

Harpoon Meshing Guidelines

Harpoon is unlike most other meshing packages you will have used in the past. To gain the most from the Harpoon software, we have listed below a few guidelines to help you in your evaluation. (These guidelines reflect Harpoon version 3)

General Guidelines

1. It is best to assign single levels (cell sizes) to individual parts/lines in the parts list.
2. Alternatively, (for example) assign level 3 to the surfaces and level 5 to the feature lines. Feature lines can be separated by feature and region just like the surfaces. You can then assign different levels to different feature lines. To select a feature line or surface from the screen, simply double-click the left mouse button on the surface or line and it will be highlighted in the parts list.
3. If using the 'Auto' option for cell size distribution, limit the number of levels that the calculation will use. By default this is levels 1-5. It is perhaps better to use 3-5 or 2-4 for example. This helps convergence in the solvers and usually gives a similar number of cells.
4. You will probably need to generate meshes that are larger than you would have previously created. This is a characteristic of automatic meshing. However, you will easily make up for the slightly longer solve time by the huge speed up in meshing time. In fact, due to the hex-dominant nature of Harpoon meshes, they will converge far quicker than conventional tet meshes and so your solve time may not increase even with a larger cell count.
5. In the current version, avoid asking for boundary layers with a total height larger than 2x the smallest cell size on that part.
6. If using your own windtunnel (farfield) follow Appendix E in the User Guide
7. If you are using Fluent, do not think that skew values of 0.98 or 0.99 are bad. Your mesh will run fine at these values. Use aggressive optimisation if you have convergence problems.
8. If you are using STAR-CD, increase Max Face Warpage from 40 to 50 (or even 60 if user happy to do so)
9. Read FAQ 21 "Tips on Convergence/skew" (http://www.sharc.co.uk/faq/faq_21.htm)

Boundary Layer (BL) Meshing Procedures

1. Assign BL levels to geometry or surface parts. For best results keep BL below 2x smallest cell height. No smoothing is required between meshing and BL creation but may help
2. Smooth at end of meshing. "Smooth all cells" works well followed by a target smooth (ie "smooth all cell" turned off)
3. For cells with remaining high skews, try hand-adjustment of nodes, in mesh fixing options, "Adjust Cell Nodes"
4. Compute and display Mesh Quality histogram. Display Worst Skew and hand modify these as desired.