

harpoon

User Guide for Version 4.4

Table of Contents

Table of Contents.....	2
1 Overview	5
2 Installation	6
3 GUI Overview	7
3.1 Icons	7
Display and Feature Icon Description.....	7
4 Main Menu	8
4.1 File	8
Load	8
Import	8
Append.....	8
Replace	8
Export	8
Save	8
4.2 Geometry	8
Sweep Select.....	8
Separate	8
Mirror.....	8
Transform	9
Delete Parts	9
Merge Parts	9
Merge Similar Parts	9
Rename Parts	9
Select Part	9
4.3 Lines	9
Sweep Select.....	9
Separate	9
Delete Lines	10
Merge Lines	10
Rename Line	10
4.4 Volumes.....	10
Delete Volume(s).....	10
Merge Volumes	10
Rename Volume	10
Group Volumes.....	10
Reduce Hexa count	10
4.5 Surface	10
Sweep Selected Surfaces	10
Group Surfaces	10
Rename Surface	10
Copy to Geometry.....	10
Separate by Region.....	10
Clean Surface Mesh.....	10
Delete Surface Holes	10
Delete Baffles	10
4.6 Mesh Metrics	11
Mesh Stats.....	11
Mesh Quality	11
4.7 Display	11
4.8 Preferences.....	12
Geometry Preferences.....	12
CAD Preferences	12
Boundary Layer Preferences.....	12
Meshing Preferences.....	13
Export Preferences.....	14
Run-Time Preferences	14
GUI Preferences.....	15
Edit Part Attributes	15
Edit Background Colour	15
5 Features.....	16
5.1 STL Modification	16
Holes.....	16
Add Tris/Lines	16

Delete Tris	17
Move Node	17
5.2 Refinement Zones	18
Adding Refinement Zones	18
5.3 Farfield Controls	19
5.4 Mesh Controls	20
Advanced Options for Surface Control	21
Boundary Layer Creation	23
5.5 Clipping Controls	24
5.6 Mesh Modification	25
Smoothing	25
Holes	26
Add/Delete Cells	27
Mesh Trace	28
Modify Surface	28
5.7 Batch	29
Keywords	29
5.8 Config files	35
5.9 Miscellaneous	36
Parts List	36
GUI	36
Shortcut Keys	36
Harpoon command line options	37
Import Formats	39
Export Formats	39
Harpoon machine ports	40
6.0 Parallel Meshing Using MPI	41
Geometry Examples	42
G1 Importing CAD	42
G2 Line Selection/Separation	44
G3 Hole Filling using Line Groups	44
G4 Appending Feature Lines	45
G5 Merge Similar Parts	45
Meshing Examples	46
M1 Creating a Mesh	46
M2 Slow/Fast Expansion	49
M3 Meshing Internal Geometry	50
M4 Moving the Far-Field	51
M5 Refinement Shapes	52
M6 Meshing Using Base Level	53
Single Level	53
Multi-Level	53
M7 Multi-level Meshing	54
Fully Automatic	54
User control	54
Auto Calculation	55
M8 Stretching Cells	57
M9 Wrapping geometry	57
Boundary Layer Examples	58
BL1 Basic Boundary Layer	58
BL2 Boundary Layer with Multi Levels and constant First Cell Height	59
BL3 Boundary Layer with Multi Levels and Local Cell Height	60
BL4 Boundary Layer on Farfields	61
Mesh Modication Examples	62
MM1 Hole Finding using the Mesh Trace	62
MM2 Filling Small holes	63
MM3 Filling Large holes	64
Appendix A	66
Equivolume Skew	66
Equiangle Skew	66
Warpage	66
Appendix B	67
Windings for Cell creation	67
Appendix C	68
Line format	68

Appendix D.....	69
Boundary Layer Preferences.....	69
Appendix E.....	70
Meshing with own Windtunnel (Farfield)	70

1 Overview

Harpoon is a fully automatic body-fitted hex-dominant mesher. It uses different size hexas (levels) to control the detail of the final mesh. As the level increases, the size of the hexas decrease by a factor of two (ie level 4 is half the size of level 3). The Surface Size (4.4) is the minimum level to which Harpoon will mesh. Other levels used (eg Max Vol Size (4.4), Refinement Shapes (4.2)) are defined as offsets from the Surface Cell Size

A specific cell size may be typed in to get the exact size required. This size will replace the Level 1 default size and is known as the Base level.

The Surface Size (4.4) has an AUTO level. This allows for automatic surface level assignment with varying levels on the surface. Users may override part or all of the assigned levels by manually picking parts or selecting regions of the geometry.

Harpoon puts the imported geometry in a large box then flood fills this until the level at the surface is the requested one (ie surface level). Harpoon will then concentrate on either the External, Internal or All cells to complete the meshing. At this stage Harpoon examines the hexas and local geometry to determine whether to maintain the hexa or to split it into wedges, pyramids or tetrahedra.

The mesh density can be controlled by either slow/fast expansion from the surface or by refinement boxes.

Harpoon uses around 150MBytes per million cells when meshing both internally and externally. Time to mesh one million cells depends on cpu and memory access speeds. A low end machine can produce one million cells in about 60 seconds whereas more expensive ones can be twice as quick.

2 Installation

Notes For Installation

****UNIX/LINUX****

Create a SHARC_HOME environment variable to point to the directory where you unpacked your downloaded tarball. e.g. if you unpacked the tarball in /home/fred/sharc set your SHARC_HOME to be /home/fred/sharc

Add the following lines to your .cshrc file in your home directory

```
setenv SHARC_HOME /home/fred/sharc
set path=($path ${SHARC_HOME}/bin)
```

At a command line issue the command 'source ~/.cshrc'

To run Harpoon, simply type 'harpoon'

Make sure you have a valid license key (wale2.key) placed in the **SHARC_HOME/license** directory

If you have been issued a full license (i.e. NON demo) you will need to start the license manager to be able to use Harpoon. To start the license manager simply run waled2_start

****WINDOWS****

- 1) Run the executable setup.exe
- 2) Follow the installation instructions in the install GUI
- 3) Be sure to Reboot your machine on completion.
- 4) Obtain a valid license key from your harpoon distributor and place this file in the license directory under \$SHARC_HOME
- 5) You can now run harpoon from the Start-Up Menu or the Desktop Shortcut

Make sure you have a valid license key (wale2.key) placed in the SHARC_HOME/license directory

If you have been issued a full license (i.e. NON demo) you will need to start the license manager to be able to use Harpoon.

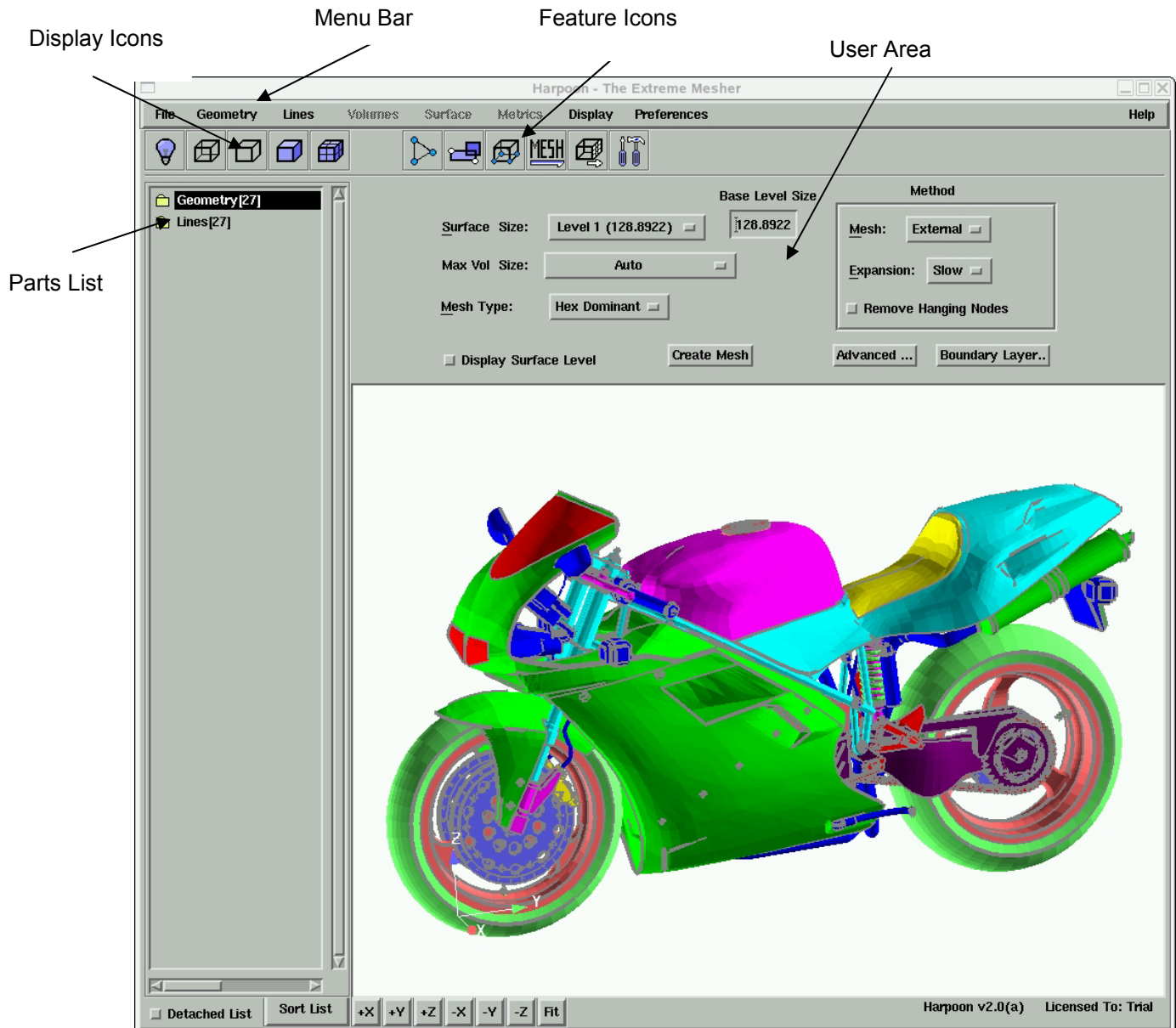
To start the license manager simply run SHARC_HOME/license/machines/win32/waled2.exe

OR

To start the license manager as a SERVICE

- 1) simply run SHARC_HOME/license/machines/win32/waled2_install.exe
- 2) Reboot your machine.

3 GUI Overview



3.1 Icons

Display and Feature Icon Description

Visibility Hidden Line Shaded Hidden Line



Wireframe

Shaded

STL Tools

Far Field

Clip Plane



Refinement

Mesh

Mesh Tools

View from +X +Y +Z -X -Y -Z direction



Fits visible geometry into Window

4 Main Menu

4.1 File

Load

Loads previously saved Harpoon(hrp) file
Loads previously saved config(cfg) file

Import

Imports Geometry to be meshed. Formats include STL, Iges, Catia V4, Unigraphics, Nastran, Fluent/Tgrid and StarCD. See [Miscellaneous \(5.9\)](#) for more details.

Append

Appends geometry files once one has been imported
Also appends Feature Lines (See [example G4](#))

Replace

Replaces the selected part with another one from file.
Can only select and replace one part at a time.
Part information is copied from old part to new part
Formats allowed: STL, Nastran, Fluent/Tgrid or StarCD format

Export

Exports mesh in various following formats:
EnSight, Fluent, StarCD, CFD++, CFX5, Gambit, Cobalt/AVUS,
Acusolve, OpenFOAM, DTF*, CEDRE,
Nastran, Marc, LSDyna, Ansys, ABAQUS, STL.
See Miscellaneous section for details of the export formats.
* Not available on Mac OSX

Save

Saves Harpoon file (Geometry only, whole mesh if it exists, info only and an old format geometry only). The info file contains commands executed, and some general parameters about the meshing case
Saves STL Geometry and modifications.
Saves config file

4.2 Geometry

Sweep Select

Sweep selects the visible parts. (Use left mouse button to scribe a box)

Separate

by Region

Separates the STL part based on contiguous regions

by Feature

Separates the STL part based on feature angle. Angle is defined in Preferences

by Selection

Separates the STL part based the selected triangles.

by Local tri segment

Separates the STL part based the selected triangle. Creates only two parts

NOTE: Only one part may be highlighted in the parts list at a time for the Surface separation.

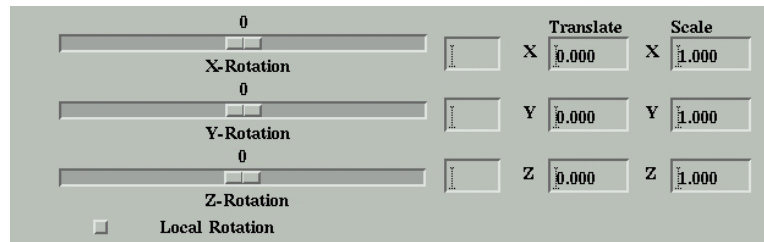
Mirror

In X,Y, Z

Mirrors selected STL in X, Y or Z

Transform

Rotates, translates and scales selected parts.



Rotate

Select a part (or parts) either:

- 1) Move the slider ($\pm 180^\circ$)
- 2) Enter a value in the text box next to the slider.

(Hit 'Enter' on keyboard to accept the values)

Rotation will be about the whole model centre.

Rotation about the Part centre can be done by

Clicking the 'Local Rotation' Toggle.

Translate

Select a part (or parts) and enter value in box

Hit 'Enter' on keyboard to accept the values

Scale

Select a part (or parts) and enter value in box

Hit 'Enter' on keyboard to accept the values

Delete Parts

Deletes selected part(s). Only imported geometry may be deleted

Merge Parts

Merges selected parts. Only imported geometry may be merged. Cannot unmerge. This just groups parts and does not merge nodes.

Merge Similar Parts

Merges parts of similar size. See [example G5](#)

Rename Parts

Renames selected part

Select Part

Selects part(s) which by wildcard (eg wheel*)

4.3 Lines

Sweep Select

Sweep selects the visible lines. (Use left mouse button to scribe a box)

Separate

Line by Region

Separates the selected line by region.

Line by Feature

Separates the selected line by feature.

Line Segment by Region

Separates the selected line segment by region. (See [example G2](#))

Line Segment by Feature

Separates the selected line segment by feature. (See [example G2](#))

Delete Lines

Deletes selected part(s). Only imported geometry may be deleted

Merge Lines

Merges selected parts. Only imported geometry may be merged. Cannot unmerge. This just groups parts and does not merge nodes.

Rename Line

Renames selected part

4.4 Volumes

Delete Volume(s)

Deletes selected volume(s)

Merge Volumes

Merges selected volumes. Volumes need to be touching to merge

Rename Volume

Renames selected volume

Group Volumes

Groups selected volumes. Volumes will have the same ID even if not physically touching each other

Reduce Hexa count

This command is to be used just before exporting. "All cell" smoothing should be done after this to maximize the reduction. Choose X Y or Z to coarsen. Harpoon will merge two neighbouring hexas in the desire direction. The can give from 10%-40% reduction in cell count, depending on the geometry and setup. This very useful when used with refinement shapes.

Note: After using this command, volume operations will be limited to smoothing only (ie no deleting, merging, updating etc).

Compact Cells

Reduces the RAM used. Useful if RAM is an issue when creating boundary layer. Compact before BL creation.

4.5 Surface

Sweep Selected Surfaces

Sweep selects the visible surfaces. (Use left mouse button to scribe a box)

Group Surfaces

Groups selected surfaces. Surfaces do not need to be touching

Rename Surface

Renames selected surface

Copy to Geometry

Copies selected surface to the geometry folder

Separate by Region

Separates selected surface by region

Clean Surface Mesh

Standard Removes bumps and other surface flaws

Aggressive As standard but tries more complex problems

Delete Surface Holes

Will delete holes/edge found by the hole finding in the Surface mesh

Delete Baffles

Will delete baffles found by the baffles finding in the Surface mesh

4.6 Mesh Metrics

Mesh Stats

Displays dialog with mesh statistics such as number of cells

Mesh Quality

Displays dialog with mesh quality information (See Appendix A for details):

- Equiangle Skew for all cells

- Equiangle Skew for Hexas only

- Equiangle Skew for Wedges only

- Equiangle Skew for Pyramids only

- Equiangle Skew for Tetrahedra only

- Equivolume Skew for Tetrahedra only

- Quad Face Warpage (available in the full listing)

Cells may be displayed in a for a given range and then selected and deleted or just deleted

4.7 Display

Line labels may be toggled on/off

Perspective may be toggled on/off

Pick rotation will reset rotation point with left mouse click (shortcut is 'r')

4.8 Preferences

Geometry Preferences

Create Intersections:

This toggle button will analyse the input geometry and decide whether to add extra lines (intersection lines) where the geometry intersects itself. This is useful if your geometry is not a perfect surface mesh.

Dirty Geometry

If Clean is selected then Harpoon will mesh as normal.

If Dirty is selected Harpoon will perform extra checks to help avoid holes. This is useful for larger gaps in geometry.

Separation Angle

Used in Geometry>separate>by feature to separate geometry parts by separation angle

Extraction Angle

Used to create the feature line parts. Recreates the feature line parts dynamically when angle is changed and preference file is saved.

Part Description

Select Use STL name to use name given in STL file

Select Use Filename to use name of file

CAD Preferences

Import CAD as

Sets the way in which the parts are created in Harpoon. Choices are:

Separate Parts

Single Object

Separate Faces

The heirachy is Single Part, Separate Parts and then Separate Faces

Construction Surface (Unigraphics only)

Include or Ignore construction surfaces

Blanked Surfaces

Include or Ignore blanked/non-visible surfaces

Chord Error

The maximum chord error allowed in tessellation

Max Size

The maximum size triangle created in tessellation

Min Size

The minimum size triangle created in tessellation

Initialisation Size

The initial triangle size in pre-tessellation

Edge Ratio

The maximum edge ratio allowed in tessellation

Boundary Layer Preferences

Use BL collapse

Toggle to collapse BL for acute angles

Angle for BL collapse

Acute angle under which BL will be collapsed

No. of Neighbours fo BL collapse

The number of faces away from the angle to be collapsed

Distortion Limit for Farfield Hexas

Limit to stop squashed Hexas in volume

Baffle Treatment

Allows BL 'On one side' or 'On both sides' of a surface

See [Appendix D](#) for further explanation

Meshing Preferences

Mesh Type

Sets the type of mesh: External or Internal

Expansion

Sets the type of expansion: Slow or Fast

Hanging Nodes

Allows hanging nodes or not

Name Volumes

Names the volumes by fluid number or attached surface

Optimization

None - no optimisation

Standard – basic meshing improvements to help with any convergence issues in solvers

Aggressive – use only if “Standard” has problems with convergence in solvers

Number of Mesh Smooths

Sets the number of automatic smooths to be done at the end of meshing. All cells with a skew above the Target skew will be smoothed.

Target Skew

Sets the Target skew for automatic smoothing

Number of Cells Between walls

Set minimum no. of cells produced between two close walls. This will only be used in the AUTO calculations

Max No. of Separate Fluids

Select the maximum number of fluids Harpoon will create. If actual number exceeds this preference, then Harpoon will place all fluids in last volume.

No. of Cells for Auto Vol Delete

Select the number of cells allowed in a volume. If a volume has less than this number, then those cells and volume will be removed from the mesh

Maximum Warp

Sets angle for worst quad warp in mesh

Wall PID

Sets the part ID to be used if a cell is deleted (using the mesh tools) of the faces which are subsequently created

Export Preferences

Reset Global Rotations

On/Off option for rotating mesh back when export (if geometry rotated prior to meshing)

BDF Format

Select Long/Short format

BDF Grouping

Select Volume/Element grouping

FEA Pyramid Treatment

Select Pyramid treatment (works only for FEA exports):

Degenerate CHEXA

Degenerate CPENTA

Split into two Tets

Fluent Thin Wall Treatment

Select Double sided, Single sided or BcSingle sided.

Double sided will export with walls pointing to only one cell. Two walls will be exported.

Single sided will export with walls pointing to two cells if wall has fluid on either side. One wall will be exported and a shadow wall will be created when imported into Fluent

BcSingle sided will as Single sided. If boundary condition is set to interior, will export the bc and neighbour volumes eg: interior:hx1:vol1:vol2

Fluent Hanging Treatment

Use Hanging Node – will export mesh with Parent/child hanging nodes

Use Polyhedra – will export mesh with hanging nodes as Polyhedra

Use All Polyhedra – will export mesh with all cells Polyhedra

StarCD Cell Offset

Offset value for StarCD cell table number (cell type number)

Export Format

Choose to export Binary or ASCII. ASCII only for Fluent, Star and OpenFOAM

Does not apply to those formats which are ASCII only (eg FEA exports, Gambit and STL)

Endian Format

Select Native to let Harpoon control the endian format or Little to export as little endian

Endian Precision

Select single or double. (Only available for Fluent and CFX)

Run-Time Preferences

-fastdisplay

Displays all parts and surfaces as bounding boxes

-nodal

Uses nodal spacing in the Auto calculation area

-firstgeom

Will select the first geometric part as a wall if multiple parts overlap

-lastgeom

Will select the last geometric part as a wall if multiple parts overlap

-UTM

Will help separate out the volumes in complicated meshes. External, Internal and All meshes can use this option

-acusplit

Acusolve format with acusplit option for Tets

-volumeperpart

Will separate parts out which touch different volumes

-cfxvol

Exports cfx with full volumes

-fillnewpart

Puts the new triangles into a new part when filling holes in Geometry

-extractfaces

Extracts faces from volumes element (Nastran and Star only)

GUI Preferences

Mouse Button

Defines mouse buttons

Perspective

Select on/off

Draw Method

Choose from display type

Line Thickness

Set thickness on lines

Edit Part Attributes

Colour:

Select a part then select a colour to change colour

Display:

Select a part and change Display Attributes to Bounding Box or 3D Border-2D Full

Type:

Geometry Parts

Select a part and change Display Attributes to Wall, Farfield or Mesh.

Wall is default

Farfield can be used to assign parts to act as farfield walls. This will allow large cells on these parts than base level. Useful if use own data for windtunnel shape

Cartesian will put a Cartesian mesh in that part only

Lines

Can change the line to be a Level line. This will assign levels and expansions but will not move nodes to the line

Surface Parts (Fluent export only)

Sets up boundary conditions (BCs)

Select a part and change Display Attributes to Velocity Inlet, Mass Flow Inlet, Pressure Outlet, Symmetry, Radiator, Interior or Wall.

Wall is default

If Radiator is used and the export is set to "Single-sided" for "Fluent Thin Wall treatment", then the part will be split into wall and non-wall sections (eg "rad" will become "rad_wall" and "rad")

NB: Surface BCs are not saved to the Harpoon file

Thickness:

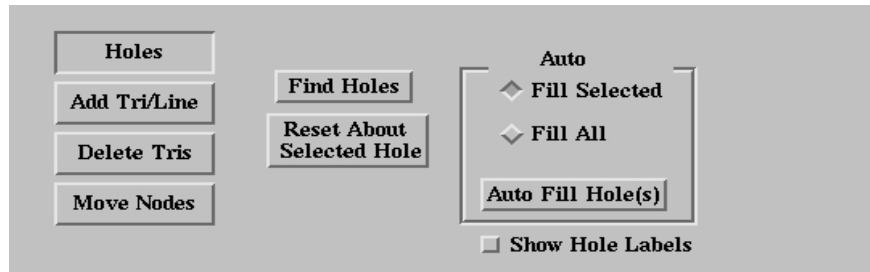
Changes lines thickness

Edit Background Colour

Select a colour to change background colour

5 Features

5.1 STL Modification



Pressing the **STL Modify** icon produces the above user area

This dialog is used for manipulating the imported geometry and consists of 4 areas

- | | |
|------------------------|--|
| 1) Holes | Find and fill holes |
| 2) Add Tri/Line | Add triangles or lines |
| 3) Delete Tris | Delete triangles individually or by selecting them |
| 4) Move Nodes | Move nodes of the triangles |

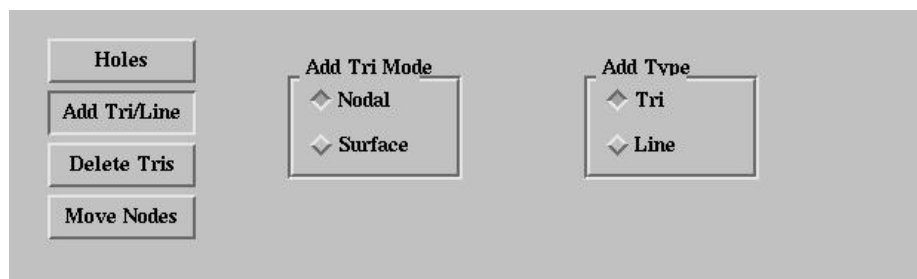
By default, the user is in the hole finding area. This is shown by the **Holes** button being indented. To enter any of the other areas simply click on the other raised buttons.

Holes

- 1) Click the **Find Holes** button. This will search for any holes in the model based on free edges. These holes are placed in the parts list under their own directory. Their display can be toggled on or off just as normal geometry parts can be. To see how the holes in the list are associated with the holes in the display, simply toggle the '**Show Hole Labels**' and the holes in the display window will have a number drawn next to them which relates to the number in the parts list.
- 2) Select a hole of interest in the parts list. Click on the '**Reset About Hole**' button. The display will be reset about that hole.
- 3) Click on the 'Auto Fill Hole(s)' button. Harpoon will attempt to fill the hole with triangles automatically. These triangles are put into the part the hole was attached to. To place the new triangles into a new part, the run time option [-fillnewpart](#) must be toggled on
- 4) Alternatively, the user can toggle on the 'Fill All' button and click on the 'Auto Fill Hole(s)' button. This will go round all holes that have been found and attempt to fix them all.

Add Tris/Lines

The **Add Tri/Line** button takes the user to the following dialog



NB: all triangles created in this area will not be added to any part. They will be stored in memory and used in meshing. The best way to manage this is to save out the geometry as STL format, restart Harpoon and load in the new file.

Tris:

Nodal Surface

Triangles are added by picking existing nodes only
Triangles are added by picking points anywhere on the geometry surface

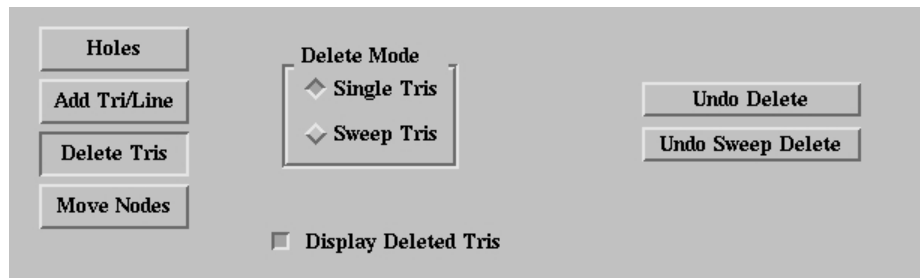
Lines:

Nodal Surface

Lines are extracted from 2 points selected on the edged of a triangle.
Not used

Delete Tris

Clicking on the **Delete Tris** button will take the user to the following dialog



Single Tris

Delete triangles by picking individual triangles

Sweep Tris

Delete triangles by sweeping an area using the mouse
All triangles within the area will be deleted

Undo Delete

Undo the added triangles one by one

Undo Sweep Delete

Undo deleted swept triangles

Single Tris allows the user to pick (using 'P') individual triangles for delete.

Sweep Tris allow the user to sweep an area using the mouse to select multiple triangles for delete.

Triangles that have been deleted can either be displayed (in yellow) or have their visibility turned off by toggling the 'Display Deleted Tris' button.

Move Node

Clicking on the **Move Nodes** button takes the user to the following dialog.



Global

Allows the user to move nodes in global 3-D space

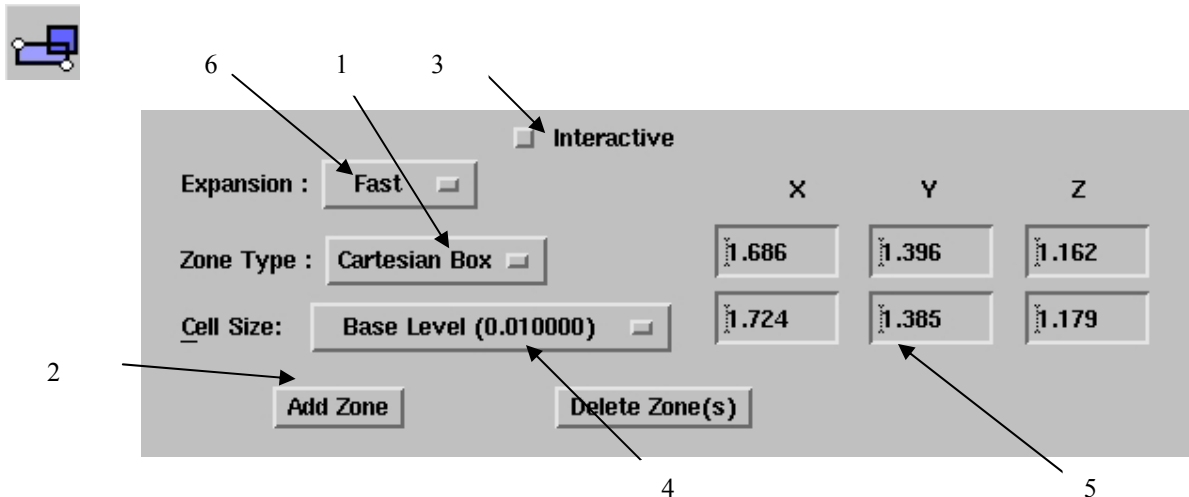
Restricted

Node movement is restricted to the geometry surface

Use 'P' to select the desired node to operate on.

To exit any of the above modes, the user should simply come out of the dialog area by clicking on another icon such as the Mesh icon

5.2 Refinement Zones



Pressing the Refinement icon produces the above user area.

Adding Refinement Zones

Click on **Refinement** Icon (Second from left on top row)
The above user area will appear.

1. Choose **Cartesian Box**, **Sphere** or ***Trapezoid Box** (1)
2. Click **Add Zone** (2)
3. Select 2 points on STL surface

*The Trapezoid is actually a box where all of the corners may be moved. NB the trapezoid can only have numbers enter in the same way as the box. To enter all 8 points a config file is needed (see [example M5](#))

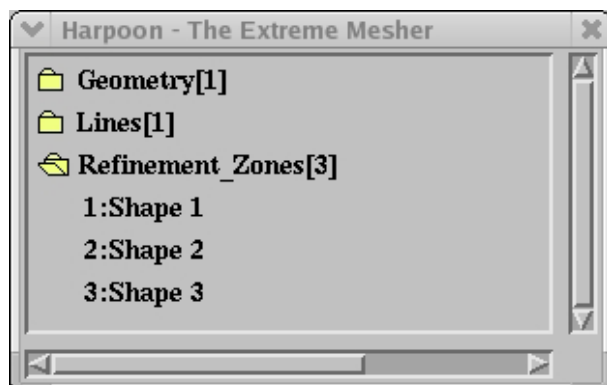
A Refinement Zone folder will be added to the Parts list.

Shape can be moved by clicking **interactive** button (3)
The middle blue node moves the whole shape
The end nodes adjust size/moves the node.

To change the Cell Size (refinement level), from the default of Base level, use the pull down menu (4).
The rate of cell size expansion is controlled by the Expansion pull down menu (6).

Refinement shapes cannot be deleted but the visibility can be turned off. Harpoon will ignore any zones not visible.

To change the visibility of a zone select the Shape in the parts list and hit the visibility icon.



Refinement zones may be changed from ,say, a box to a sphere by selecting a zone and changing the **Zone Type**. The user may also edit the values in the text boxes (5) to place the zones more accurately.

5.3 Farfield Controls



Coordinates: <input type="text" value="Global"/>			<input type="checkbox"/> Interactive	
	X	Y	Z	
Min	<input type="text" value="-581.844"/>	<input type="text" value="488.003"/>	<input type="text" value="-407.810"/>	<input type="checkbox"/> Max X <input type="checkbox"/> Min X
Max	<input type="text" value="382.207"/>	<input type="text" value="476.048"/>	<input type="text" value="556.242"/>	<input type="checkbox"/> Max Y <input type="checkbox"/> Min Y
				<input type="checkbox"/> Max Z <input type="checkbox"/> Min Z
Wall :	<input type="text" value="Max X"/>		Level :	<input type="text" value="Auto"/>

Pressing the **Farfield** icon produces the above user area

Use **Max/Min buttons** to automatically snap the bounding box to a particular location on the geometry.

e.g. Snap to **Min Z** to create a floor for a car geometry, or a symmetry plane for a half model.

Any farfield wall may be refined by selecting the 'Wall' pull down menu and entering a base level offset. Select 'auto' to get back to default settings if required.

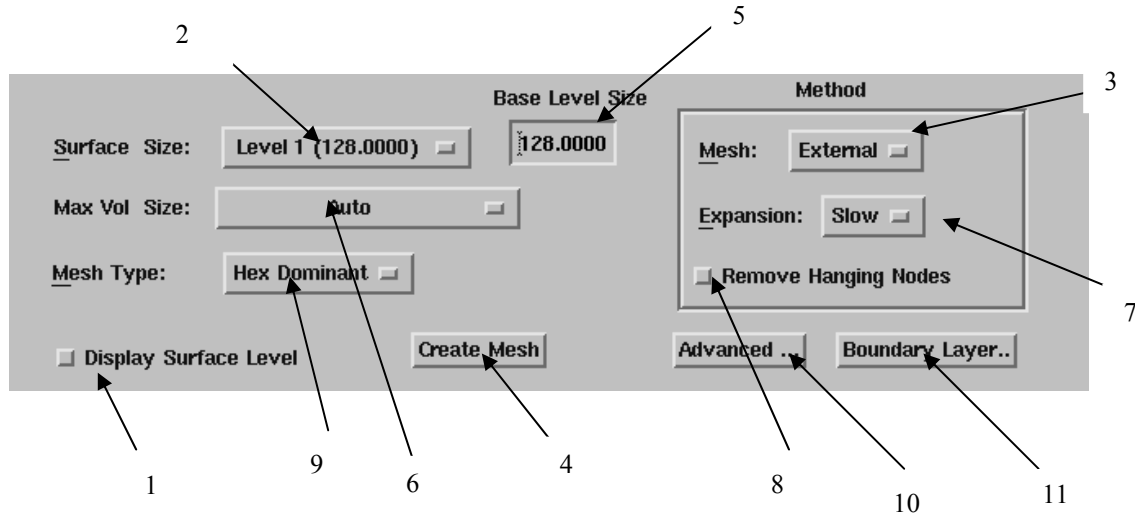
Clicking on the **interactive** button allows the user to position the bounding box

Using the mouse. The box is moved by clicking on one of the 3 pale green squares that appear (one in the centre of the box and one at each of the box extremities).

Specific values can be typed into the text boxes by the user. The values will be updated once the user hits the **Enter** key on the keyboard

The **Coordinates** pull-down menu gives you the option of Global coordinates or Body multiples. This is useful in setting up a windtunnel configuration. The multiple is based on the maximum body length

5.4 Mesh Controls



Pressing the **Mesh Icon** will produce the above user area.

Select **Display Surface Level (1)** to show the physical size of surface cells on model. The levels will increase the final mesh size by a factor of 4. To get mesh sizes in between two levels simply change the base level. When you input a new base level all levels are updated.

Example:

Assume the following:

If Base level of size 10 then Level 1 will give size 10 cells and Level 2 will give size 5 cells.

If Level 1 gives 100,000 cells and level 2 gives 400,000, then to get about 200,000 cells input a Base level of 7.5 and use Level 1.

See [example M6](#) for details of Levels and the Base Level

Adjust **Surface Size** to the required level. **(2) or (5)**

Select **External, Internal** or **All(3)**

Click **Create Mesh (4)**

Only VISIBLE parts will be processed for meshing

The **Base Level size** text box **(5)** shows the size of the hexa representing the base level. Harpoon creates a default value and places it here.

The user may input a specific value in the text box. This will set the 'Base Level' to be that physical size. Harpoon will use this size to mesh. Selecting any of the levels in the pull down menu for the Surface Cell size will change the Base Level back to the default value. See [example M6](#) for more details

Other Options

The **Max Vol Size** menu **(6)** allows the user to limit the maximum cell size in the Volume region. I.e. away from the surface. Levels are offsets from the **Base Level**

The **Expansion** menu **(7)** allows the user to choose between slow and fast expansion of the volume mesh

The **Remove Hanging Nodes** toggle **(8)** allows the user to specify whether hanging nodes should be removed from the mesh.

The **Mesh Type** menu **(9)** allows the user to choose Hex dominant, 100% Tet, Cartesian or Wrapped meshes.

The **Advanced** button **(10)** allows the user to have greater control over the surface cell sizes.

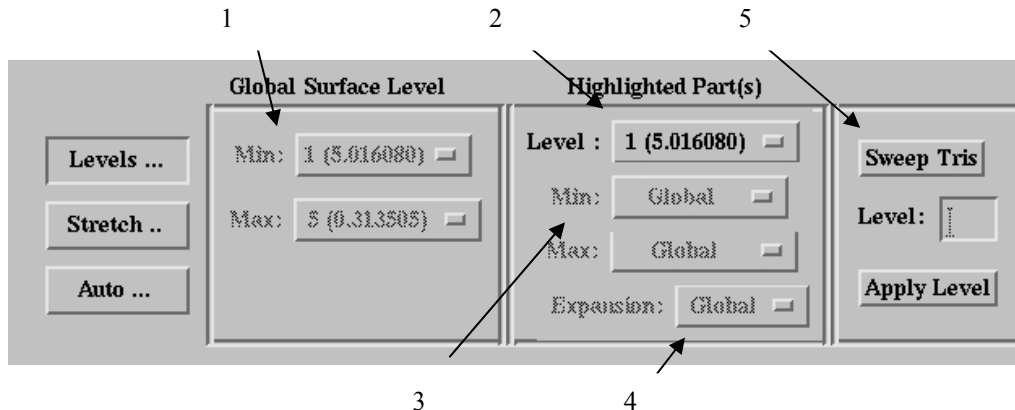
The **Boundary Layer** button **(11)** allows the user to generate a boundary layer on selected parts.

Advanced Options for Surface Control

This section allows the user to control the surface refinement more closely and to change the basic shape of the cells.

Levels...

The user will be put into the **Levels** area after clicking the **Advanced** button from the main meshing area. The levels created by the automatic option may be controlled in this area. Priority runs from right to left (ie any swept triangles will override Highlighted Parts). All options refer to surface refinement levels.



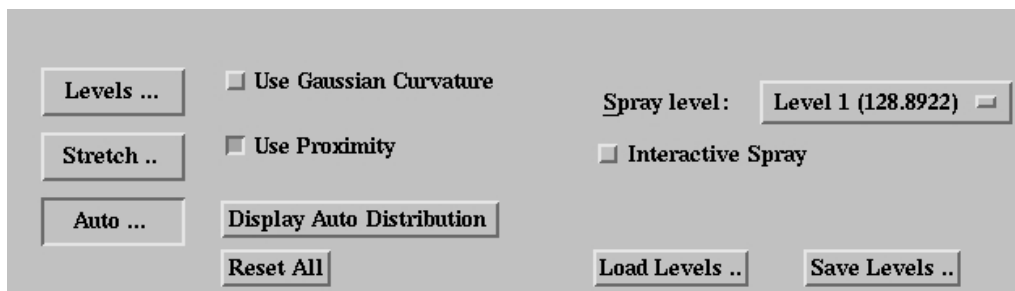
To limit the global automatic surface levels, select the Min or Max boxes in the **Global Surface Level** area (1). These will only be highlighted if the **Surface Size** in the main meshing area has been changed to **AUTO**.

To change the levels on a part (or parts), highlight parts in parts list and select level (2). If AUTO is selected, then the values calculated may be restricted on a part by part basis by changing the Min and Max values (3). Values selected here will override those chosen in the Global Surface Level area (1). Expansion on a per part basis maybe changed to slow (4). Be careful with this option since the expansion can penetrate deep into the mesh and can give you a large number of cells. See [example M2](#) for more details.

To change the levels on a set of triangles, press the **Sweep Pick Tris** button (5) and sweep select triangles using left mouse button. Enter a value in the Assign Level box and click Apply Level.

Auto...

Select the **Auto...** button to control the AUTO calculation by Harpoon.



If the Gaussian Curvature off and the Proximity on, the user will get the AUTO calculation used in versions of Harpoon prior to v130b.

The Gaussian Curvature uses a Gaussian calculation of the curvature to calculate levels on the model.

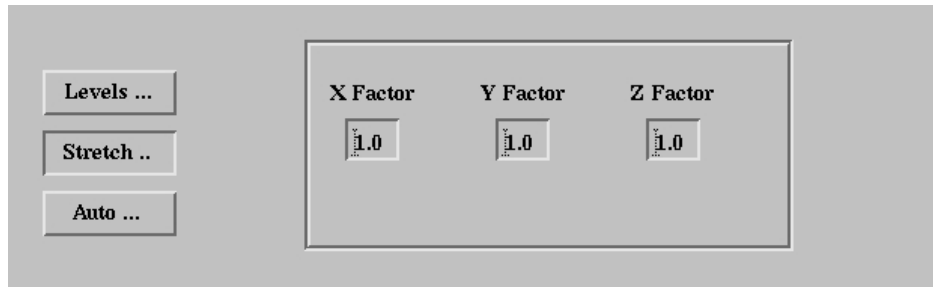
If the Gaussian option is not used, Harpoon will use a default method for estimating the curvature.

The Proximity option forces Harpoon to check for walls that are very close to each other and assign levels to capture this closeness. This calculation can take some time on very large models and should only be used where necessary.

The levels to be used may be displayed and altered. See [example M7](#) for more details.

Stretch...

Select the **Stretch...** icon to control the aspect ratio of the hexas



The image shows a software interface for adjusting the aspect ratio of hexas. On the left, there is a vertical stack of three buttons: "Levels ...", "Stretch ..", and "Auto ...". The "Stretch .." button is highlighted. To the right of these buttons is a large rectangular panel. Inside this panel, at the top, are three labels: "X Factor", "Y Factor", and "Z Factor". Below each label is a small input box containing the value "1.0".

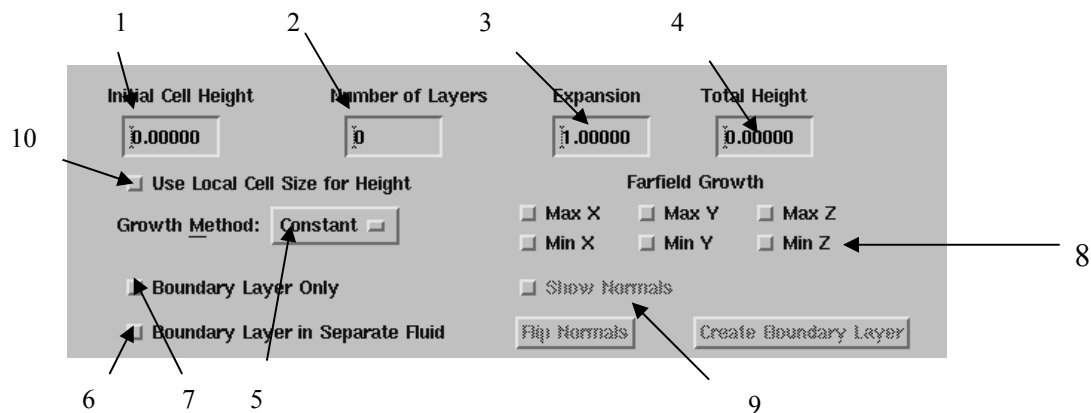
Enter the aspect ratio required and hit **Apply Settings**.

To stretch a hexa 3 times in the X direction, enter 3.0 in the **X factor** box and hit apply. There is a maximum limit of 4.0

See [example M8](#) for more details.

Boundary Layer Creation

This section allows the user to create boundary layers on one or multiple parts.



To create a boundary layer select a part (or parts) and choose an **Initial Cell Height** (1) and **Number of Layers** (2). (Note: hit enter in the text box for the values to be accepted). The **Total Height** of the boundary layer is shown in text box (4) which will not accept input values. Choose either a Constant or Linear **Growth Method** (5). If Linear is chosen, then the **Expansion** (3) rate may be changed. To base the cell height on the neighbour cell size, toggle on the **Local Cell Size** (10) button. When this is toggled, (1) changes to **Local Height Factor** which will be used to change the BL height. The **Local Cell Size** (10) is applied on a per part basis. This is useful when a large range of cell sizes is used.

If only the boundary layer is to be kept, then select the button **Boundary Layer Only** (6). If the boundary layer is to be placed in a separate volume select the button **Boundary Layer in Separate Fluid** (7).

If a boundary layer is required on a farwall then select the appropriate **Farfield Growth** (8). See [example BL4](#).

Once the user is happy with the settings, they should return to the **Mesh** dialog and start the meshing process. The boundary layer will be created at the end of the meshing process.

If the user has already created a mesh with no boundary layer but wishes to add one to the mesh, then this too is possible. If a mesh already exists, the greyed out area (9) will become active.

This time, to generate a boundary layer, the user should apply the settings as described above, on the Volume parts or Surface Mesh Parts, **not the Geometry parts**.

Once these have been applied, the user should then click the '**Create Boundary Layer**' button, and Harpoon will add the boundary layer to the existing mesh.

If a **Surface Mesh** part is attached to more than one volume, the direction of growth for the boundary can be controlled by the '**Flip Normal**' button. The boundary layer will grow in the normal direction, which can be displayed by turning on the '**Show Normals**' toggle

Once a part has had a boundary layer inserted, it cannot be redone. If the user wants to try different boundary layer heights, then the Harpoon mesh file should be saved prior to any boundary layer creation. The file can be reloaded to try different heights

Different heights may be created on touching parts by altering the **Initial Cell Height**, **Growth Method** or **Expansion**

See [BL examples](#) for more details.

5.5 Clipping Controls



130.928	<input type="checkbox"/>	130.928	<input type="checkbox"/> Clip Selected Volumes
-477.401	<input type="checkbox"/>	477.401	<input type="checkbox"/> Full Cell Clipping
1251.181	<input type="checkbox"/>	1251.182	<input type="checkbox"/> Shaded Clip Plane
			<input type="checkbox"/> Geometry Clipping
			<input type="checkbox"/> Flip Clip Plane

Clicking the **Clip Icon** will produce the above user area.

Clips on the Volume cells is done by turning on the X and/or Y and/or Z clip buttons.

The clip can then be moved by moving the appropriate clip slider or by entering a value in the text box.

Full Cell Clipping shows the full cells in the clip region as opposed to clipping the cells. Only when Full Cell Clipping is on will the user see all cells

Geometry clipping toggles on/off the clipping of the STL and the surface mesh if created

Flip Clip Plane toggles the direction for the geometry clipping

Note: Currently the standard volume clip only performs a clip on the Hexahedral cells. Any cells that are not of this type will not be included in the clip. Select Full Cell Clipping to view all cell types in the clip

5.6 Mesh Modification



The dialog box is titled 'Mesh Modification'. It contains three main panels. The left panel, 'Modify Surface', has four buttons: 'Smooth' (bold), 'Mesh Holes', 'Add/Del Cells', and 'Mesh Trace'. The middle panel, 'Auto Smooth', has two input fields: 'No. of Smooths' with the value '2' and 'Target Skew' with the value '0.950'. Below these is a checkbox labeled 'Smooth All Cells' and a 'Smooth Volume' button. The right panel, 'Manual', has a 'Display Worst Skew' button, a checkbox labeled 'Adjust Cell Nodes', an 'Undo' button, and a 'Worst Skew =>' field showing the value '0.947865'.

Pressing the **Mesh Modify** icon produces the above user area

This dialog is used for manipulating the generated mesh. The user can **smooth** the mesh to a desired target skew, find poor quality cells and manually adjust them, find **holes**, **delete** faces or **add** new faces.

By default, the user is in the mesh **smoothing** area. This is shown by the **Smooth** button having a bold text. To enter any of the other areas simply click on the greyed out buttons.

Smoothing

There are two distinct methods for mesh smoothing. The first is the **Auto Smooth** area where the user simply enters the number of smoothing iterations and the target worst skew. The **'Enter'** button should be hit on the keyboard to apply the changes to the values chosen. Clicking on the **Smooth Volume** button will then perform the relevant smoothing on the selected volumes in the parts list. If no volumes are selected, smoothing is performed on all volumes. The target smooth works on Tets only and is based on equivolume skew. Once smoothed the worst skew in the mesh is reported in the bottom right of the dialog and this cell is highlighted. Toggling the **Smooth All Cells** button will do a global smooth on the whole mesh when the **Smooth Volume** button is pushed.

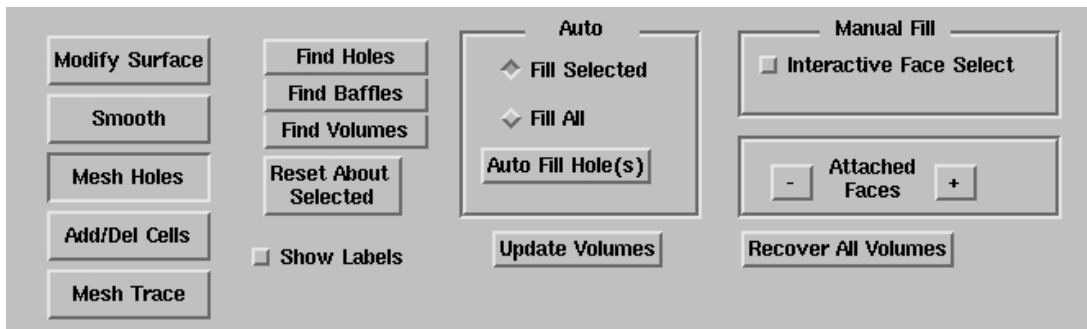
The second method of improving cell quality in the mesh is by **Manual** means.

Here, the user clicks the **'Display Worst Skew'** button and the view resets about the worst skewed cell in the mesh. The skew value is reported in the bottom right of the dialog. The user can adjust any of the nodes on this cell by simply toggling on the **'Adjust Cell Nodes'** button. This will allow the user to interactively adjust the nodes (by using the left mouse button to select the node and holding down this button while moving the mouse) and see how the skew is affected by the node movement. The updated skewness is continuously reported in the bottom right of the dialog. As well as the nodes of the worst skewed cell, the view also displays the neighbouring cells. The nodes for these cells can also be manipulated. The graphics window and the dialog always report the worst skewed cell attached to the node that is being manipulated.

The user can reset the node movement by clicking on the **'Undo'** button

Holes

By clicking on the Mesh Holes button, the user is taken to the following dialog



Note: The Find Holes and Find Baffles now work on selected parts only

Holes

- 1) Select the Surface parts you want to search
- 2) Click the **Find Holes** button. This will search for any holes in the selected surface mesh based on free edges. These holes are placed in the parts list under their own directory. Their display can be toggled on or off just as normal mesh parts can be.
- 3) Select a hole of interest in the parts list. Click on the '**Reset About Hole**' button. The display will be reset about that hole.
- 4) Click the '**Auto Fill Hole(s)**' button. Harpoon will attempt to fill in the hole. The new faces are displayed in grey.
- 5) Click '**Update Mesh**' to apply the new faces to the existing mesh. This will also rescan the volumes and separate them out if fixing the hole produces multiple volumes.

Alternatively, once the user has reset about a selected hole, faces can be added **manually** to fill in holes.

If the user toggles on the '**Interactive Face Select**' button, a number of faces will appear attached to the hole will be displayed. The user can then select a face using the cursor and by hitting 'p'. A new selection of faces will be offered. Repeat process until hole is filled. If the selection is not what you need, simply turn the '**Interactive Face Select**' button off then on again. This will move onto the next segment of the hole.

If the user has a large surface mesh part, select the hole and press + in the **Attached Faces** box. This will give you a copy of the surrounding faces so the surface mesh part may be turned off. This allows much easier hole filling with large models.

When all holes are finished, hit the '**Update Mesh**' button.

Baffles

- 1) Select the Surface parts you want to search
- 2) Click the **Find Baffles** button. This will search for any baffles (a face attached to two faces in the same volume) and display them in a Baffles folder

Baffles behave in a similar way to holes. They can be deleted (Surface-> delete baffles), reset about, have faces grow from them or have interactive faces selected. They cannot be filled.

If a baffle is deleted, then the physical quads/tris which represent the baffles will be removed

Volumes

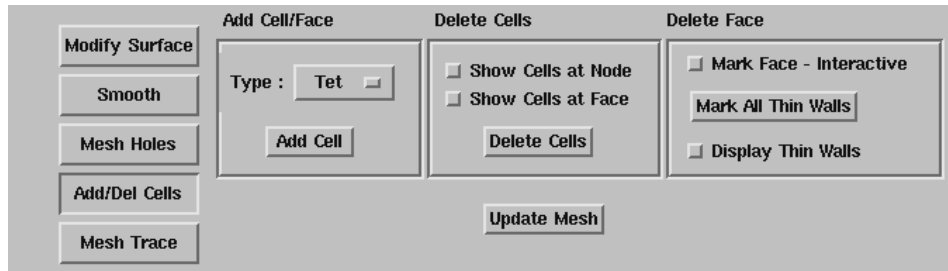
- 1) Select the Surface parts you want to search
- 2) Click the **Find Volumes** button. This will search for any groups of cells which might cause a problem with convergence. It will look for areas of single cells and put them in separate volumes

'**Recover All Volumes**' will give you all volumes internal and external no matter what you asked for

See also [Mesh Trace](#) for those hard to find holes.

Add/Delete Cells

By clicking on the Add/Del Cells button, the user is taken to the following dialog



Cells may be added **manually** to the mesh by selecting **Type** (Tet, Pyramid, Wedge, Hexa, Tri or Quad) and clicking on the 'Add Cell' button. Select the relevant surface nodes on the mesh. A description of the cell windings (order in which the nodes should be picked) for each cell is given in Appendix B.

Note that faces can only be created on the cells external faces. Also, a tri face cannot be created on the quad face of a hexa for example.

Again, newly created faces are shown in grey. These faces can be included in the mesh by hitting the '**Update Mesh**' button.

To delete all the cells attached to a node, turn on the **Show Cells at Node** button and select a node. Clicking on the Delete Cells button will delete them.

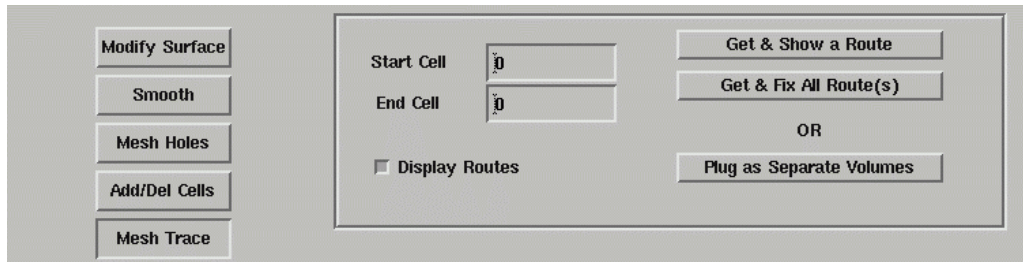
To delete all the cells attached to a face, turn on the **Show Cells at Face** button and select a face. Clicking on the Delete Cells button will delete them.

Specific faces can be deleted. The user can individually select faces by toggling the '**Mark Face - Interactive**' button and using the 'P' key. **Only thin wall or baffle type faces can be deleted.** The user can display any thin walls in the mesh by toggling the '**Display Thin Walls**' button. This will colour any faces on a mesh part a dark grey colour.

Once the faces have been marked the user should click on the '**Update Mesh**' to update the changes to the mesh.

Mesh Trace

By clicking on the Mesh Trace button, the user is taken to the following dialog



This is a very useful tool for finding holes in the mesh. Sometimes it is hard to see where the hole in the mesh has occurred. The Mesh Trace works out the shortest distance between two cells.

To use, display a clip plane (using the clip icon) from which the user will select the cells. Click on the Mesh Trace button and use 'P' to select. Hit **Get & Show a Route** and Harpoon will display the route between the cells.

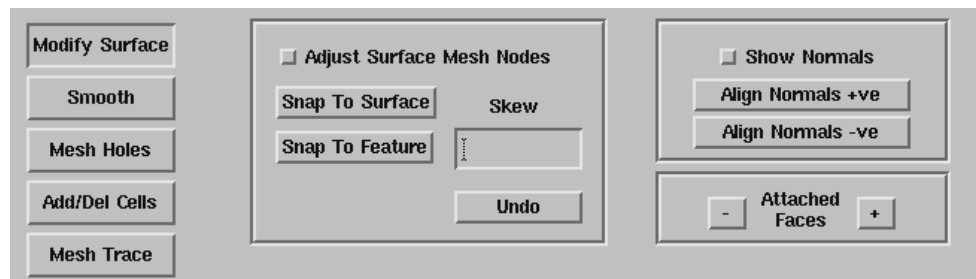
For small holes, hit **Get & Fix All Route(s)**.

For large holes, hit **Plug as Separate Volumes**.

See [example MM1](#) for further details.

Modify Surface

By clicking on the Mesh Surface button, the user is taken to the following dialog



Turn on the **Adjust Surface Mesh Nodes** button. Select a node and click either the **Snap to Surface** or **Snap to Feature** buttons. The skew, of all cells attached to that node, will be displayed. Click the Undo button if result is not satisfactory.

For large models, select a node and then press + in the **Attached Faces** box. This will allow the visibility of the large surface mesh to be turned off .

If the normals on a part are in more than one direction, then select the part, click **Align Normals +ve** (or **Align Normals -ve**) and then use 'P' to select a quad or tri on that part. The normals will be aligned starting from the selected face in the positive (+ve) or negative (-ve) direction. This will only work on faces which have the same volume on either side.

5.7 Batch

To run Harpoon in batch type:

Harpoon -batch filename

where filename points to config file

Keywords

I/O commands

import

Imports either stl, nastran, Fluent/Tgrid or StarCD files

Usage: import stl wheel.stl

imports stl file wheel.stl

import nastran wheel.dat

imports nastran file wheel.dat

import stl *.stl

imports multiple files according to wildcard

CAD files cannot be loaded in batch

export

Exports mesh

Usage: export fluent vol filename

exports fluent volume mesh

export star surf filename

exports star surface mesh

Use also gambit*, cfx5**, cobalt, cfd++, acusolve, marc, ls-dyna, ansys, abaqus, foam and ensight

For Nastran an extra word is required to determine long/small field export

Usage: Export nastran small vol filename

exports small field Nastran volume

Export nastran long surf filename

exports long field Nastran surface

* Harpoon only exports gambit as a volume mesh

** Note cfx5 cannot read in surface mesh on its own. This export is the same one produced by ICEM. Use the ICEM option in CFX5 to import.

load

Loads a Harpoon file

Usage: load harpoon test.hrp

will load file test.hrp.

If a Harpoon file contains geometry only, then all the batch commands will work.

If a Harpoon file contains the full mesh, then only the volume commands and smoothing will work

save

Exports mesh in Harpoon native format

Usage: save harpoon filename

exports Harpoon file format

Geometry Part commands

split

Splits geometry into separate parts

Usage: split region 1

split part 1 by region

split feature 3 30.0

split part 3 by feature by 30 degrees

merge

Merges (groups) geometry

Usage: Merge 3

1 3 5

merges parts 1,3 and 5

delete

Deletes geometry

Usage: delete 1

deletes part 1

rename

Renames geometry

Usage: rename 1 squid

renames part 1 to squid

plevel

Controls the levels on a part

Usage: plevel 1 4 5 0

part 1 range from level 4 to 5. Can also use part name (including wildcards) instead of part number.

0 at the end represents normal wall and 1 a farwall

pexp

Controls the expansion on a part

Usage: pexp 1 4

expand part 1 four times. Can also use part name (including wildcards) instead of part number.

llevel

Controls the levels on a part

Usage: llevel 1 4

line 1 with level 4. Can also use part name instead of part number.

findgeomhole

Finds holes in the geometry parts

Usage: findgeomhole dog*

will check parts dog* parts that have been imported/loaded. Multiple lines possible. Will not actually find holes until **fillgeomhole** is called

fillgeomhole

Fills holes in the geometry parts

Usage: fillgeomhole

will find and fill holes in geometry parts described in **findgeomhole**.

rotate

Rotates a part about a vector

Usage: rotate 1 30.0

rotate part 1 through 30 degrees

1.3 1.4 2.4

point of rotation and first point

2.3 1.4 2.4

second point which forms the vector (with first point) to rotate about

This will rotate part 1, 30 degrees about the X axis.

allrotate

Rotates a part about a vector

Usage: allrotate 30.0

rotate all parts through 30 degrees

1.3 1.4 2.4

point of rotation and first point

2.3 1.4 2.4

second point which forms the vector (with first point) to rotate about

This will rotate part 1, 30 degrees about the X axis.

xrot, yrot, zrot

Rotates a part about x, y or z axis

Usage: xrot 1 30.0

rotate part 1 through 30 degrees in x axis

translate

Moves part(s)

Usage: translate 1 10 20 30

moves part 1 in X, Y and Z

Usage: translate * 10 20 30

moves all parts in X, Y and Z

Mesh commands

mesh

Sets the mesh volume – internal/external/both

Usage: mesh internal
mesh external
mesh both

type

Sets the mesh type – hex/tet

Usage: type hex	sets type to hex dominant
type tet	sets type to 100% tet
type cart	sets type to cartesian
type wrap	sets type to wrapped

baselev

Sets the base level

Usage: baselevel 0.5 sets the base level to 0.5

level

Sets the global surface cell size for mesh

Usage: level 5	sets surface level to 5
level 0	set surface level to automatic

maxsize

Sets max cell size for mesh

Usage: maxsize -1 set max cell size to be base level -1

wlevel

Set farwall level using offset from baselevel

Usage: wlevel xmin 0 set level on xmin farwall to be base level

farfield

Sets farfield position

Snapping farfield walls

Usage: farfield snap xmin	snaps farfield to min x
farfield snap xmax	snaps farfield to max x

Use also ymin,ymax,zmin,zmax

Setting farfield values

Usage: farfield xmin 9.758	sets xmin value to 9.758
farfield xmax 9.758	sets xmax value to 9.758

Use also ymin,ymax,zmin,zmax

stretch

Sets stretch for the mesh

Usage: stretch 2.0 1.0 1.0 sets cells in X direction to be twice normal size

remove

Removes hanging nodes

expand

Controls the expansion of the mesh level

Usage: expand slow	slow expansion
expand fast	fast expansion

presmooth

Smooths the mesh before boundary layer creation

Usage: presmooth 2 0.97	smooths tets cells twice with target skew of 0.97
Usage: presmooth 2 all	smooths all cells twice

smooth

Smooths the mesh

Usage: smooth 2 0.97

smooths tet cells twice with target skew of 0.97

Usage: smooth 2 all

smooths all cells twice

blcreate

Creates boundary layer on a part if a Harpoon mesh has been loaded.

Usage: layer 1 1.0 2 0 1.5 0

Part 1 initial cell height 1.0, 2 layers, constant expansion, expansion 1.5

blcreate

Create BL using layer info

layer

Creates boundary layer on a part. This command only works on parts. It will not work on surfaces or volumes

Usage: layer 1 1.0 2 0 1.5 0

Part 1 initial cell height 1.0, 2 layers, constant expansion, expansion 1.5

Usage: layer 1 1.1 2 1 1.5 1

Part 1 local cell height factor 1.1, 2 layers, linear expansion, expansion 1.5

Can also use part name (including wildcards) instead of part number

Usage: layer zmin 1.1 2 1 1.5 1

Will apply BL to zmin. Can use zmax, zmin, xmax, ymin and ymax

wrapline

Sets "Use Feature Lines" to be on when wrapping surface

xclip

Clips mesh in X after meshing

Usage: xclip 1.5

yclip

Clips mesh in Y after meshing

Usage: yclip 1.5

zclip

Clips mesh in Z after meshing

Usage: zclip 1.5

vgroup

Groups volumes after meshing

Usage: vgroup vall v1 v2 v3

Groups volumes v1,v2 and v3 into vall

extrude

Replaces existing volume with pentas. Use for cooling work

Usage: extrude cac-fr cac-rr 4

Creates 4 rows of pentas between cac-fr and cac-rr. The geometry input for cac-fr must be a mesh. Will create a non-conformal interface to rest of mesh

refine

Reads in a config file defining refinement shapes

Usage:

refine

type level

x1 y1 z1

....

xn yn zn n is 2 for box and sphere; 8 for trapezoid

where type is 0 for cartesian box, 1 for sphere, 2 for trapezoid and 3 for a cylinder

level is size of refinement size based on base level

example 1 - a box with refinement size of base level

refine

0 0

1.5 1.5 1.5

2.5 2.5 2.5

example 2 - a sphere with refinement size of base level-1. Centre is 1.0 1.0 1.0, diameter is 0.5

refine

1 -1

1.5 1.5 1.5

2.5 2.5 2.5

example 3 - a trapezoid with the 8 points given with refinement size of base level-1

refine

2 0

1.34 1.59 1.75

1.44 1.59 1.78

1.44 1.65 1.78

1.34 1.65 1.78

1.34 1.59 1.76

1.44 1.59 1.70

1.44 1.65 1.70

1.34 1.65 1.70

example 4 - a cylinder given with refinement size of base level-1

refine

3 0

1.34 1.59 1.75 point at centre of disc1

1.44 1.59 1.78 point at centre of disc2

1.34 1.58 1.74 point at edge of disc1

1.44 1.58 1.74 point at edge of disc2

refexpand

Assigns an expansion to refinement shapes

Usage: refexpand 3 – assigns expansion of 3 to refinement zones

Setbc

Sets boundary conditions to be saved in Fluent exported file

Usage:

setbc farfield_minx velocity-inlet sets minx farwall to be a velocity-inlet

Allowable parts to set are the part names in the geometry folder, farfield_minx, farfield_maxx, farfield_miny, farfield_maxy, farfield_minz and farfield_maxz.

Allowable BCs are interior, wall, pressure-outlet, symmetry, velocity-inlet, interior, radiator, and mass-flow-inlet

Volume commands

vdelete

Deletes volumes after meshing

Usage: vdelete 3
1 3 5 deletes volumes 1,3 and 5

vkeep

Keeps listed volumes and deletes the rest after meshing

Usage: vkeep 3
1 3 5 keeps volumes 1,3 and 5

vptget

Prints volume which contains the point given

Usage: vptget 10.2 34.5 6.7 prints volume to the console window

vptfix

Fixes small holes between volumes

Usage: vptfix 10.2 34.5 6.7 Tries to fix holes between all points
vptfix 11.2 34.5 6.7

vptplug

Fixes large holes between volumes

Usage: vptplug 10.2 34.5 6.7 Tries to fix large holes between all points
vptplug 11.2 34.5 6.7

vptdelete

Deletes volume which contains the point given

Usage: vptdelete 10.2 34.5 6.7 deletes the volume which contains the point

vptkeep

Keeps the volume which contains the point given and deletes the rest. Can use multiple entries

Usage: vptkeep 10.2 34.5 6.7 Keep all volumes requested and deletes the rest

vptrename

Renames the volume which contains the point. Can use multiple entries

Usage: vptrename 10.2 34.5 6.7 Fluid1

vptcheck

Checks that no points are in the same volume. Need at least 2 points

Usage: vptcheck 10.2 34.5 6.7
vptcheck 11.2 33.6 7.7

vfind

Finds volumes on listed parts

Usage: vfind 3 Finds volumes on surface mesh parts 1,2 and 3
1 2 3

Usage: vfind 2 Finds volumes on surface mesh parts Grill and Bonnet
Grill
Bonnet

Usage: vfind all Finds volumes on all surface mesh parts

5.8 Config files

A config file is created whilst Harpoon is running. It provides most information needed to setup and run Harpoon. Below is an example of a config file.

```
import stl /home/sharc/wings.stl
split feature 1 25.000000          /*split part 1 by feature angle of 25 */
merge 4                             /*merge 4 parts */
1 2 3 4                             /*parts to be merged */
**VERSION v2.0(a)**
**MACHINE LINUX**
**PREFERENCES USED**
**Max Skew 0.999500**
**Target Skew 0.980000**
**Max Face Warpage 40.000000**
**noreset**
**nointersect**
**Separation Angle 45.0**
**Setting No. of Cells Between walls to 4**
**Setting BDF Exports to Short Format**
**Setting BDF Pyramid Treatment to use degenerate CPENTA elements**
**Setting Max No. Separate Volumes to 50**
**Setting Part Description to use Filename**
**Setting Fluent Thin Wall Treatment to Double Sided**
farfield global
farfield xmin 3.888
farfield ymin 1.486
farfield zmin 0.311
farfield xmax 7.188
farfield ymax 4.786
farfield zmax 3.611
**MESH METHODS**
type hex
expand fast
mesh external
**SINGLE LEVEL
level 3
plevel 1 -1 -1 0
llevel 1 -1
**MESHING STATISTICS
**Total No. Cells 108665
**Total No. Hexas 98867
**Total No. Wedges 7762
**Total No. Pyramids 1321
**Total No. Tets 715
**Time to Mesh : 3.69 s
```

Config files do not record individual triangle operations (eg moving, adding, deleting, and applying levels)

5.9 Miscellaneous

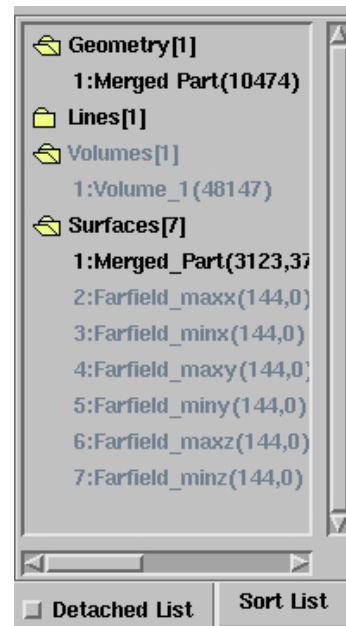
Double click (left mouse button) or 'P' on any part and Harpoon will print the part number in start up window and highlight part in parts list.

Parts List

The parts list in Harpoon has a simple tree structure associated with it.

There are Four main folders.

- 1) **Geometry**
This contains all the imported surface geometry
- 2) **Lines**
This contains all the feature lines from the imported surface geometries or imported lines
- 3) **Volumes**
This contains all the volumes created during the meshing process.
- 4) **Surfaces**
This contains all the surface mesh parts created during the meshing process. This includes any far-field walls.



Each folder can be opened or closed by clicking on the folder icon (under windows this is achieved by double-clicking on the folder).

Any item which is non-visual is coloured a light grey colour to allow the user to quickly see which parts have been turned off.

The '**Detached List**' button toggles between the parts list in the main gui and a separate parts list. Windows does not have this option since the window is already detached.

The **Sort List** button alpha-numerically sorts the parts in the sub directories.

GUI

- To get the x,y,z coordinates of the geometry or surface mesh, place the mouse over a point and hit the c key
- The colour of the volume (and volume clip) can be changed by selecting the volumes and changing the colour in the part attributes section

Shortcut Keys

- | | |
|--------|--|
| c | Will display the X,Y and Z values of the node at the mouse pointer |
| p | Will select the part under the mouse pointer |
| Ctrl+p | Will add the part under the mouse pointer to the selected list |
| r | Will set rotation point about the mouse pointer |

Harpoon command line options

Geometry

-fillnewpart	Will place the triangles created when filling holes on the Geometry into a new part
-noedgeline	Will activate –nolineperpart and will not create lines between parts unless there is an angle greater than that defined in the preferences
-nofreeline	Will not create feature lines on the edge of just one triangle
-nolines	Will not create feature lines
-nolineperpart	Will have feature lines in groups of loaded Geometry file. Default is a new Line entity per part
-stl_tol 0.001	Tolerance for stl files

Mesh

-acusplit	Will split tets for use in Acusolve. Could be used in other solvers if desired
-cartsurf	Will not include hexas on the surface when using Cartesian meshing
-fea	Allow a quad to be next to two tris. This is useful for FEA
-fillall 4	Repeat autofind command on all holes 4 times. The autofind command runs once by default in a standard run. Each autofind command will then attempt to fill the holes that were found using the –fillhole limit. Do not use unless experienced Harpoon user
-fillhole 5	Will autofix holes with 4 edges to them. Do not use unless experienced Harpoon user
-firstgeom	Will select the first geometric part as a wall if multiple parts overlap
-keeplevel	Keeps levels that were assigned if the part is set to farwall. If not used, then Harpoon will try and minimize the number of cells on all parts marked as farwalls
-lastgeom	Will select the last geometric part as a wall if multiple parts overlap
-maxskew 0.98	Removes all tets above 0.98. Do not use this unless experienced user of Harpoon. Use the smoothing function within Harpoon
-nodal	Will force Harpoon to use the nodal spacing of the input geometry to assign levels when using the AUTO mode
-nofixhnode	Turns off hanging node reset. Might be useful when exporting Poly meshes. Not recommended for traditional Hanging mode mesh exports
-nokeepvolname	Does not attempt to keep volume name when updating the mesh
-partpervol	Separates surface mesh into multiple parts if attached to different volumes
-UTM	Will help separate out the volumes in complicated meshes. External, Internal and All meshes can use this option

Boundary Layer

-blboth	Allows BL on both sides of a baffle
-blbulge	Stops a bulge on a non-BL surface when attached to a BL surface
-noblchk	Turns off BL check. Useful for BL cells with aspect ratio > 500
-noblsmooth	Turns off the surface smoothing in the BL creation

Exports

-2ndorder	Exports nastran as 2 nd order elements
-cfxvol	Exports cfx with full volumes
-export	Allows multiple format exports in batch
-hyper	exports optistruct format when nastran export is called
-little	Exports Star-CD, CFX5, Cobalt and CFD++ as little endian (default is big endian)

Miscellaneous

-batch <filename>	Runs Harpoon in batch. See batch section for details
-dlists	Uses display lists to render the geometry and the surface mesh
-fastdisplay	Displays all imported data as bounding box. This is useful for large input data. (See also Preferences -> Part attributes)
-guibatch <filename>	Starts GUI and runs the cfg file <filename>
-geomvis	Turns off geometry after meshing
-nostartscreen	Will not display Harpoon logo at start
-oldauto	Uses the auto calc in from versions before 3.5
-token_interval 5.0	Check for a free token every 5 minutes
-token_wait_for 60.0	If cannot get a license token, keep trying for 60 minutes. Eg "-token_interval 5.0 -token_wait_for 60.0" will check for a token every 5 minutes for 60 minutes and then stop
-version2	Use version 2.5b meshing algorithms. Try only if v3.0 gives strange results
-winmem	Use if windows runs out of RAM. This preallocates a large amount of RAM to avoid the memory fragmentation that can happen on Windows
-memfactor 0.8	Use if -winmem fails. Values between 0.1 and 1.0. A value of 0.8 will pre-allocate 80% of what -winmem would demand
-X	Use the Mesa libraries supplied for rendering

See the preferences for additional options

Import Formats

To load in multiple files use wildcards eg *.stl.

StarCD

The Star import reads the .inp, .vrt, .cel and .cpl file. If the data is ascii then the .inp file is not needed. If the data is binary, then the .inp file needs the word 'binary' in it.

Example .inp file

```
binary
SHEL,1
CTNA,1,part1
BAFF,2
CTNA,2,part2
```

The STL and Fluent/Tgrid imports automatically detect whether the file is binary or not.

The Nastran file will read quads/tris and automatically extract any faces required from volume elements.

Export Formats

NB No extension added if the export produces a single file. Extensions are described where applicable

EnSight

Harpoon exports a binary EnSight6 file.
Byte swapping is done by EnSight.

Fluent

Harpoon exports a binary cas file.
Byte swapping is done by Fluent.

STAR-CD

Harpoon exports binary .cel, .vrt and .cpl files as well as the .inp file.
Byte swapping is done by Harpoon.

OpenFOAM

Harpoon exports .points, .faces, .cells and .boundary files.
Byte Swapping is done by OpenFOAM

CFD++

Harpoon exports binary cellsins.bin, nodesin.bin and exbcsin.bin files.
Byte swapping is done by Harpoon.

Cobalt/AVUS

Harpoon exports a binary Blacksmith file.
Byte swapping is done by Harpoon.

The other formats are ascii.

CFX5

Harpoon exports binary ICEM-CFX5 file.
Byte swapping is done by Harpoon

Acusolve

Produces acusolve.inp plus a MESH.DIR folder.
Byte swapping is done by Harpoon.

Gambit

Harpoon exports an ascii gambit neutral file.

DTF

Harpoon exports a binary file using the DTF libraries

Cedre

Harpoon exports an ascii cedre file

Nastran

Harpoon exports an ascii bdf file.

ANSYS

Harpoon exports an ascii prep7 file.

Harpoon machine ports

Below is a list of ports available:

Linux 32bit 2.4.18 core (Red Hat 8)

Linux 64bit 2.6.18 and 2.4.15 cores

HPUX11.0 64 bit

IRIX 6.5.16m 64 bit

Itanium2 64bit Altix

IBM AIX 5.3 64 bit

MAC OSX 10.3 32bit

Solaris 8 64 bit

Windows 32/64bit

6.0 Parallel Meshing Using MPI

Harpoon 4.0 allows the user to run meshes in parallel.

To try the parallel version, please contact your Harpoon distributor.

Geometry Examples

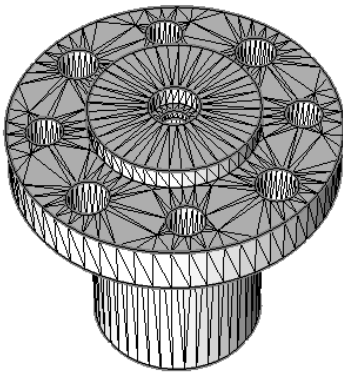
G1 Importing CAD

Harpoon now supports CAD interfaces for IGES, Catia V4 and Unigraphics CAD models. These interfaces are available upon request and interested parties should contact their local distributors for information on how to access these features.

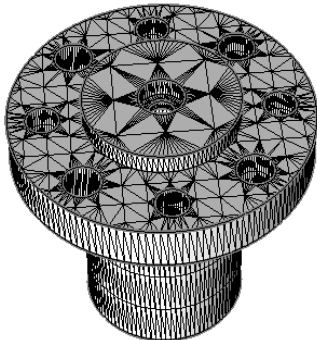
During the import of CAD models, Harpoon will tessellate the models based on CAD Preferences set in the users preference menu. The default settings should be suitable for most models, but the effect of the settings is described below. These settings are used in the import of all CAD types.

The user should note that changing the settings can have a huge effect on the tessellation of a model and they should only be changed by a user who has knowledge of the dimensions of the model that is being imported.

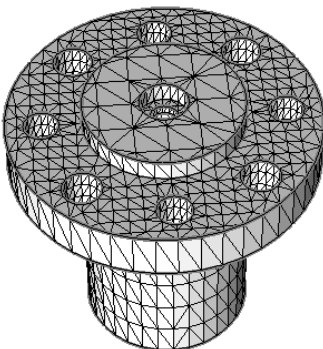
If the user imports the Catia V4 example model test.model in the examples directory, the default settings should produce the following tessellated geometry in Harpoon



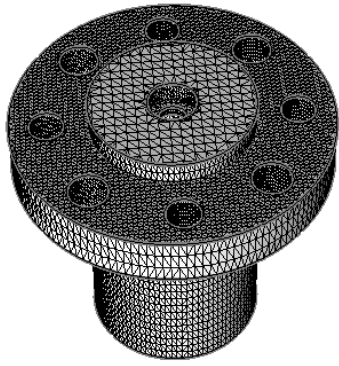
The following images illustrate the effect of turning off the default 'Auto Override' settings in the CAD preferences dialog and the effect of adjusting some of the parameters available to the user.



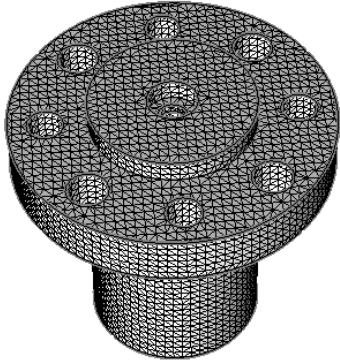
Decreasing the Chord Tolerance causes the curved edges and surfaces to be caught with more accuracy, but the number of triangles in the model is increased.



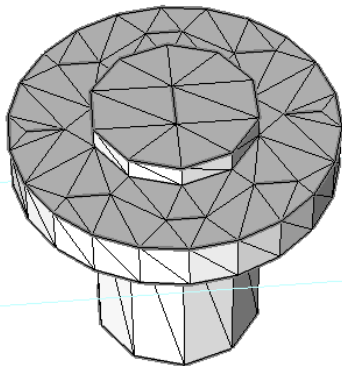
Decreasing the Edge Ratio causes the triangles to be much less sliver like in shape, but can also cause the number of triangles in the final model to be increased.



Combination of Edge Ratio and Chord Error Reduced. This image shows the joint effect of these two parameters, again capturing the curvature much more accurately but with the penalty of more triangles in the resulting model.



Reducing the Max Size.. This is another method for controlling the level of detail that is caught in the CAD model



Effect of increasing the Min Triangle Size. This parameter is useful for avoiding the tessellation of unwanted small artefacts in a model. As can be seen, the holes in the original model have been removed due to increasing the minimum triangle size to be bigger than the hole features

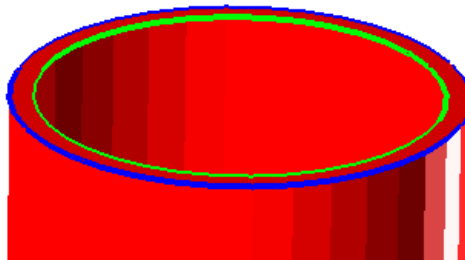
If an error occurs during the import of CAD data, the user should turn off the auto settings and as a first attempt, adjust the Initial Tessellation setting up/down according to the dimensions of their model to see if this helps in the tessellation. If problems still occur, the user should contact their local distributor for support.

G2 Line Selection/Separation

Lines are displayed in groups. In each group there may be more than one line segment (lines which are contiguous). This example explains an easy way of separating out lines without the problem of producing hundreds of line groups when using the 'Separate by Region' option.

Import the file thicktube.stl, provided in the download. Zoom in towards one end of the tube. Double click on the feature line on the inside of the tube (The image below shows the line segment to be split). At this stage all the lines will be coloured grey. Check you have selected a line correctly. (A line number is printed in the startup window and the Line folder should have a line group highlighted). If you have problems selecting the line segment try shading the geometry or making it invisible.

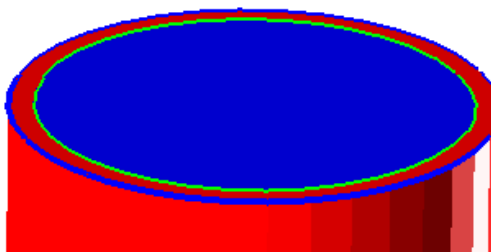
Select 'Separate-> Line Segment by Region'. Below is what you will see. The line segment has been placed in a separate group so it may have its own levels assigned or used as a template for hole filling



G3 Hole Filling using Line Groups

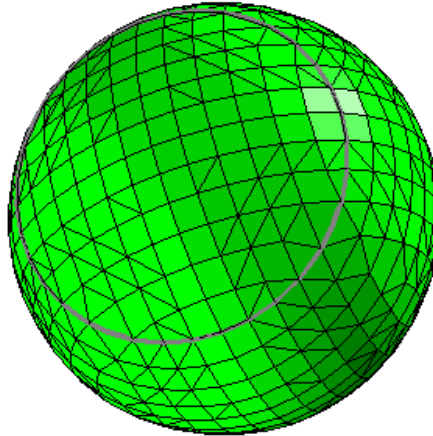
Follow [example G2](#) before continuing with the hole filling.

To fill in the hole at the end of the tube, select the required line group. In the case of example 5.11 it would be the green line shown. Press the **STL Modify** icon and then hit the 'Auto Fill hole' button. Harpoon will fill in the hole and assign the new triangles to the existing part. If you want these triangles to be in a separate part, you must make sure that the run time preferences has –fillnewpart turned on (Preferences-> Run Time Preferences). The picture below shows the new created part.



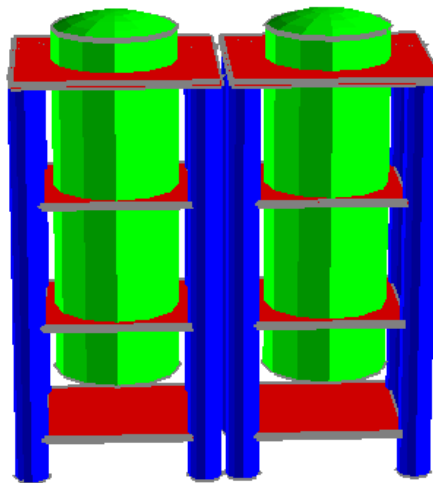
G4 Appending Feature Lines

Load the example file ball_line.stl and then append the file ball_line.fls. Set the base level to be 0.05 and hit create mesh. Below shows the ball with the line on it. Appendix C describes the format of the fls file used to import lines into Harpoon



G5 Merge Similar Parts

Load the example file similar.stl. Select the one of the 2 large tubes by double clicking on the part in the graphics window. Choose 'Geometry->Merge Similar Parts'. Repeat this operation but with the smaller tubes and then the flat plates. Below shows the 18 parts merged into 3 parts. Best results are achieved when selecting the larger parts first.



Meshing Examples

M1 Creating a Mesh

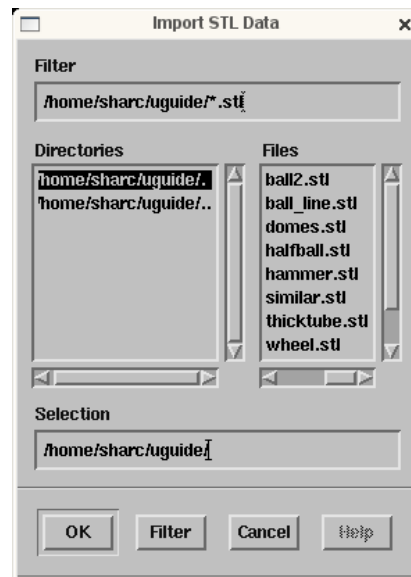
From the **File** Menu choose **Import→ STL**

This will bring up the dialog shown on the right. Click on the STL file of your choice and press the **OK** button.

The STL will then be imported into Harpoon

OR

Input *.stl to load all files that satisfy the wildcard.

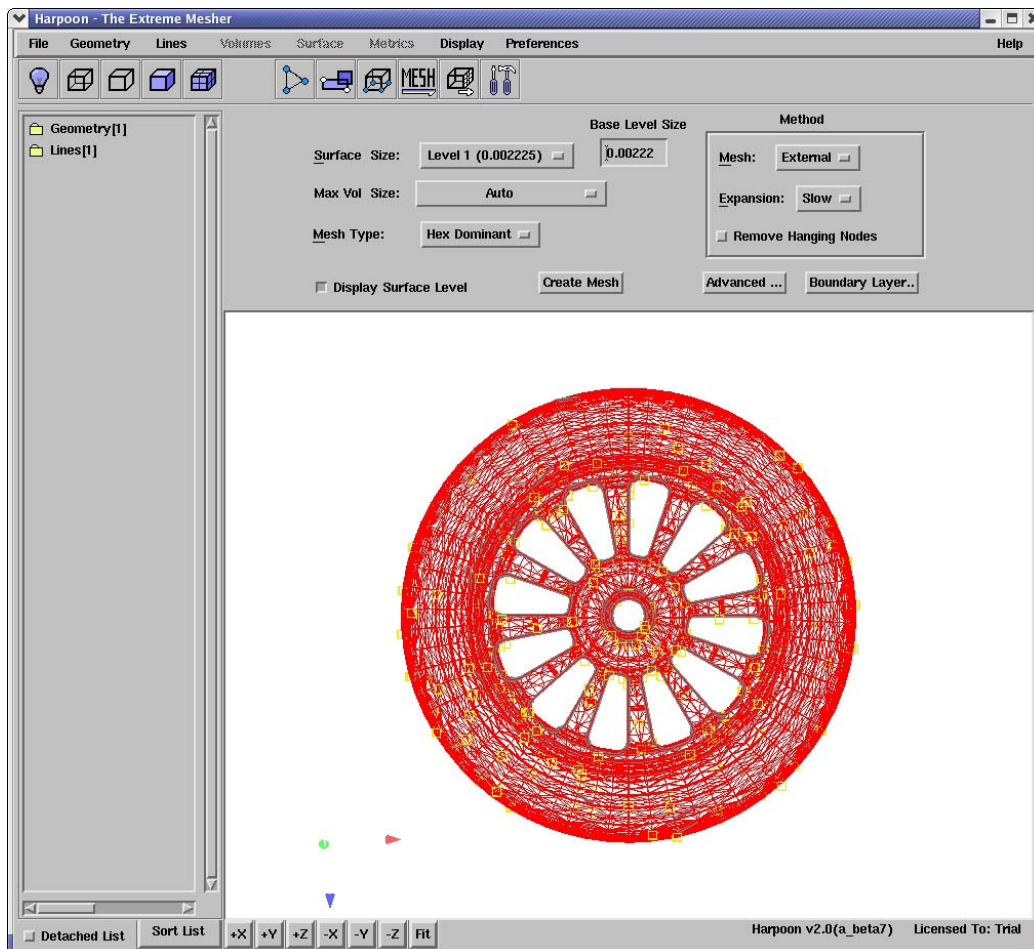


The STL Geometry will be displayed in the graphics area.

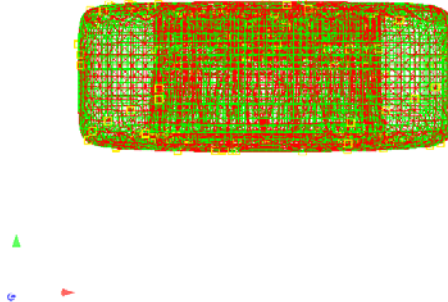
Next click on the Mesh Icon and your gui should now look like the image below.

By clicking on the 'Display Surface Level' Toggle you will see the initial size of the surface cells that will be created, shown as yellow hexas.

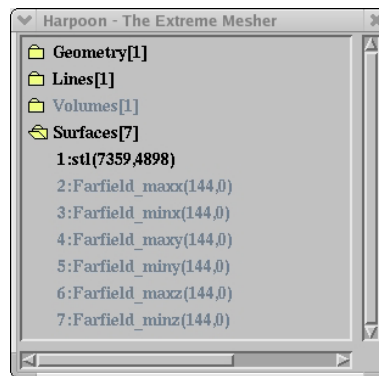
The size of surface cell can be adjusted by clicking on the 'Surface Size' menu and choosing which level is desired. The yellow hexas will automatically update in the graphics area to show the size chosen.



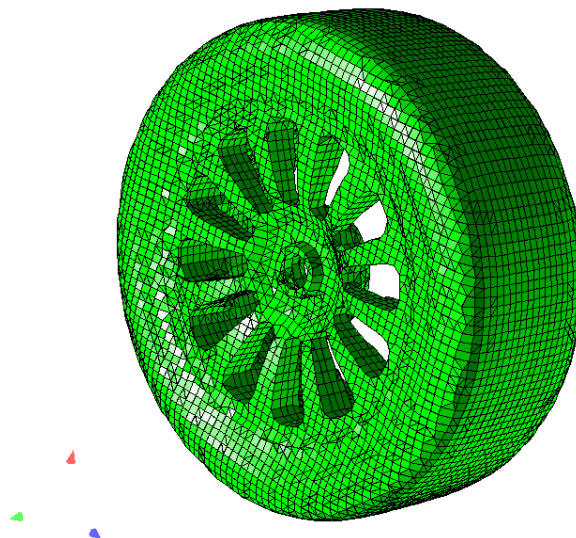
To create a mesh, simply hit the 'Create Mesh' button and Harpoon will start meshing the geometry. In this case, the default values of an External Mesh and Slow Expansion will be used. Once finished, you will see the mesh in the graphics area as shown below.



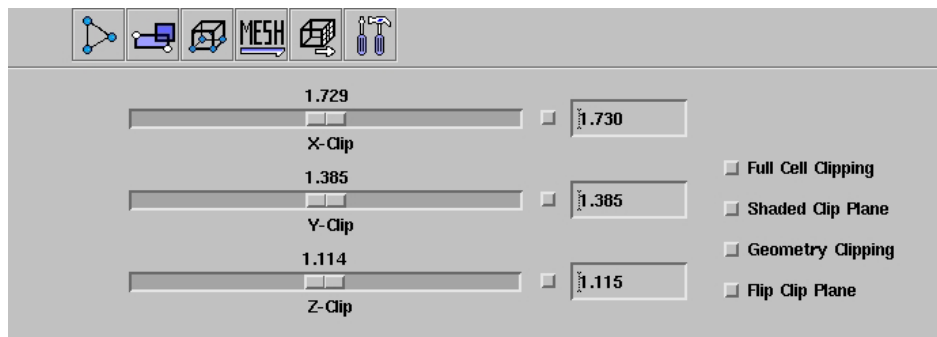
The farwalls are turned off but can be turned back on by selecting them in the Parts list



Next highlight the STL Geometry in the Parts List and turn its visibility off. You can now zoom in on the model to view the surface mesh created on the geometry as shown below

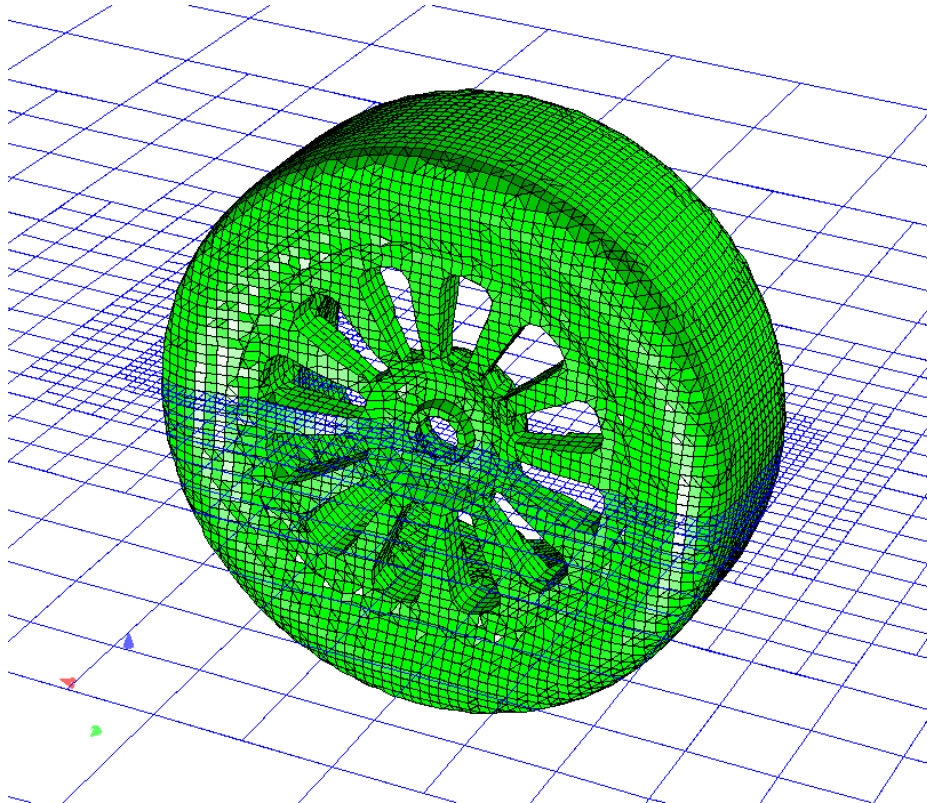


To get an idea of the mesh created in the volume, you can create a clip through the volume region. To do this, click on the Clip Icon.



To generate a clip, simply click on any of the On/Off toggles. In this case, we have chosen the Z-Clip.

This will create a clip as shown below

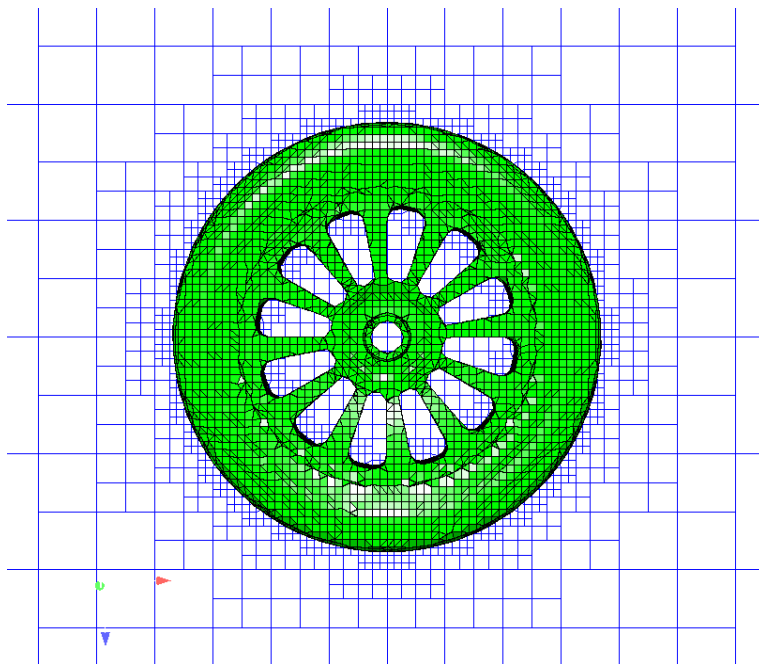
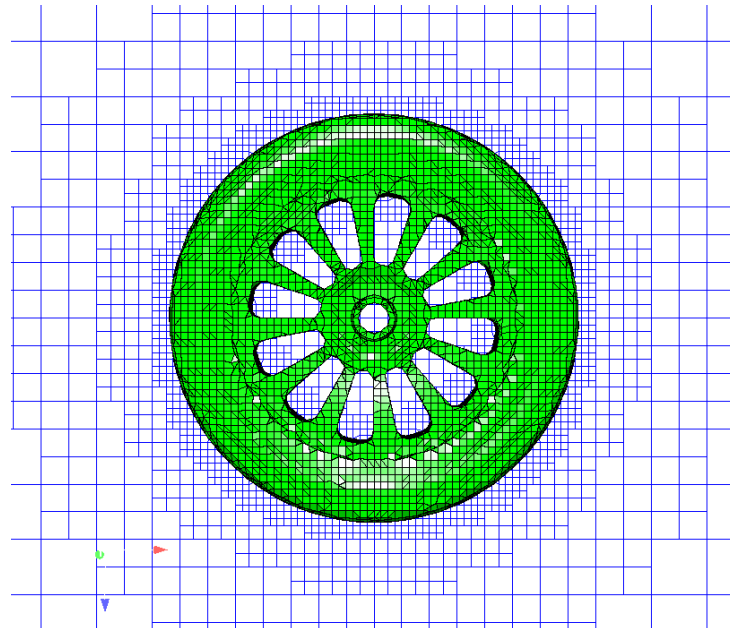


The clip can be moved interactively by simply dragging the slider bar.

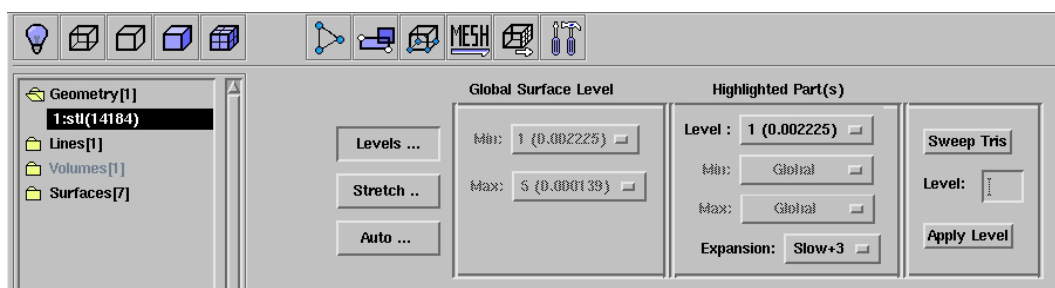
M2 Slow/Fast Expansion

In the Meshing Dialog Area, you will see a button labelled Expansion:

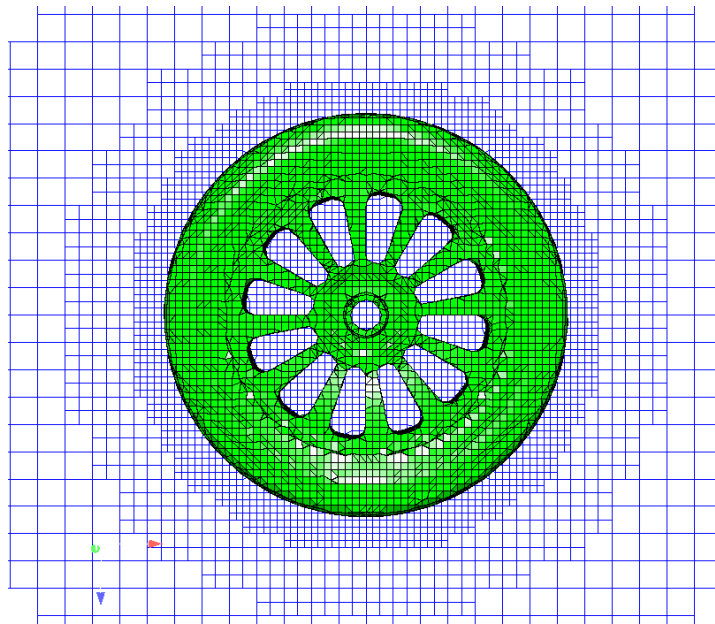
This controls the rate at which the volume mesh grows from one hexa size to the next. There are two options, Slow and Fast. The two images below show the differences between the two. The Fast expansion produces less cells in the volume.



To have more control over the expansion, go to the Advanced area, select a part and change the expansion value to **Slow+3**. The +3 relates to (roughly) the number of cells that will occur before transition.



Got back to the main meshing area by hitting the mesh icon and hit 'Create Mesh'. Below is the result.

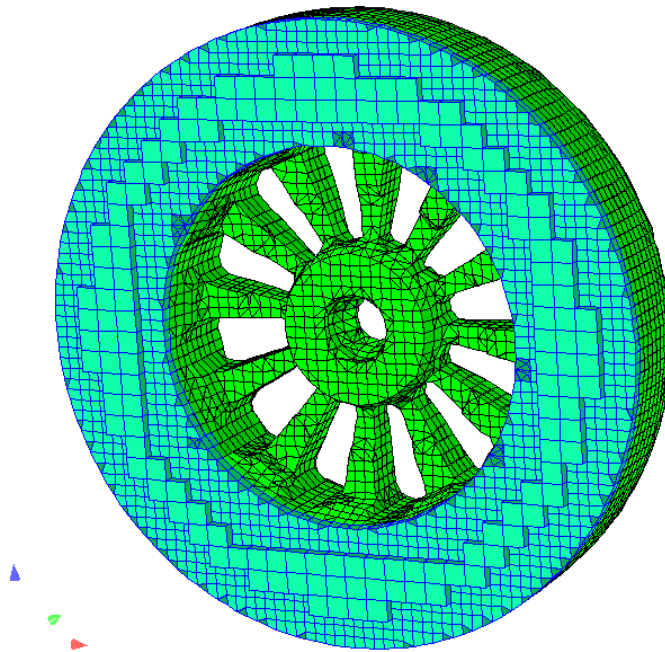


M3 Meshing Internal Geometry

Harpoon can be asked to mesh only the internals of a geometry. To do this, simply select 'Internal' from the 'Mesh:' button in the Meshing Dialog area.

Then hit the 'Create Mesh' button and Harpoon will generate a mesh on the inside of the geometry.

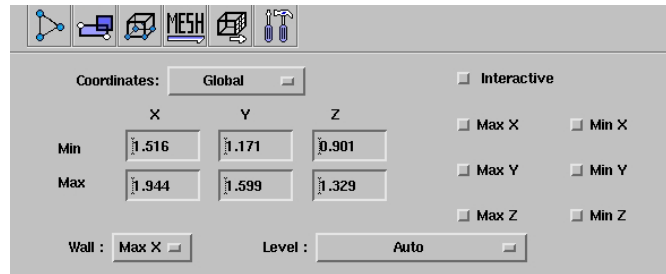
Creating a clip of the internal volume after meshing is sometimes difficult to see. A method of helping this is to click on the 'Geometry Clip' toggle in the Clip Dialog Area which will clip the geometry to allow the user to view inside the model. An image similar to below will result.



M4 Moving the Far-Field

Sometimes the user will need to move the position of the farfield bounding box. By default Harpoon will position the model in the centre of the bounding box. An example of this being incorrect would be for a car, where the user would want to position the model on the floor.

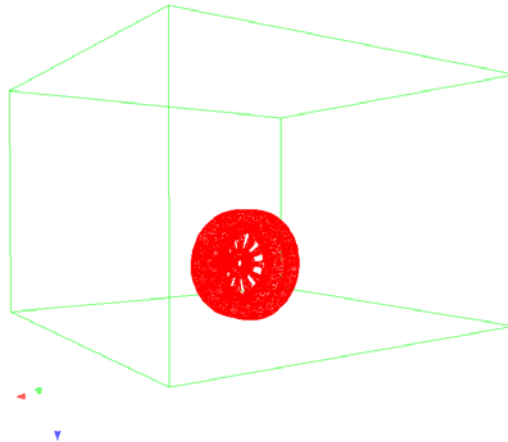
To manipulate the Far Field, click on the Far-Field Icon which will bring up the dialog area below



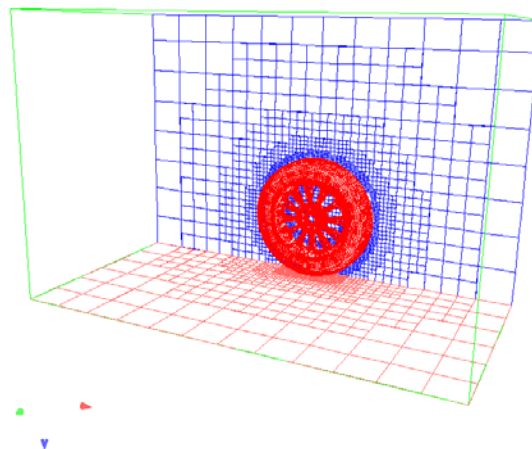
	X	Y	Z
Min	1.516	1.171	0.901
Max	1.944	1.599	1.329

Wall : Max X Level : Auto

A quick method of moving the bounding box is by clicking on any of the toggle buttons on the right hand side. This will 'snap' the bounding box to the models Min or Max direction. In the example below, we have 'snapped' the bounding box to the Max Z position of the model



To highlight the change in position, below is a picture of a clip through the volume after meshing for the above example



Another method of positioning the bounding box is to use the 'Interactive' button. By clicking this toggle button, three white markers will appear on the bounding box.

The two markers on the corner of the bounding box can be selected with the mouse and moved interactively to change the outer bounds of the box. The centre marker can be picked to interactively translate the whole box.

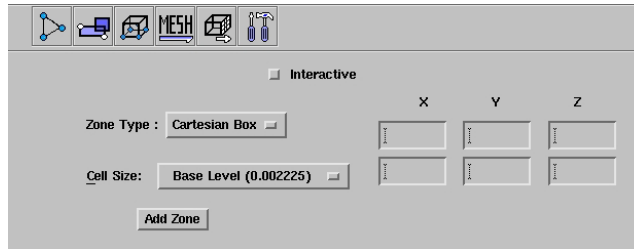
Alternatively, the user can type in his own coordinates into the text boxes which are updated once the user hits the 'Enter' key on the keyboard.

M5 Refinement Shapes

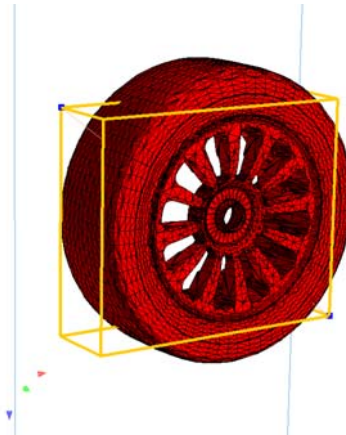
ADD TRAPEZOID EXAMPLE

There will be times when the user wants to have control of the size of the cells in the volume of the mesh. To do this Harpoon uses refinement areas, which can be either a box, sphere or trapezoidal box with all corners moveable.

To add a refinement area to your model click on the Refinement Icon. This will bring up the dialog shown below.



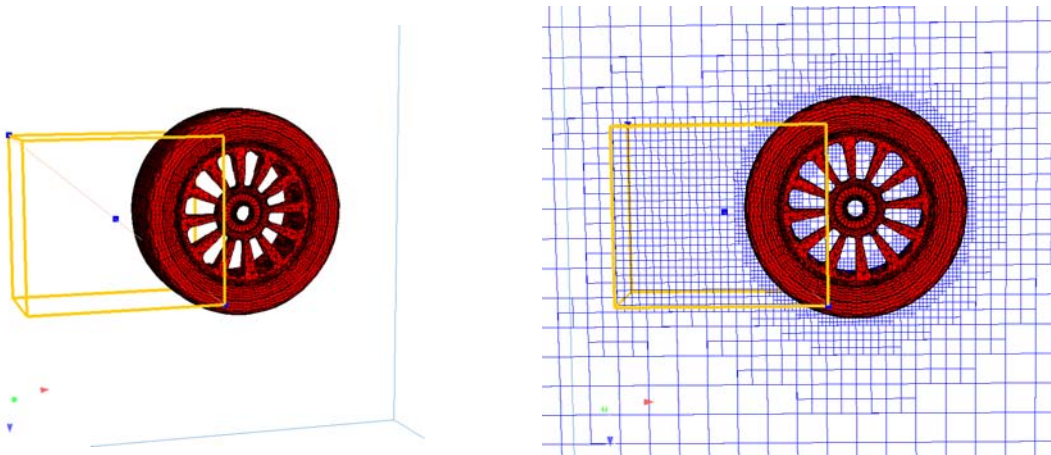
Next, click on the 'Add Zone' button. Your cursor will change to a cross-hair and you should then click two points on the model using the left mouse button. These two points will define the extremities of the shapes position. Don't worry that the shape is the wrong size or in the wrong position. You will easily be able to change these once the shape is created. The image below shows an initial shape (default box) created by clicking the two points shown



Now that we have an initial shape, we can change its size and position by clicking the 'Interactive' button. By picking the blue handles on the corner of the box with the mouse, we change its size and position. Picking the centre blue handle inside the box, will allow the user to translate the whole box.

Remember to turn off the Interactive toggle once the shape has been positioned.

The images below shows our refinement shape moved to behind the model and the final mesh.



The shape, in this case a box, has a default level of the base level. To change this expand the Refinement Zones folder in the parts list and select the Shape. The user may now change the Cell size. In fact, the user may now change the type from box to any other shape or edit the text boxes to have more precise zone limits.

M6 Meshing Using Base Level

The default base level in Harpoon is Level 1. The user may change this to achieve a more desirable hexa size on the surface. Subsequent levels decrease the hexa size by a factor 2. Example:

If Base Level (Level 1) is set to 0.05 then

Level 2 will be 0.025

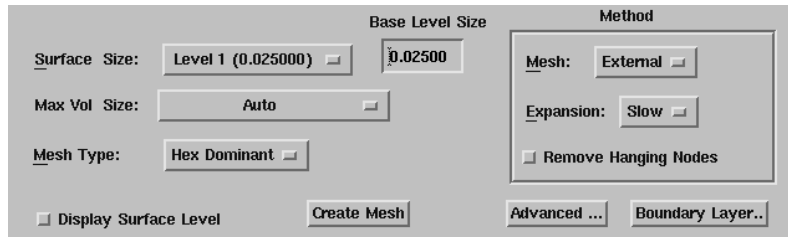
Level 3 will be 0.0125

Level 4 will be 0.00625

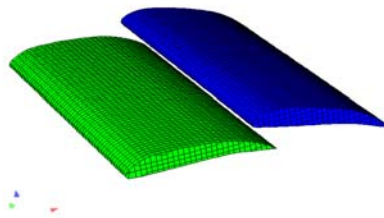
These values apply for Refinement Shapes, Max Vol Size and Advanced levels.

Single Level

Import the file wings.stl, provided in the download. To change the Base level to 0.025, enter that value into the Physical Size text box. Hit enter for Harpoon to accept (see below). There will be a confirmation note written to start up window. Hit 'Create Mesh'

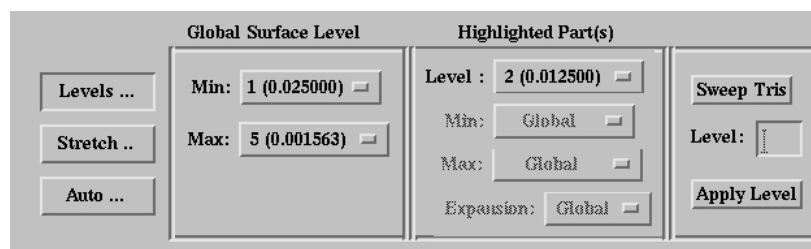


Below shows the resulting mesh

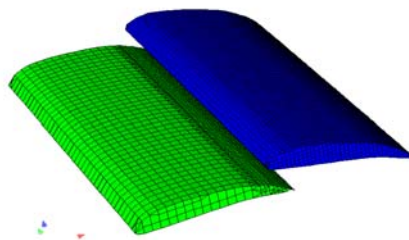


Multi-Level

Import the file wings.stl, provided in the download. Change the Base level to 0.025 by entering that value into the Physical Size text box. Hit enter for Harpoon to accept. There will be a confirmation note written to start up window. Go to the Advanced area and select part 2 and change the Highlighted Part level to 2 (see below)



Select the Mesh icon to bring you back to the main meshing area. Hit 'Create Mesh'. Below shows the resulting mesh.

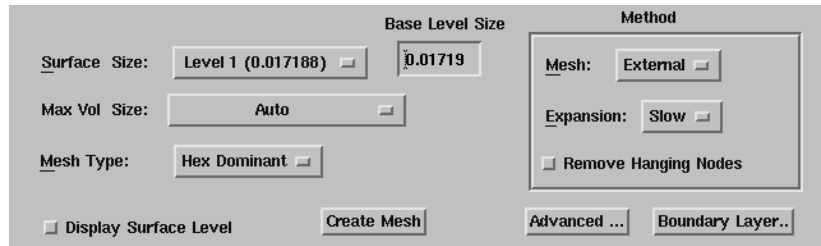


M7 Multi-level Meshing

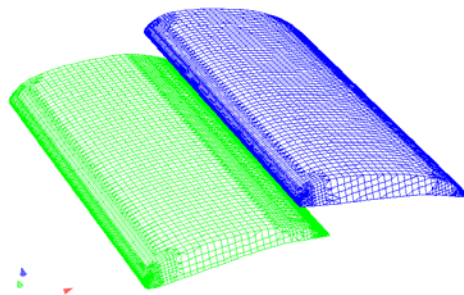
In many cases it is desirable for the size of the hexas to vary on the surface. Harpoon can automatically calculate the sizes and apply these during meshing or the user can input specific values.

Fully Automatic

Import the file wings.stl, provided in the download. Select the Mesh icon and change the Surface Size to Auto. Click 'Create Mesh'.

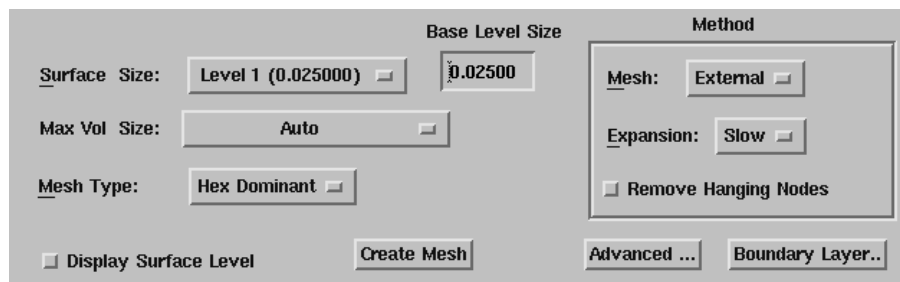


Below is the mesh produced by Harpoon using the Auto function

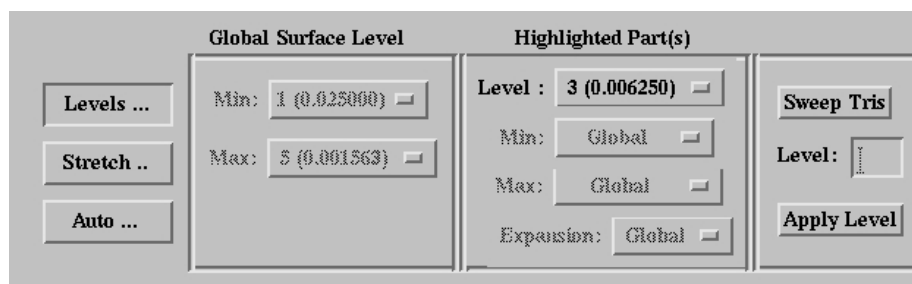


User control

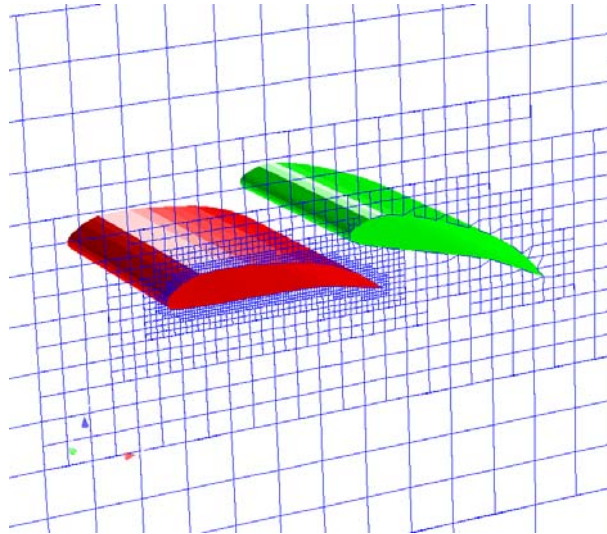
Import the file wings.stl, provided in the download. Select the Mesh icon and put 0.025 in the Base Level text box and hit enter (see next example for more information on Base Levels). Go to the Advanced area.



Select part 1 in the parts list and change the level to 3 (see below). Select the Mesh icon again to bring you back to the main meshing area. Click 'Create Mesh'

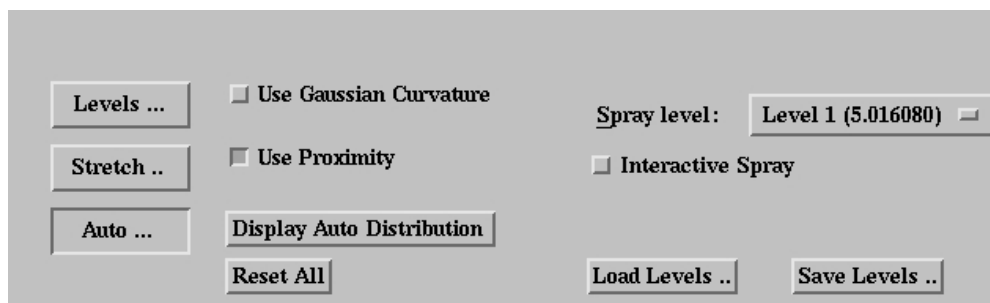


Below is an image of the mesh created. Note that the higher surface level expands onto the second wing. This is all automatic and the user need not worry about the finer cells expanding across different parts

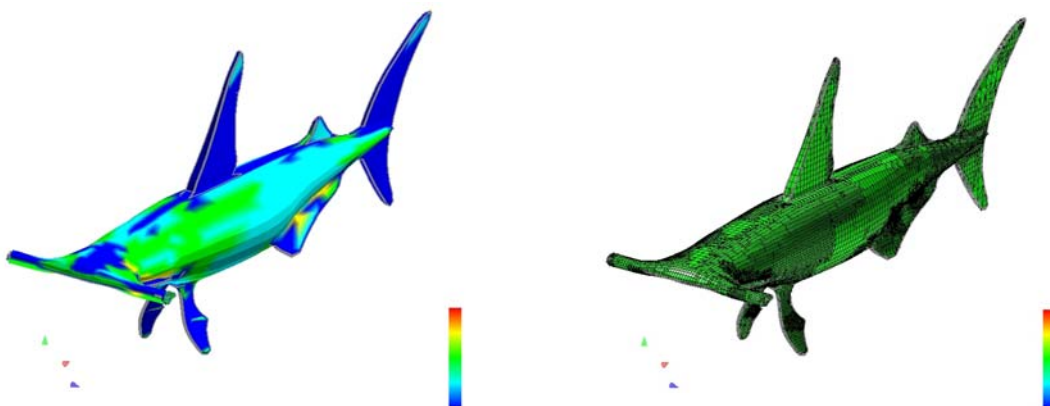


Auto Calculation

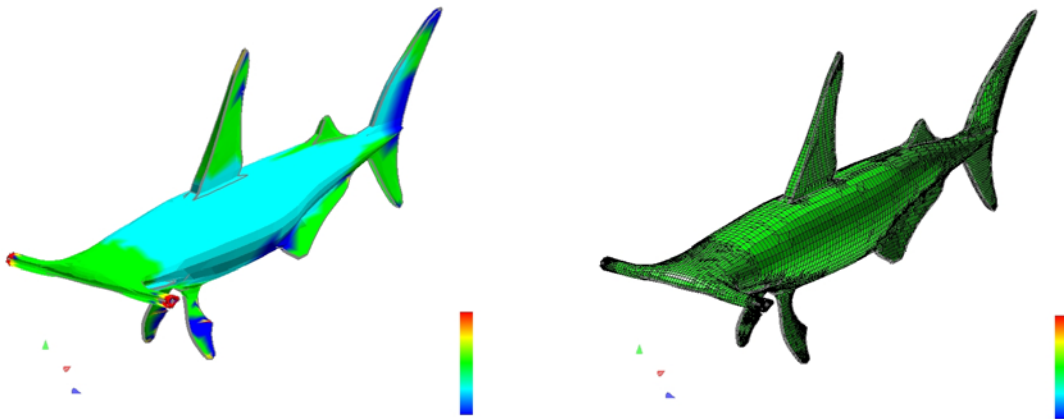
To change the AUTO calculation, select the Auto... button and choose any combination of the Gaussian Curvature and Proximity. If both are used, Harpoon will choose the smallest cell from both calculations for a given area of the geometry. If Gaussian is not used, Harpoon will use a default curvature calculation. Import the file hammer.stl, provided in the download. Select the Mesh icon and go to Advanced area. Select the Auto... button and make sure both the Gaussian Curvature and Proximity are turned off.



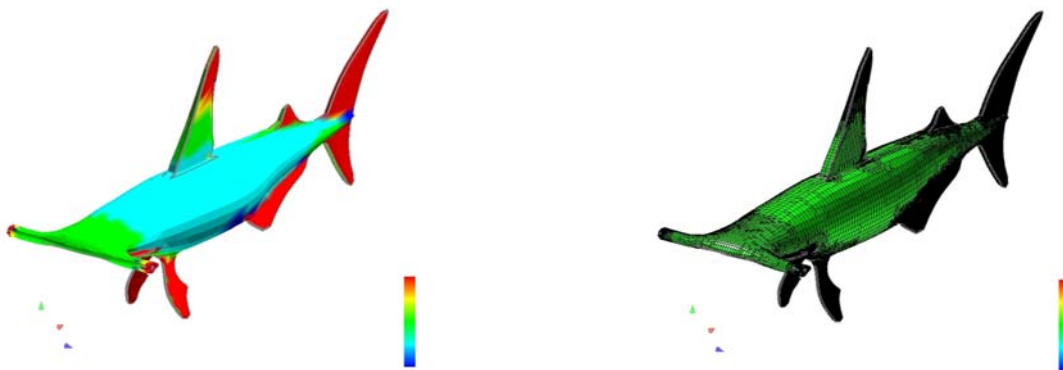
By clicking on the 'Display Auto Distribution', the user can see the level distribution that will be used if only the default curvature method is used. The distribution and mesh are shown below



If the user now selects the 'Gaussian Curvature' option and clicks the display distribution button again, Harpoon will recalculate the distribution with Gaussian curvature and display the results as shown below. Note how the Gaussian curvature distribution differs from the 'default' curvature distribution, especially around the head and eyes of the shark.

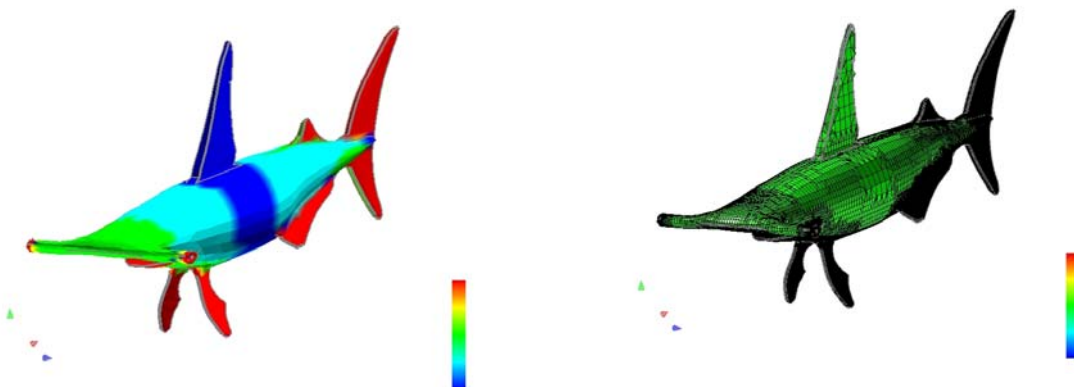


Next, selecting the 'Proximity' option and clicking on the display distribution button, will cause Harpoon to recalculate the level distribution with Gaussian curvature and a proximity calculation. This results in the distribution and mesh below.



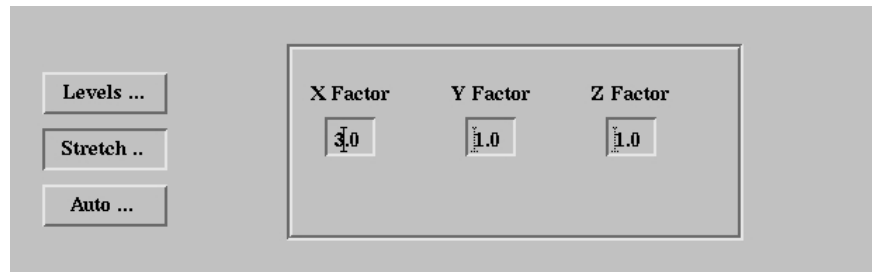
Note how the 'Proximity' option has chosen higher levels where walls are very close to each other such as the walls on the opposite sides of the tail.

All of the previous methods for calculating distributions can be over ridden by using the 'Spray Shop' feature. Here the user simply selects which level he would like to 'spray' the model with and then select 'Interactive Spray'. You can then move the cursor over the model as if it were a spraycan of paint and watch the levels get updated to that levels colour. In the example below, the dorsal fin and mid section have been sprayed to a level of 1 (blue).

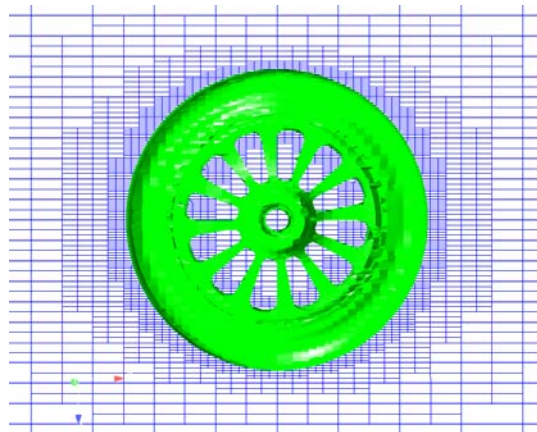


M8 Stretching Cells

Import the file wheel.stl, provided in the download. Select the Mesh icon again to go to the main meshing area (change base level to 0.001), then the Advanced area and finally the Stretch area. Change the X Factor to 3.0 and hit 'Apply Settings'. Hit 'Create Mesh'.

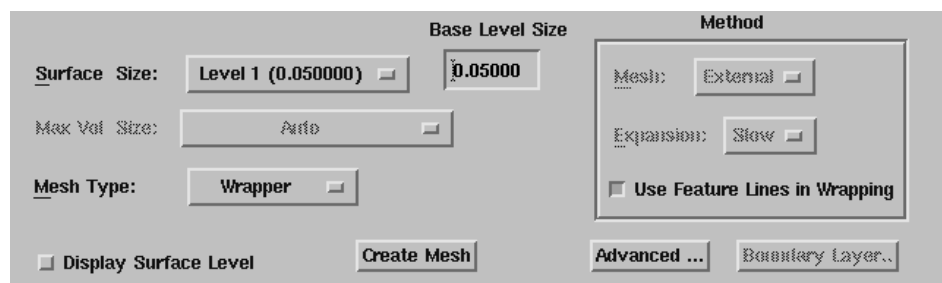


Below show the result.

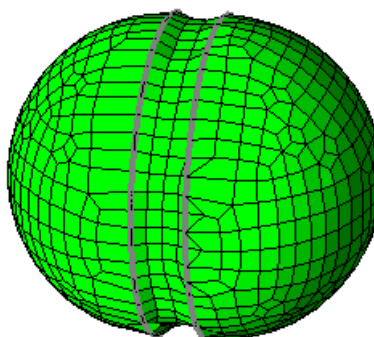


M9 Wrapping geometry

Load the example file domes.stl. Hit the Mesh icon and set the base level to be 0.05. Select Wrapped under the Mesh Type pull down menu and toggle on the “Use Feature Lines in Wrapping” button.



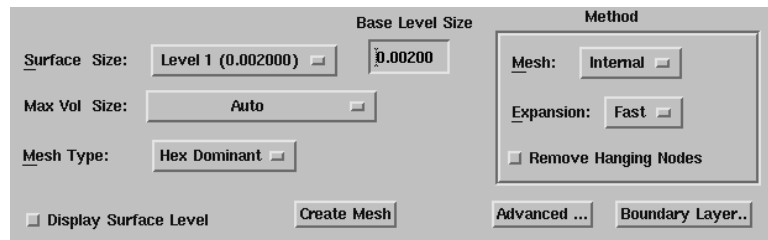
Hit create mesh. Below shows the resulting wrapped surface mesh



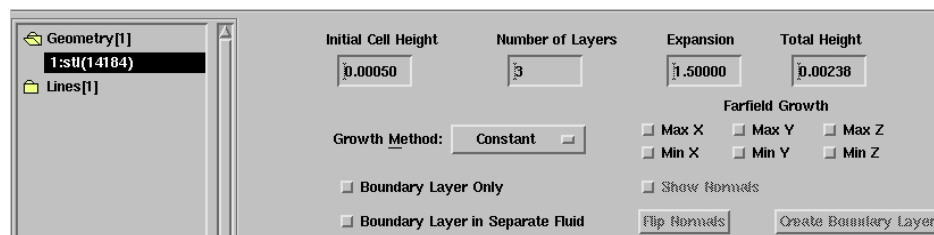
Boundary Layer Examples

BL1 Basic Boundary Layer

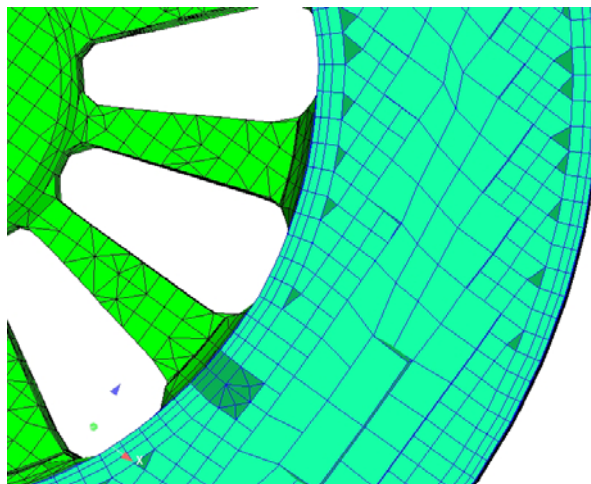
Import the file wheel.stl, provided in the download. Select the Mesh icon again to go to the main meshing area. Change the Base level to 0.002. Select Internal for the mesh method and fast for the expansion.



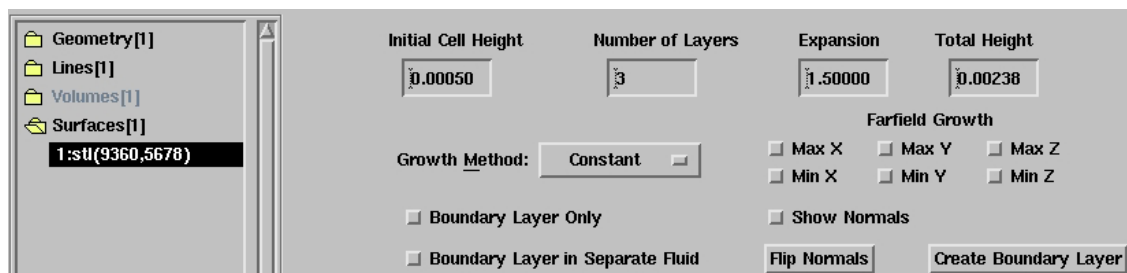
Select the Boundary Layer button. This will take you to the boundary layer setup area. Highlight part 1 and enter 0.0005 in the initial cell height text box, 3 in the number of layers text box and 1.5 in the Expansion text box. Hit enter for Harpoon to accept these values



Select the Mesh icon again to bring you back to the main meshing area. Click 'Create Mesh' Below shows the resulting mesh being clipped



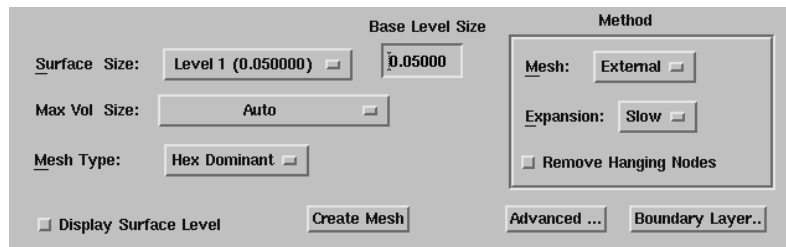
The same result may be achieved by creating the mesh without a boundary layer and then selecting the wheel part under the Surfaces folder. Input the initial cell height and number of layers, then click 'Create Boundary Layer'.



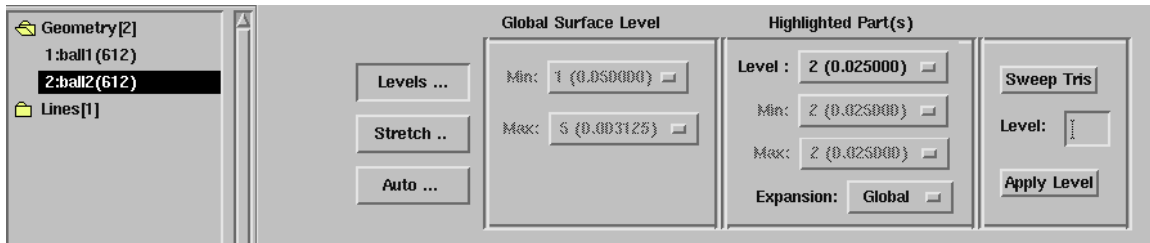
The exact values to use will vary from model to model. A guideline is to have max BL height of not more than 2 times the smallest cell on the BL part.

BL2 Boundary Layer with Multi Levels and constant First Cell Height

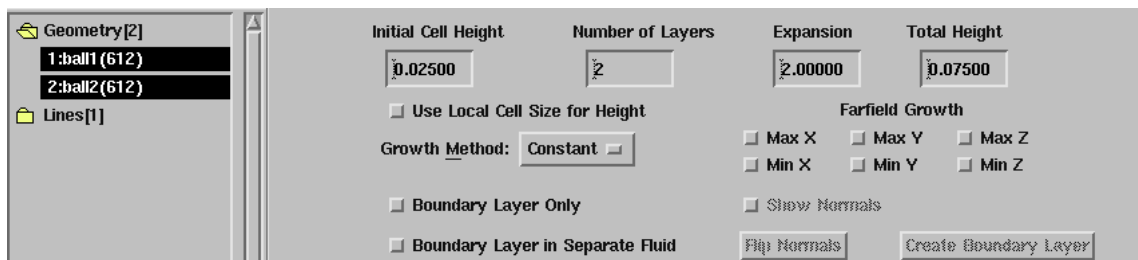
Import the file ball2.stl, provided in the download. Select the Mesh icon again to go to the main meshing area. Change the Base level to 0.05. Select External for the mesh method and slow for the expansion.



Highlight the second part, ball2, and go into the Advanced area. Change the level to 2.

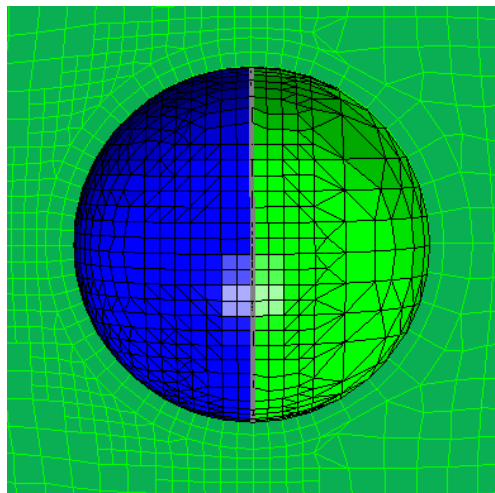


Select the mesh icon and then the Boundary Layer button. Highlight parts 1 and 2 and enter 0.025 in the initial cell height text box, 2 in the number of layers text box and 2.0 in the Expansion text box. Hit enter for Harpoon to accept these values.



Select the Mesh icon again to bring you back to the main meshing area. Click 'Create Mesh'

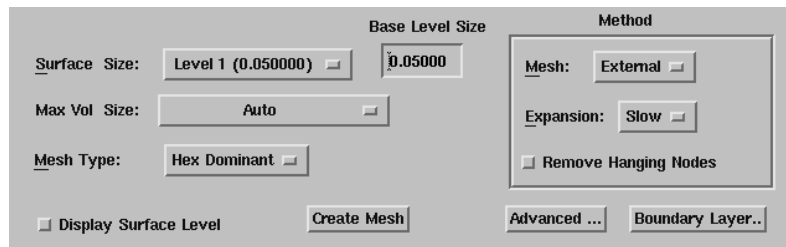
Below shows the resulting mesh being clipped



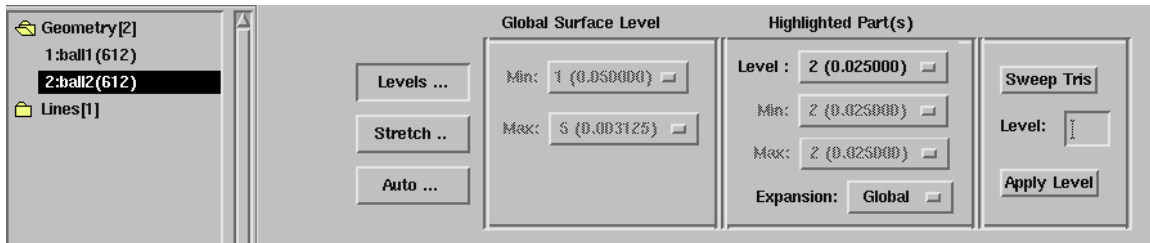
Note the first cell has been maintained. If the cells sizes of the non-boundary layer mesh were smaller, then the boundary layer would reduce in size. It is possible that the first cell height would then have to decrease

BL3 Boundary Layer with Multi Levels and Local Cell Height

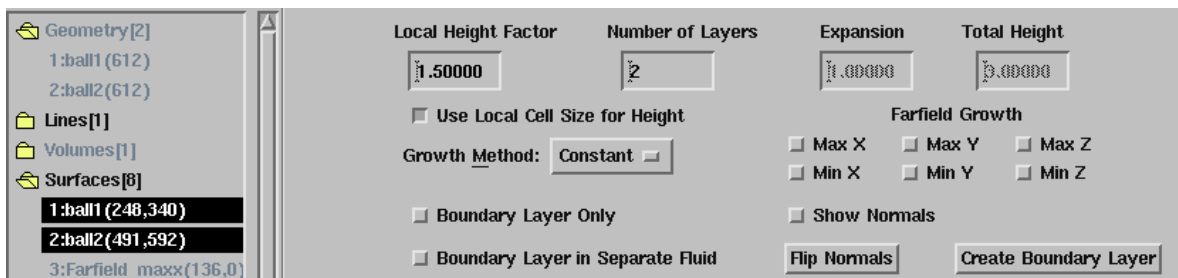
Import the file ball2.stl, provided in the download. Select the Mesh icon again to go to the main meshing area. Change the Base level to 0.05. Select External for the mesh method and slow for the expansion.



Highlight the second part, ball2, and go into the Advanced area. Change the level to 2.

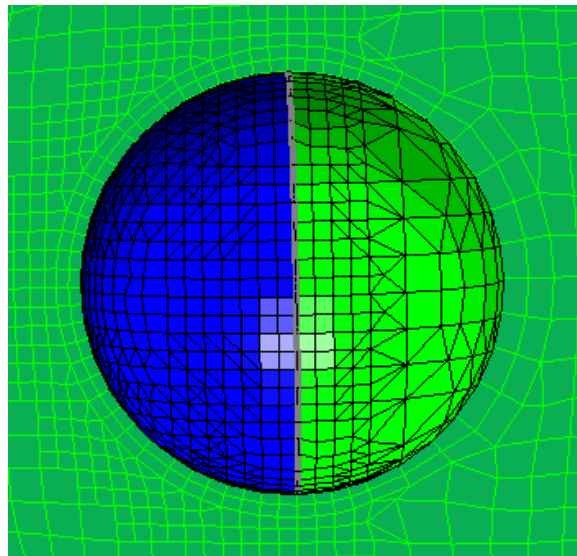


Select the mesh icon and then hit Create Mesh. Go to the boundary layer area and select surfaces ball1 and ball2. Select 'Use Local Cell Size for Height'. Note the wording above the 'Initial Cell Height' box changes to 'Local Height Factor'. Enter 1.5 for the local height factor and 2 for the number of layers



Hit 'Create Boundary Layer'

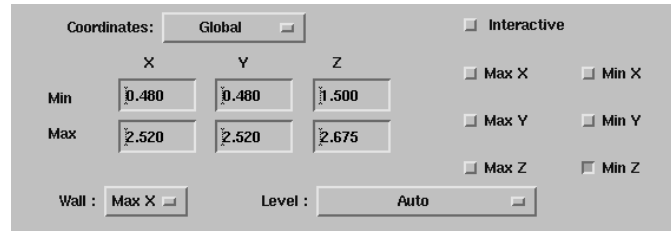
Below shows the resulting mesh being clipped



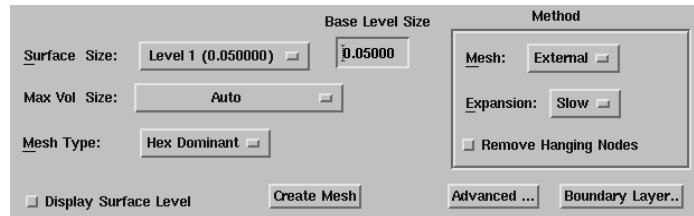
Note the first cell height is not constant. This is useful for complex multi-level meshes

BL4 Boundary Layer on Farfields

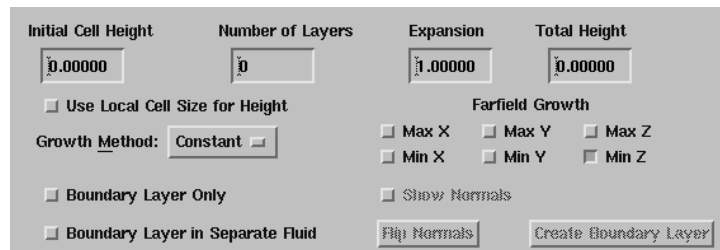
Import the file halfball.stl provided in the download. Select the Farfield icon and select the 'Min Z' button



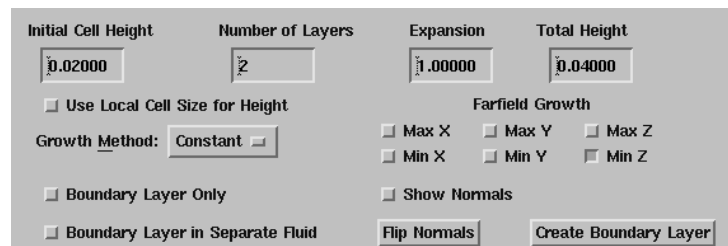
Select the Mesh icon to go to the main meshing area. Change the Base level to 0.05. Select External for the mesh method and slow for the expansion.



Select the Boundary Layer and turn on the Min Z button under 'Farfield Growth'. This will remove hanging nodes on the min Z wall during meshing. If you will not have hanging nodes on the min Z wall, then the button does not need to be turned on

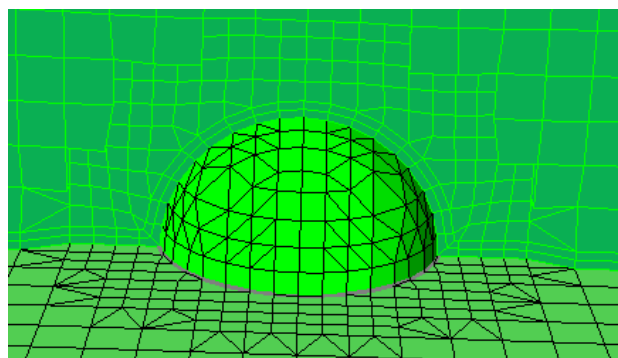


Select the Mesh icon to go to the main meshing area and hit 'Create Mesh'. After meshing select the surfaces 1 and 7. Go to the Boundary Layer area and choose 0.02 for the initial cell height and 2 for the number of layers. Hit 'Create Boundary Layer'



Below shows a the resulting mesh.

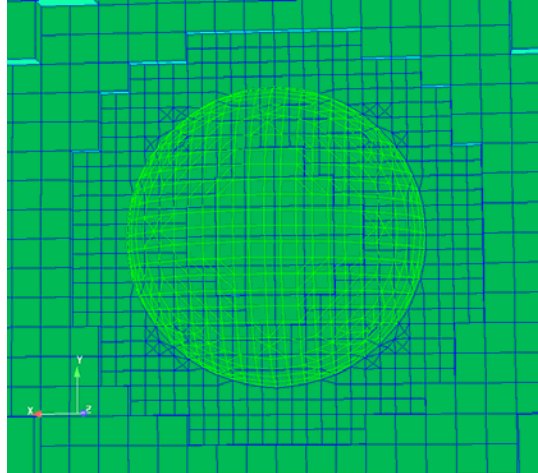
Note that boundary layers on farfields can only be created using the gui.



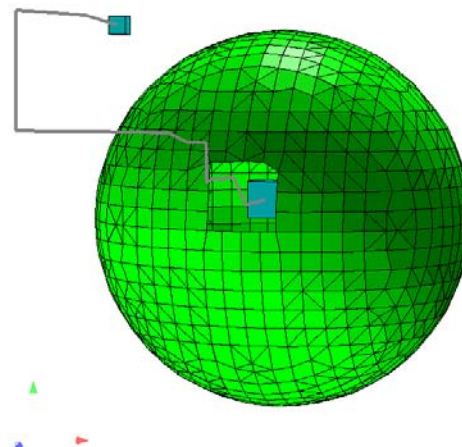
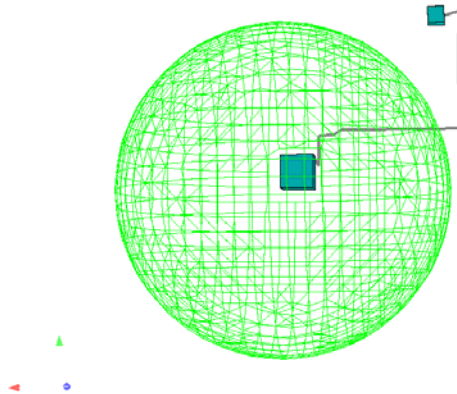
Mesh Modication Examples

MM1 Hole Finding using the Mesh Trace

Import the file xball.stl, provided in the download. Select the Mesh icon again to go to the main meshing area. Change the base level to 0.03. This change is only so that Harpoon produces less cells. Hit 'Create Mesh'. Go to the Clip icon and create a clip in Z. Go to the Mesh Fixing icon and then into the Mesh Trace area. Select (using 'P') two cells, one inside the ball and one outside it. Hit 'Create Trace'.

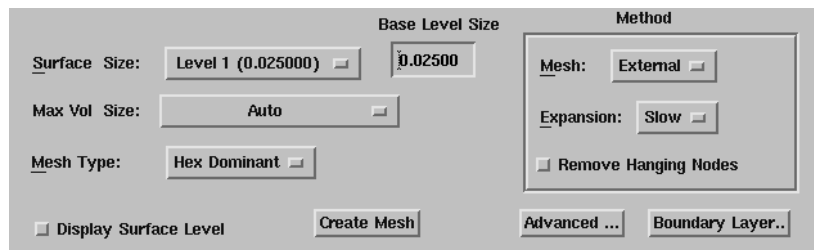


Below shows the route between the two chosen cells. This is very useful with very complicated geometry.

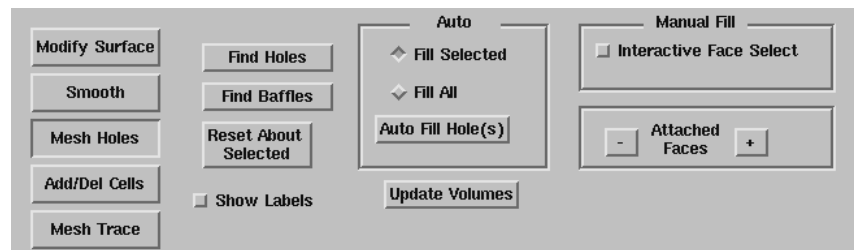


MM2 Filling Small holes

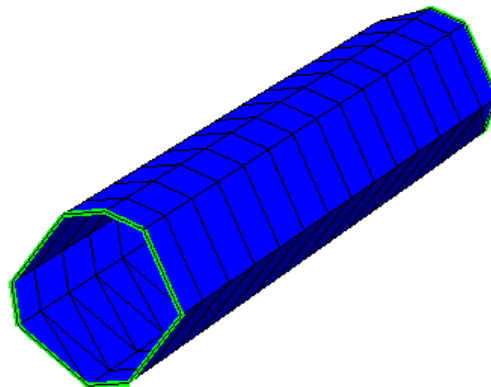
Import the file tube.stl, provided in the download. Select the Mesh icon to go to the main meshing area. Change the base level to 0.025. Ensure that the mesh is external and slow expansion. Hit 'Create Mesh'



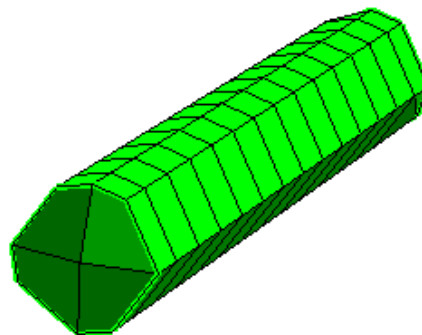
After meshing to the Mesh Tools icon, select 'Mesh Holes' and then hit 'Find Holes'



The mesh will have green holes at both ends of the tube.

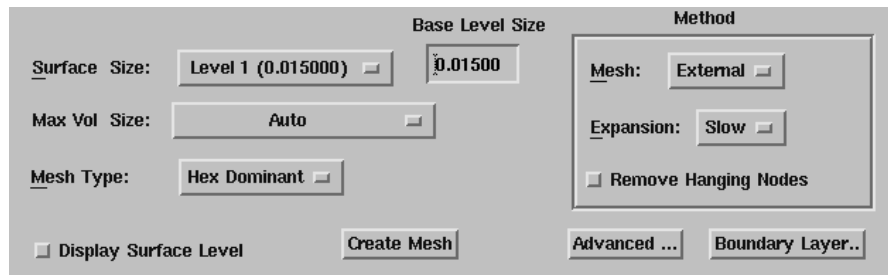


Select Fill All and hit 'Auto Fill Hole(s)'. When the holes have been filled, press 'Update Volumes'. This is essential to create an exportable mesh. Note the number of Volumes in the parts list has gone from 1 to 2.

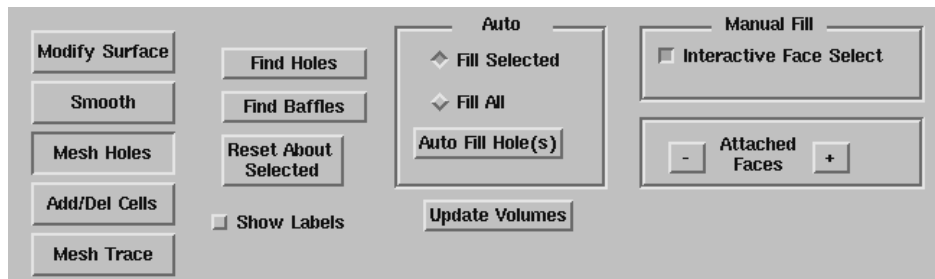


MM3 Filling Large holes

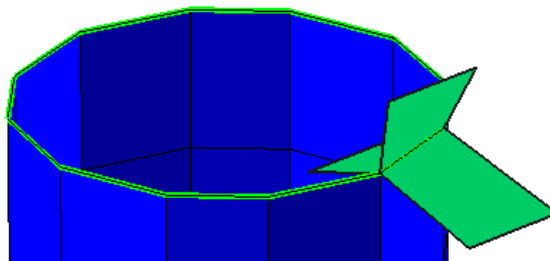
Import the file tube.stl, provided in the download. Select the Mesh icon to go to the main meshing area. Change the base level to 0.015. Ensure that the mesh is external and slow expansion. Hit 'Create Mesh'



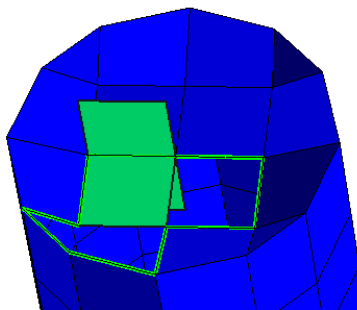
After meshing to the Mesh Tools icon, select 'Mesh Holes' and then hit 'Find Holes'



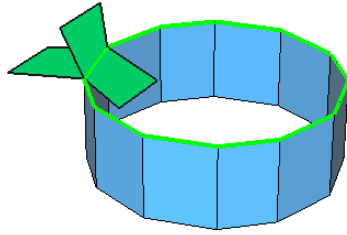
Expand the Holes folder in the parts list and select the first hole in the list. Turn on the 'Interactive Face Select' button. Green faces will appear on the hole. Move the mouse over the face you want to turn to a wall and hit the P key.



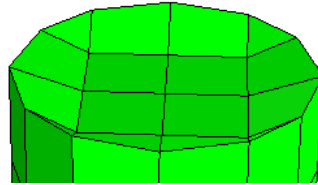
The green faces will move around until there is no hole left. Sometimes the green faces offered are not what is required. When this happens turn the 'Interactive Face Select' button off and then on again. This will offer a new set of faces. If nothing happens hit 'Find Holes' to give other face options



After filling the first hole, turn off all parts except the remaining hole. Select the hole and hit the + button next to Attached Faces. Now hit 'Fit'. This operation is useful to understand complex holes on large parts. Hit the + and – to see the temporary faces grow and shrink. Fill the hole in as above



When finished, hit 'Update Volumes' and the mesh is then ready to export



Appendix A

Equivolume Skew

Equivolume skew is only calculated on tetrahedra and triangles. It is a non-dimensional value and is calculated thus:

$$(\text{Optimum Cell Volume} - \text{Cell Volume}) / (\text{Optimum Cell Volume})$$

A value of 0 represents the perfect shape, whereas a value 1 of represents a degenerate cell/face.

Equiangle Skew

Equiangle skew is calculated for all cells and faces. It is a non-dimensional value defined as the maximum of the two following values:

$$\frac{(\text{max_ang} - \text{elem_ang})}{(180 - \text{elem_ang})}$$
$$\frac{(\text{elem_ang} - \text{min_ang})}{(\text{elem_ang})}$$

where

max_ang = maximum angle in the face or cell

min_ang = minimum angle in the face or cell

elem_ang = angle for an equiangular face or cell

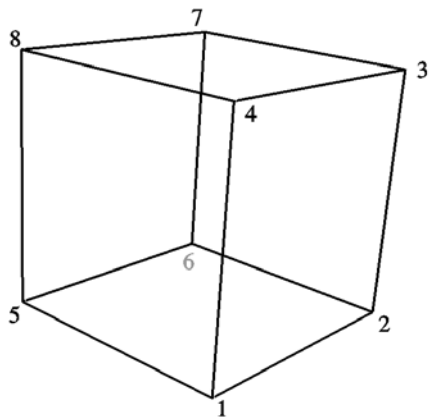
A value of 0 represents the perfect shape, whereas a value of 1 represents a degenerate cell/face.

Warpage

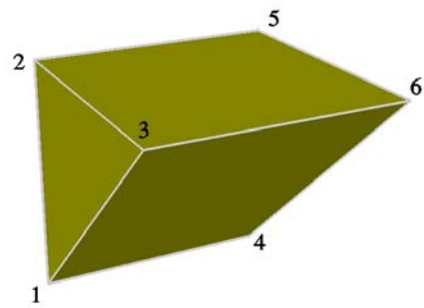
It is simply the angle of warp on a quad. Whilst manually smoothing, if the warpage of a quad on a given cell exceeds the preference warpage value, then the skew for that cell will be set to 1. This can be avoided by increasing the warpage preference value.

Appendix B

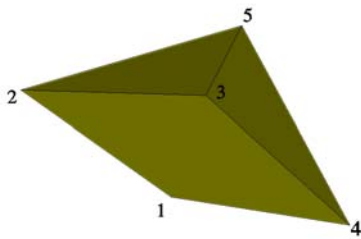
Windings for Cell creation



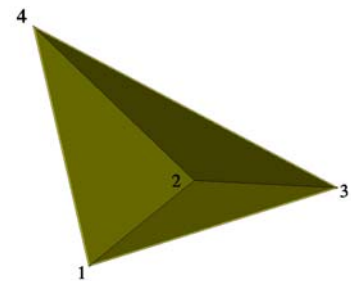
Windings for Hexas



Windings for Wedges



Windings for Pyramids



Windings for Tetrahedra

Appendix C

Line format

The format for lines which are to be imported is below. The file is free format.

```
36          ! Number of points
1.25160e+00 1.55141e+00 1.79154e+00
1.25160e+00 1.60125e+00 1.77818e+00
.
..
1.25160e+00 1.44859e+00 1.79154e+00
1.25160e+00 1.50000e+00 1.79604e+00
1          ! Number of Lines
Added_line !Title for line
36         ! Number of segments in line
    0      1 ! Connectivity of line segments
    1      2
.
..
    34     35
    35     0
```

Appendix D

Boundary Layer Preferences

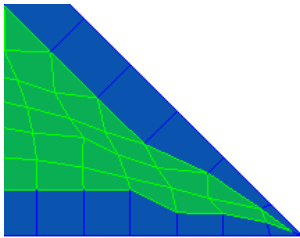
BL collapse

Angle 40

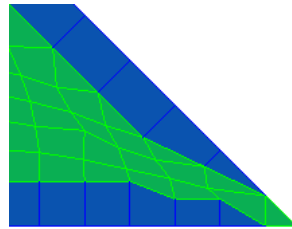
Neighbours 1

makes the BL stop if an acute angle of 40 or less is seen on the surface mesh

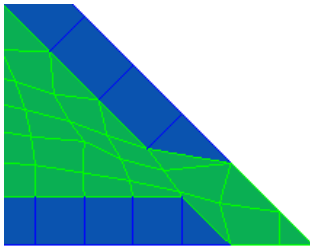
stops BL creation for every face that touches an acute angle below the above crease limit



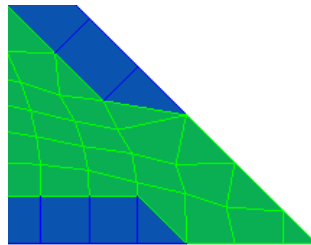
No collapse



Angle = 60



Angle = 60, neighbours = 1

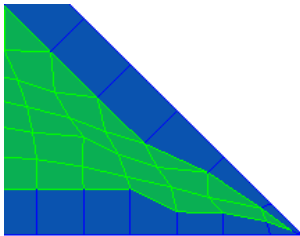


Angle = 60, neighbours = 2

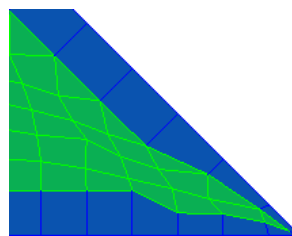
Distortion Limit

Limit 0.8

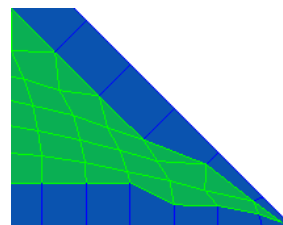
limits the amount of a quad maybe squashed



Limit of 0.94



Limit of 0.80



Limit of 0.5

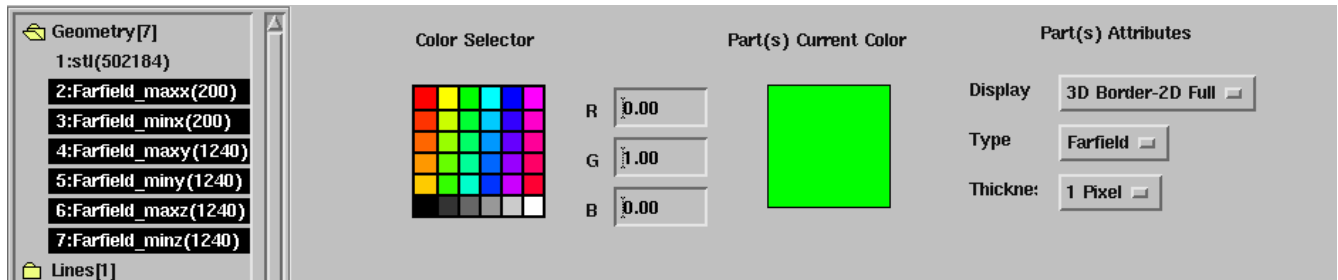
Appendix E

Meshing with own Windtunnel (Farfield)

It is always best to use Harpoon's farfields when meshing. If this is not possible, then Harpoon may take a very long time to mesh if care is not taken. There are 2 reasons for this which are:

- 1) geometry intersections
- 2) proximity calculations

If you need either or both then it is possible to avoid this time penalty by simply changing the type on your farfield. Select your farfield parts and go to 'Part Preferences'. Change Type to Farfield



Below are some timings in seconds

Geometry Intersections

Number of Input Tris	Type as unchanged	Type Farfield
1M	124s	16s
4M	1834s	70s

Proximity Calc

Number of Input Tris	Type as unchanged	Type Farfield
1M	89s	3s
4M	1623s	36s