANT WEIGHT
      1st DERIV. WT.
      2nd DERIV. WT.
   BOUNDARIES
      I Start ON/OFF
      I End   ON/OFF
      J Start ON/OFF
      J End   ON/OFF
      K Start ON/OFF
      K End   ON/OFF
```

11.2.4 The Boundary Condition Menu

BOUNDARY COND.

PICK ZONE/BNDY	Select zone and boundary for changes.
IDENTIFY PNTS.	Mark boundary conditions of particular types with keyboard symbols.
MODIFY BNDY	
CHANGE ALL	Change all points in the selected area.
UNDEFINED	
REFLECTION	
INFLOW OUTFLOW	
VISCOUS WALL	
FREESTREAM OUTF	
DIFFUSER INFLOW	
ARBITRARY INFLO	
CONFINED OUTFLO	
INVISCID WALL	
SELF CLOSING	
SINGULAR AXIS	
MIXED AXIS WALL	
BLEED	

`    PINWHEEL AXIS`	
`    FROZEN`	
`    CHIMERA BOUNDAR`	
`    TRAILING EDGE`	Boundary condition for FLO67.
`    DISCONTINUOUS`	Boundary condition for FLO67.
`  CHANGE BY TYPE`	Change only points of a certain boundary condition within the selected area.
`  COUPLE`	
`    COUPLE`	Perform zone coupling.
`    CONNECT`	
`    SHOW OTHER BNDY`	
`    HIDE OTHER BNDY`	
`    SEL OTHER BND`	
`    SET TOLERANCE`	
`      RESET ALL TOLER`	
`      CONTAINMENT TOL`	Set edge-of-cell tolerance.
`      NORMAL TOLER.`	Set normal distance tolerance.
`      ANGULAR TOLER.`	Set normal vector angular tolerance.
`      POINT MATCH TOL`	
`    SET COUP MODE`	
`      POINT MTCH ON/OFF`	
`      BILIN FIX ON/OFF`	
`      BILIN BEST ON/OFF`	
`      BNDY LAYER ON/OFF`	
`  IDENTIFY PNTS`	Mark selected boundary condition types with keyboard symbols.
`AUTO COUPLE`	
`  RUN AUTO COUP`	
`  SET AUTO MODE`	
`    - MODE IS -`	
`    COUPLE FACE`	
`    COUPLE OVERLAPP`	
`    RECOUPLE FACE`	
`    RECOUPLE OVERLA`	
`  SET REPT FILE`	
`    - FILE IS -`	
`  SELECT ZONES`	
`    - ZONE IS -`	
`  ** SET TOLER **`	
`    RESET ALL TOLER`	
`    CONTAINMENT TOL`	
`    NORMAL TOLER.`	
`    ANGULAR TOLER.`	
`CAPTURE AREA`	
`  MANUAL ENTRY`	Set capture (reference) area for this zone.
`  COPY FROM ZONE`	Copy area from another zone.
`  RESET`	Sets area back to its original value (within the current session).

- `END` — Return to boundary condition menu.

- `SAVE TMP ZDF` — Save the temporary ZDF (contains only boundary faces) to speed up future coupling.

11.2.5 Miscellaneous

- `SWITCH TO TEXT` — Switch to command line mode.
- `END`
  - `YES-TERMINATE` — Terminate the current GMAN session.
  - `NO-DO NOT END` — Return to `TOP` menu.

11.3 Display Selection

11.3.1 The Show Menu

`SHOW`

- `SHOW SURFACES`
  - `PICK ZONE` — Select a zone for grid display.
  - `PICK I-PLANE` — Display the I-plane menu for selection.
  - `PICK J-PLANE` — Display the J-plane menu for selection.
  - `PICK K-PLANE` — Display the K-plane menu for selection.
  - `NEXT PLANE` — Display the next plane in the direction of the currently selected plane type (I,J,K).
  - `PREVIOUS PLANE` — Display the previous plane in the direction of the currently selected plane type.
  - `TOGGLE DISPLAY`
  - `CLEAR DISPLAY`
  - `BLANKING ON/OFF`
    - `SHOW GRID ON/OFF`
    - `SHOW HOLES OFF/ON`

- `SHOW VOLUMES`
  - `PICK ZONE`
  - `DEFINE VOLUME`
  - `DISPLAY VOLUME`
  - `DISPLAY EDGES`
  - `TOGGLE DISPLAY`
  - `CLEAR DISPLAY`
  - `BLANKING ON/OFF`
    - `SHOW GRID ON/OFF`
    - `SHOW HOLES OFF/ON`

- `SHOW TRACE`
  - `PICK ZONE`
  - `GENERATE TRACE`
    - `START TRACE`

```
      SET RAKE RANGE
      SET RAKE INCR
      SET TRACE DIR
      SET TRACE RESTR
      SET PARTCLE SIZ
      SET TRACE LIMIT
    OPEN SLP FILE
    SAVE GROUP
    DELETE GROUP
    SELECT GROUP
    DISPLAY TRACE
      ALL ON
      ALL OFF
    SET TRACE COLR
      WHITE
      RED
      GREEN
      BLUE
      CYAN
      MEGENTA
      YELLOW
    CLEAR ALL TRACE
    CLEAR DISPLAY

  SET DISPLY MODE
    WIREFRAME
    SHADED
    SHADED + GRID
    SOLID PANELS
    HIDDEN LINE
```

11.3.2 The List Menu

```
LIST

  LIST OPTIONS
    -LIST OPTIONS-
    GRID
    GRID HOLES
    BOUNDARY CONDS.      (Non-OLAP boundaries only)
    FRINGE BOUNDARY      (OLAP boundaries only)
    NUMBER OF PNTS
    BNDY CND REPORT
    FRINGE REPORT
    LABELS               (OLAP boundaries only)
    LABEL REPORT         (OLAP boundaries only)

  PICK ZONE
```

```
   PICK BOUNDARY

   OUTPUT OPTIONS
      =OUTPUT OPTIONS=
      WRITE TO SCREEN
      WRITE TO FILE
      LINES/PAGE
      PAUSES ON/OFF
```

11.3.3 The Check Menu

```
CHECK

   CHECK GRID
      -GRID QUALITY-
      RUN CHECKS
      PICK ZONE
      CHECK MODES
         -CHECK MODES-
         ALL ON
         ALL OFF
         AREAS & SIDES
         ZERO AREAS/VOLUMS ON/OFF
         NEG. AREAS/VOLUMS ON/OFF
         NEG. JACBN. ON/OFF
         CLPSED SIDE ON/OFF
         CROSSD SIDE ON/OFF
         ORTHOGONALI ON/OFF
         DISCNTNUITY ON/OFF
         STRETCHING ON/OFF
         ASPCT RATIO ON/OFF
         SPACING ON/OFF
         Y-PLUS CHCK ON/OFF
         SOLUTION CK ON/OFF
      OUTPUT MODE
         -OUTPUT MODES-
         NODE BY NODE
         PLANE BY PLANE
         WORST PLANES
         REGIONS
         SUMMARY
         INTERACTIVE
      OUTPUT OPTIONS
         =OUTPUT OPTIONS=
         WRITE TO SCREEN
         WRITE TO FILE
         LINES/PAGE
         PAUSES ON/OFF
      SET TOLERANCES
```

```
        --TOLERANCES--
        SHOW TOLERANCES
        ORTHOGONALITY
        DISCONTINUITY
        STRETCHING
        SPACING
        Y-PLUS
    SET REF VALUES
        --REF VALUES--
        REF LENGTH
        REYNOLDS NUMBER
    SET QUAL DISPY
        -QUAL DISPLY-
        FAILURES(+) OFF/ON
        -COLOR CONTUR-
        AREAS OFF/ON
        JACOBIAN OFF/ON
        SIDES OFF/ON
        ORTHOGONALI OFF/ON
        DISCNTNUITY OFF/ON
        STRETCHING OFF/ON
        ASPCT RATIO OFF/ON
        SPACING OFF/ON
        YPLUS CHCK OFF/ON
        SOLUTION OFF/ON
        -SOLUTION VAR-
        "CFPOST variable"
        -CONTOUR DIR-
        I DIRECTION ON/OFF
        J DIRECTION OFF/ON
        K DIRECTION OFF/ON
        -CONTOUR LEV-
        SET CONT LEVELS

CHECK BOUNDARY
    -BND CND CHKS-
    RUN BNDY CHKS                   (Non-OLAP boundaries only)
    RUN FRNG CHKS                   (OLAP boundaries only)
    PICK ZONE
    PICK BOUNDARY
    CHECK MODES
        -CHECK MODES-
        ALL ON
        ALL OFF
        QUICK
        UNDEFINED ON/OFF
        BOUNDARY ON/OFF
        COUPLED ON/OFF
        TRIFACTORS ON/OFF
        WALLS ON/OFF
```

```
      SINGULAR ON/OFF
      RATIOS ON/OFF
      HOLES ON/OFF
      SELF-CLOSIN ON/OFF
      BL-COUPLED ON/OFF
   OUTPUT MODE
      -OUTPUT MODES-
      FULL
      SUMMARY
      FAILURES ONLY
   OUTPUT OPTIONS
      =OUTPUT OPTIONS=
      WRITE TO SCREEN
      WRITE TO FILE
      LINES/PAGE
      PAUSES ON/OFF
   TOLERANCES
      SHOW TOLS
      XYZ
      NORMAL
      CELL
      RATIO
   SET QUAL DISPY
      -QUAL DISPLY-
      FAILURES(+) OFF/ON
      -COLOR CONTUR-
      TRIFACTORS OFF/ON
      RATIOS OFF/ON
      -CONTOUR DIR-
      I/X DIRECTN ON/OFF
      J/Y DIRECTN OFF/ON
      K/Z DIRECTN OFF/ON
      -CONTOUR LEV-
      SET CONT LEVELS
```