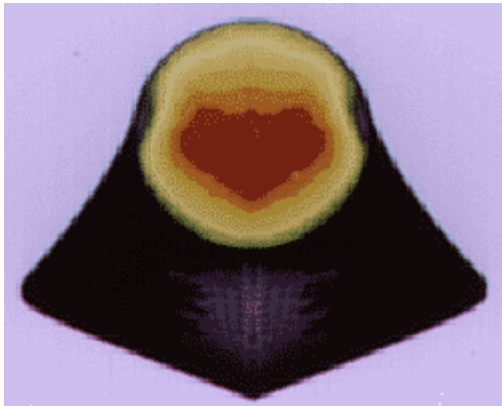


Sandia Uses CFD Software with Adaptive Meshing Capability to Optimize Complex Inlet Design

By Jim Nelsen, Senior Member of Technical Staff
Sandia National Laboratories, Albuquerque, New Mexico



Hybrid inlet total pressure contours at exit, created using an unstructured mesh CFD package.

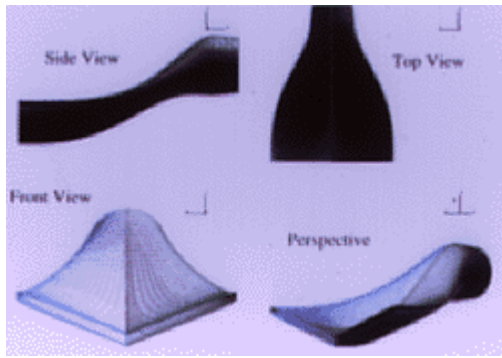
To reduce costly wind tunnel tests on the inlet of a missile engine, Sandia engineers wanted to refine the design with computational fluid dynamics (CFD) before progressing to physical testing. They ruled out structured mesh CFD packages because the complexity of the inlet geometry would have required days of meshing time. Instead, they used the unstructured mesh CFD package called RAMPANT from Fluent, Inc. of Lebanon, New Hampshire.

This software automatically generates a volume mesh using solution-adaptive triangular and tetrahedral elements. The mesh for the inlet took only four hours to complete and the resulting analysis saved \$100,000 in wind tunnel time by pointing out some problems in the initial design. More importantly, the use of RAMPANT provided the first opportunity to

use CFD on a problem with such complex geometry. (Since this analysis was performed in 1992, meshing times have been reduced even further. Meshing time today would be a matter of minutes, not hours.)

Throughout Sandia, one of the goals is to reduce testing because the lab's budget for testing has decreased. With flight tests costing about \$25,000 per test and wind tunnel time costing as much as \$20,000 a day, there is a great deal of incentive to move analysis further forward in the design process as a way of refining designs before proceeding to physical testing.

In the case of the missile system inlet, engineers needed to come up with a new design that would be "stealthy"--not detectable by radar--yet still provide good performance characteristics such as a uniform air flow with no separation. The air coming out of the inlet and into the turbine had to be uniform as well. In the past, this kind of problem would have been tackled using the transonic wind tunnel. At least two weeks would have been needed in the tunnel to test different configurations of inlet geometry to achieve the performance goals. The intent with the new inlet was to use CFD to study different design configurations on the computer instead of using the wind tunnel. However the design of the new inlet was quite complex. Instead of a standard rectangular channel, the cross-sectional area of the inlet varies from a high aspect ratio V-shape at the front to an



Isometric views of final design of hybrid subsonic inlet/diffuser

almost circular outlet at the back end.

Structured grid CFD codes were rejected for this problem because of the difficulty and time required to produce the mesh for this geometry. Nearly all commercial CFD codes employ either a simple Cartesian mesh that represents the world as a cellular-like structure or an assembly of curvilinear, mapped cubes. However, structured mesh generators are only semi-autonomous at best. And with an irregular geometry such as this inlet's, structured codes have trouble with the streamwise geometry transition and the boundary curvature. For this inlet, engineers estimate the meshing would have taken several days using a structured code.

RAMPANT is the first of a new generation of CFD software which uses an unstructured grid to provide truly automatic mesh generation. Once the surface is defined, the grid is generated without user intervention. Unstructured grids also offer the possibility of adapting the grid distribution in space to optimally solve a given problem. Adaptive mesh refinement also allows important flow features to be resolved locally while retaining a more cost effective coarse mesh in regions of smooth flow. For some production runs, CPU time savings on the order of 10 to 1 can be achieved.

For this inlet problem, the first step was to import an I-DEAS generated surface mesh into RAMPANT. There, engineers created an initial concept model with approximately 200,000 cells in about four hours. At the time, they were running a beta version of RAMPANT. They estimate that the same model might

be created in as little as 15 to 20 minutes with the current version of the software.

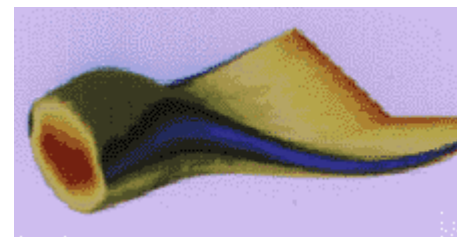
The initial analysis showed a region of air separation but the primary problem was that the pressure did not recover well at the outlet and was not uniform. They went back to I-DEAS and made some adjustments to the geometry in about a half hour. The next analysis showed a much more uniform distribution of pressure at the outlet. The solution time for these analyses was about 200 hours on a Cray supercomputer, but that too would be improved in the current version of RAMPANT. An unstructured grid can take longer to solve than a structured grid when the application involves internal flow, assuming that capturing the boundary layer is important. With most problems, however, the opposite is true--the unstructured grid requires fewer cells and a shorter solution time.

By using CFD, the engineers saw that the original design could be improved. Without the analysis, this would have been determined in the tunnel after a week of testing on the original design. CFD saved five days of wind tunnel testing or at least \$100,000 on this problem alone.

Besides being one of the few CFD codes that can handle complex geometries in a timely manner, RAMPANT is also the best Sandia has found for aerodynamics analyses. Another feature they appreciate is its ease of use. An engineer who has not used the package for six months can come back and pick it up again in five minutes. Since Sandia was a beta site for the software, they were using it before the user's manual was available. They just loaded the software and were able to use it almost immediately.



Hybrid inlet particle paths



Hybrid total pressure surface contours.