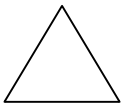
VII. Background Grids Panel

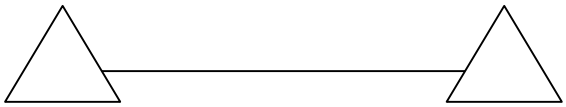
- Source Placement and Spacing Requirements

  Activate *Background Grids* panel by pressing [Background Grids] button. This will automatically activate the *Points/Curves* panel, if it is not already activated. The *Points/Curves* panel must be activated to place sources on the geometry and in 3-D space.

Types of Sources:

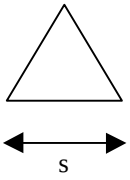


nodal source

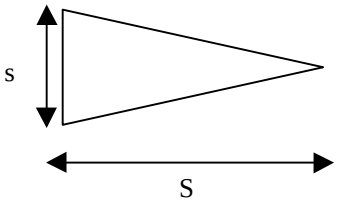


linear source

Spacing Definition



s – spacing defined for triangulation (no stretching)

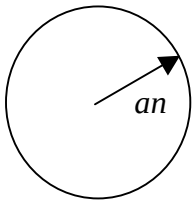


S- spacing defined for stretched side

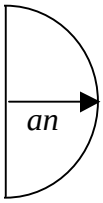
Source Propagation Controls

Contribution from: S. Pirzadeh

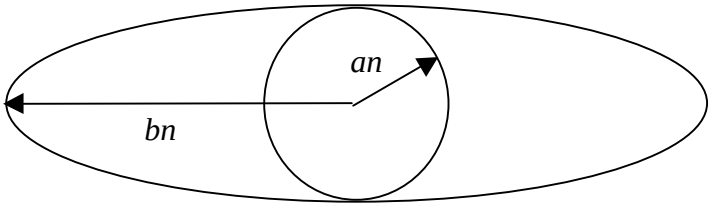
an – source propagation proportionality factor which is equivalent to a sphere of influence.
bn – source propagation proportionality factor equivalent to an ellipsoidal influence
alpha – direction along which source propagation is concentrated.



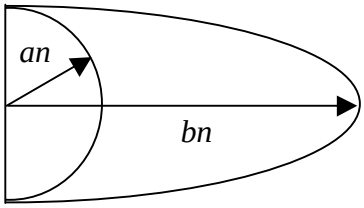
$an > 0 \quad bn = 0 \quad alpha = 0$



$an > 0 \quad bn = 0 \quad alpha = 1$



$an > 0 \quad bn > 0 \quad alpha = 0$



$an > 0 \quad bn > 0 \quad alpha = 1$

Placement of Sources:

a. Begin by pressing [Next Source] button.
b. Select a [Nodal Source] or [Linear Source].
c. Use [m] key to move next source to the desired location.

A linear source contains source elements located at each end. The first source is automatically activated when a new source is created. The [m] key is utilized to move the (1) source element. To move the (2) source element, place the mouse over the [Point] field and press the rightmost mouse button to activate the pull-down menu. Drag to the desired point, (2) to activate the second source element. The point source will turn red, indicating that it is the active point source. The active source (linear or nodal) is highlighted in magenta. The inactive sources are highlighted in yellow. Once active, the source point can be moved to the desired location via the [m] key.

- Changing source propagation attributes:

Each source element on a linear source may contain different propagation attributes. One must toggle between [Points] to show each of the propagation attributes of each of the source elements.

- Direction of source propagation, U:

Imposing a direction along the source propagation is controlled by turning on the [alpha] field or by designating a [bn] value. The direction of the U vector is manipulated by:

a. Placing the mouse over the [Point] field and select U from the drop down menu. (alpha = 1 or bn > 0)
b. Then use the [XYZ] sliders to manipulate the direction and orientation of the U vector.

Example: Place a linear source along the fuselage.

- Press [Next Source] button.
- Activate [Linear Source] button.
- Use [m] key to place source at the tip of the fuselage.
- Change spacing by typing in a value into the [s] field.
- Change the source propagation strength: [an], [bn], and [alpha] field, appropriately.
- If alpha = 1 or bn > 0, adjust the direction vector, U.
- Move the mouse to [Point] field and select source point 2. Use the [m] key to move the source to the tip of the tail. Change the spacing requirements and source propagation controls of this source.
- Place other source elements, accordingly. The location of a source element is chosen where a well-controlled distribution of grid points is desired. See Ref. 4 for details.

Cell Stretching

In viscous applications where the number of point may be too large for current computational resources, stretching may be imposed to save on the number of points. Stretching along some desired coordinate direction is useful in reducing the amount of points in a given volume mesh. Normally, the stretching direction is in a region of low flow gradients.

To Activate Stretching:

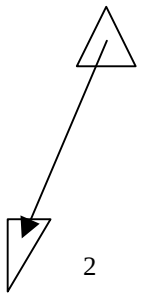
a. Depress the [Global] button to activate the *Global* panel.
b. Turn on stretching by typing (1) in the [Stretching] field.
c. Turn off stretching by typing (0) in the [Stretching] field.
d. Press [Stow] button to deactivate the panel.

To set stretching attributes on each source:

Toggle [Stretching] button to get [Stretching 1-2] or [Stretching 2-1]. The direction (1-2) or (2-1) depends on which direction is desired. Once the appropriate stretching direction is achieved, all the sources must be oriented in the same fashion. Arrows must point in the same direction, to be consistent. Not all sources must have stretching, but all that do must be oriented the same way.

Examples:

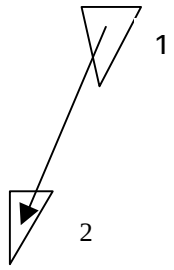
Linear Source with only one point source having stretching:



[Stretching 1-2]

Source 1	Source 2
s = value1	s=value1
S = value1	S=value2

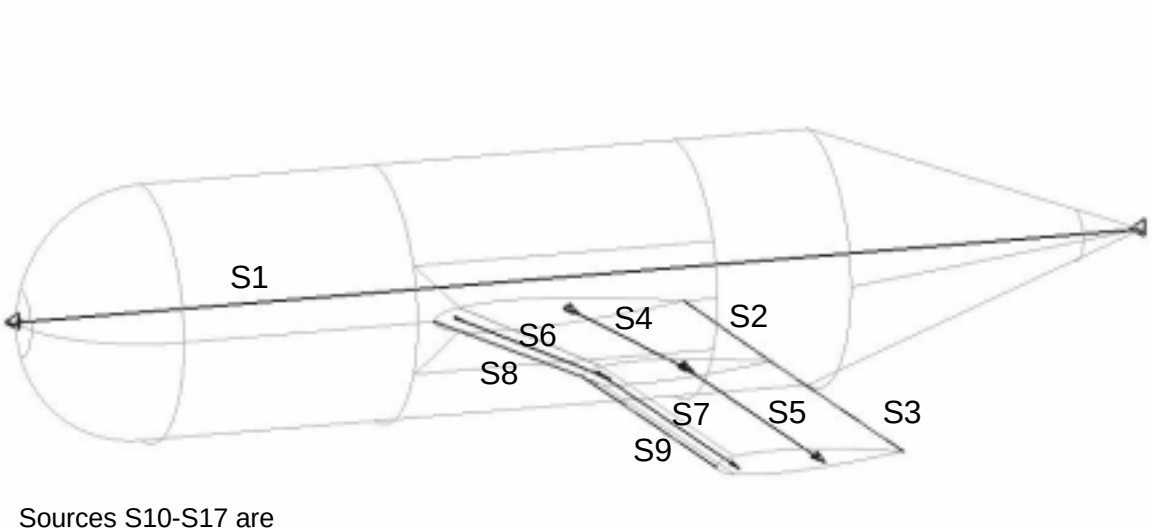
Linear Source with two point sources having stretching:



[Stretching 1-2]

Source 1	Source 2
s = value1	s=value2
S = value2	S=value3

Shown below is a suggested arrangement of linear(S1-S9) and nodal(S10-S17) sources for the class example:



Note: Sources S10-S17 are located on the eight corners of the bounding box:

$-10 \le x \le 13$

$-5 \le y \le 5$

$0 \le z \le 5$

Source Name	s	S	a_n
S1	0.050	0.000	3.0
S2	0.005	0.005	0.2
S3	0.005	0.005	0.2
S4	0.025	0.100	2.0
S5	0.025	0.100	2.0
S6	0.005	0.100	1.0
S7	0.010	0.010	1.0
S8	0.005	0.100	0.1
S9	0.005	0.005	0.1
S10-S17	1.500	0.000	2.1

Generating Meshes for Viscous Applications

Advancing Layers Method (VGRID System)

a. Activate the *Global* panel by depressing the [Global] button on the *Background Grids* panel.
b. Turn viscous on by placing (1) in the [Viscous] field.
c. Turn viscous off by placing (0) in the [Viscous] field.
d. Fill in the remaining attributes necessary to achieve the desired spacing.

Global Panel Attributes

ni - the number of points in the Cartesian background grid.
nlayer - upper bound on the number of viscous layers
delta1 - spacing off the surface; first layer of the viscous layer
rate1 -constant of rate stretching (viscous layers)
rate2 -constant of rate stretching (viscous layers)

Grid Spacing Equation:

The following equation is a simple stretching function used to distribute grid points along surface vectors so to establish the normal spacing height for the *n*th layer. Users are prompted to designate an initial first layer spacing.

$$\Delta_n = \Delta_0 \left[1 + r_1 (1 + r_2)^{n-1}\right]^{n-1}$$

Δ_0 - initial spacing

r_1 - geometric rate of expansion

r_2 - rate of expansion which slows down the rate of cells opening up, then accelerates the rate of expansion

Geometric Stretching:

To achieve geometric stretching, the value of rate1 is specified and the value of rate2 is set to 0.0.

Example:

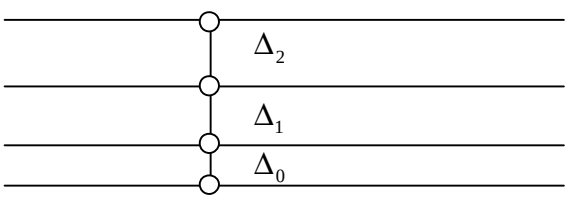
$r_1 = 0.20 \quad r_2 = 0.00$

$$\frac{\Delta_n}{\Delta_{n-1}} = (1 + r_1) = C$$

$$\frac{\Delta_n}{\Delta_{n-1}} = 1.2$$

$$\Delta_n = 1.2\Delta_{n-1} \qquad \Delta_1 = 1.2\Delta_0 \qquad \Delta_2 = 1.2\Delta_1$$

Geometric Stretching of the Viscous Layers:



Δ_0 is specified by the user.

Values for rate1 and rate2 is dependent on the type of numerical scheme used for the flow equations and on the user's estimate on the boundary layer thickness. Figure 7 gives a descriptive representation where the advancing layers and advancing front methods are applied.

Figures 8-9 show the effects of the stretching ratio by varying the values of rate1 and rate2. Figure 8 depicts the actual width between cells and Figure 9 depicts the actual physical distribution from some prescribed baseline definition.

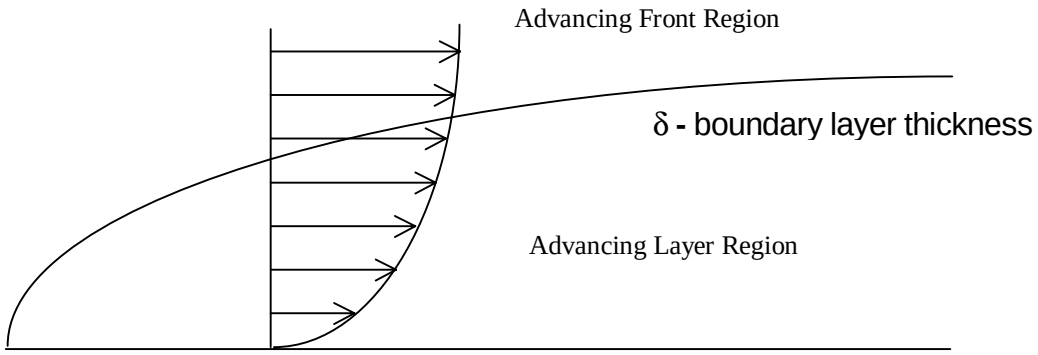


Figure 7. Generic flow description where the Advancing Layer Method and the Advancing Front Method are typically applied.

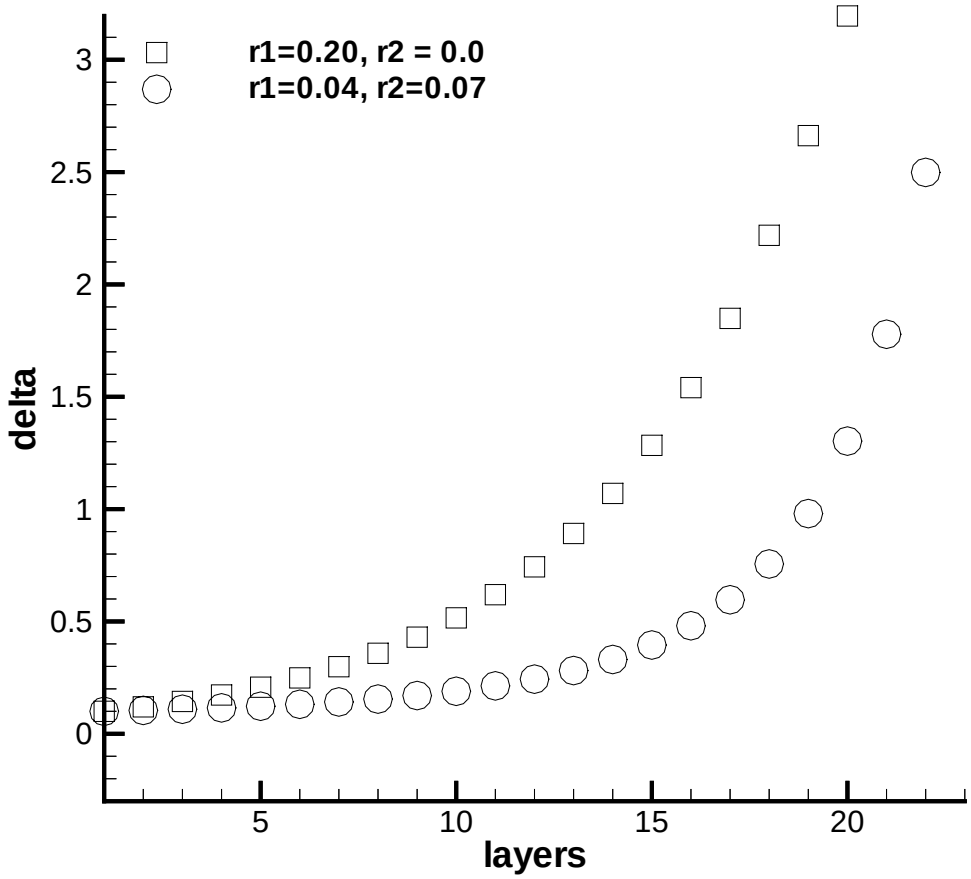


Figure 8. Plot of delta values verses the number of layers in the grid.

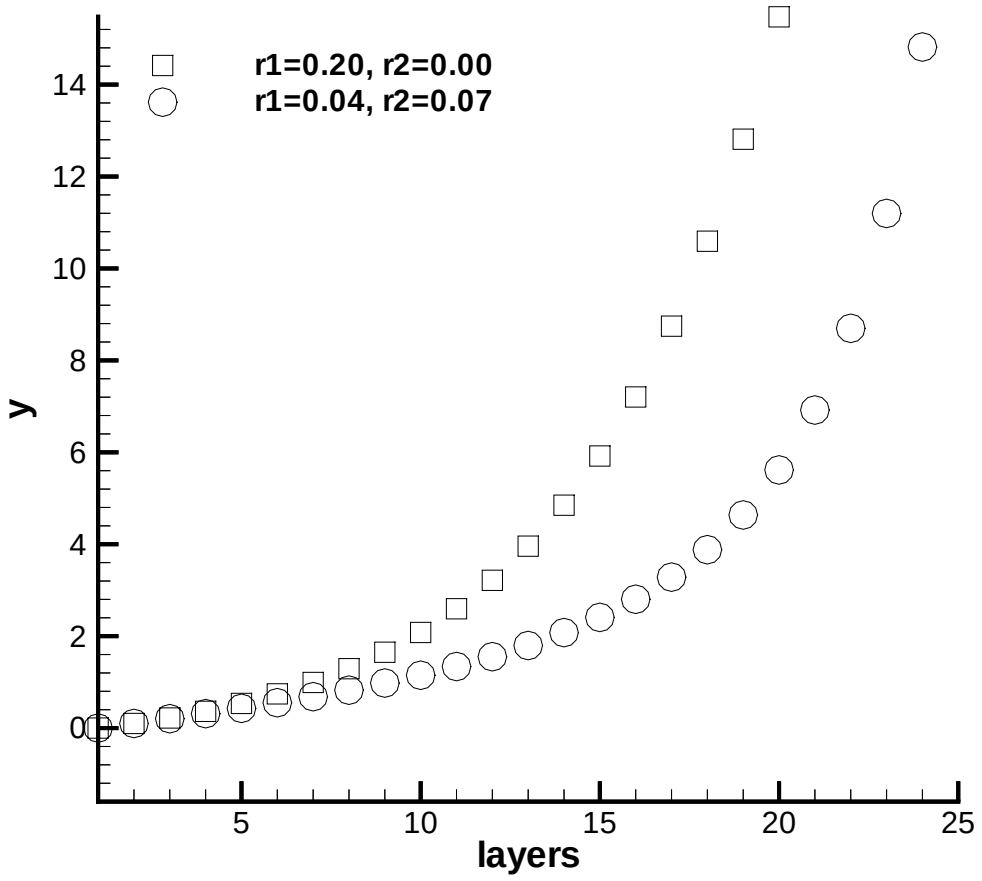


Figure 9. Plot of Distribution verses the number of layers in the grid.

VIII. VGRID/FELISA Input Files

- To generate the input files to the grid generators, the following items must be completed:

    a. Curve Representation
    b. Patch Representation
    c. Background Source Distribution

- Writing the Input Files

    a. Activate *Input/Output* panel by pressing the [I/O] button.
    b. Type in the project name in the [Project Name] field.

The name typed in the [Project Name] field will be the name used to append the file extensions which are used by the individual grid generation systems, such as:

VGRID System Files:

- .d3m
- .bc
- .mapbc

FELISA System Files:

- .dat
- .bac
- .bco

Example: Writing out the grid generation files for VGRID.

[Project Name] : Class
[File Type]: ASCII
[File Format]: d3m (VGRID)
Press [WRITE] button.

GridTool will write a Class.d3m file.

GridTool will automatically write and update the support files, such as Class.bc and Class.mapbc when a new .d3m file is created or overwritten.

IX. Unstructured Grid Panel

The unstructured grid panel is used to project the triangulation onto the original CAD definition.

- Projecting the Front

In the VGRID system, the generated triangulation does not generally lie on the original surface definition. In order to guarantee that the points are on the surface, the triangulation of each patch must be projected on to the original surface definition. Before utilizing any projection routines, it is advised that a copy of the original unprojected front (.front and .cogsg) files be saved under a different name or directory. In case there are problems in the volume grid generation due to the projections, a completely new surface mesh does not have to be generated. Instead the original unprojected front is read into GridTool as an Front(Update) file and the original projection can be undone.

- Saving unprojected front

  a. Save Class.front and Class.cogsg to another file name or directory.
  b. To undo the projection of an already projected front file (i.e. the front file has been overwritten), read in the unprojected front file saved earlier, as a Front (Update). Then proceed with undoing the projection in the *Unstructured Grid Panel*.

- To project the front

  a. Read Class.front file from I/O panel.

     [File Format]: ASCII
     [File Type] : Front (VGRID)

  b. Activate *Unstructured Grid* panel by pressing the [Unstructured Grid] button in the GridTool *Main* panel.
  c. Press the [Front] button.
  d. Display corresponding patch triangulation individually, by family, or all.

     [Active P on] – displays the triangulation of the currently active patch.

     [Turn On/Off Family] – displays the triangulation of all the patches with the current family designated in the Misc section of the *Patches* panel.

     [Turn On/Off All] – toggles triangulation display for all the patches.

  e. Display all corresponding surfaces, from the *Surface Attributes* panel.
  f. Press [Project] button.

g. Decide from the query panel whether to proceed with the projection, [yes or no].
h. Press [Undo] button to undo the projection if necessary.

To view the triangulation and check for bad projections it is helpful to hide the original surface defintion. The projection results are displayed in the [dmax] field. The [dmax] field displays the maximum distance a point on the triangulation was projected to the given surface definition. The [Distance] field is used to let the user input a value and see what points in the triangulation have move by at least that value. This is useful for seeing which points have moved the largest distance and if this is indeed the logical projection.

The [Auto] button is used to turn on/off the associated surface for each patch triangulation automatically. If a surface triangulation is displayed on a particular patch, the associated surface is also displayed. This button is useful when working with the FELISA grid generation system. The FELISA grid generation system requires that all patches have an associated surface. This is designated through the [Accept Surfaces] button on the *Patches* panel.

To use this feature, every patch must have a surface associated to it. To associate a patch to a surface:

a. activate the patch
b. display the surface (one at a time)
c. press the [Accept Surfaces] button.

- Rewriting New Front File

Once all the patches have been projected, a new Class.front file must be generated.

[File Name] : Class.front
[File Format]: ASCII
[File Type]: Front (VGRID)
Press [WRITE] button.

This overwrites the original front file. Remember to save the unprojected front before proceeding with this step.

- Proceeding with Volume Grid Generation

To proceed with the volume grid generation, the following files must be present prior to executing VGRID:

Class.d3m - main input file to VGRID and POSTGRID
Class.front - connectivity and coordinates of current triangulation
Class.bc - surface triangle, patch number, node number
Class.mapbc - patch by patch boundary conditions
Class.cogsg - coordinates and connectivities of the surface/volume triangulation