

## enGrid: Open-source mesh generation and GUI

Dr. Oliver Gloth, [ogloth@engits.com](mailto:ogloth@engits.com)

enGits GmbH  
Postfach 32, 79670 Todtnau, Germany

### Abstract

Starting in 2008 an open-source mesh generation software, called enGrid [1], has been developed at enGits GmbH. We believe it has now reached a state where it can be used for day to day CFD tasks. Together with the open-source modelling software Blender [2], enGrid offers a fully open-source work-flow for pre-processing. It has the capability to export native OpenFOAM grids as well as complete cases with boundary conditions and solver parameters. Since enGrid uses VTK internally to visualise grids, it is generally possible to include post-processing into the GUI as well. Should the demand arise, enGrid could thus serve as a complete GUI for OpenFOAM.

Internally enGrid uses NETGEN [3] to create tetrahedral grids. Prismatic boundary layer creation as well as surface meshing has been developed in-house at enGits GmbH. Currently enGrid can import a number of different surface file formats. Prerequisite for successful mesh generation with enGrid is an almost water-tight surface model (e.g. as STL file). Minor defects can be corrected, but enGrid is less tolerant to imperfect STL files than snappyHexMesh. Ideas on combining snappyHexMesh and enGrid, also for hex far field grids, exists. Implementation depends on customer demand, but we assume that a first step in this direction will be made during 2010. Particular effort has been put into the data exchange with Blender and a few Blender import and export scripts have been created for this purpose. It is, for example, possible to geometrically define boundary conditions within Blender or enGrid.

Grids can be exported as standard hybrid grids (tetras and prisms) or polyhedral grids. If the polyhedral export function is used, enGrid will keep boundary layers as they are and only convert (dualise) the tetrahedral far-field. Figure 1 shows a recent example of such a polyhedral grid together with the corresponding simulation result. For this example the full simulation process has been done using open-source tools (Blender -> enGrid -> OpenFOAM -> ParaView). Currently the polyhedral export is restricted to fairly small grids. This problem will be addressed soon and we hope to be able to present a robust and efficient poly-mesh algorithm at the workshop.

Solver parameters and physical boundary conditions can be defined within enGrid. A template mechanism exists to enable the creation of solver user interfaces within the enGrid GUI. Currently interfaces for simpleFoam and rhoSimpleFoam exist but the extension to other solvers should be fairly straightforward.

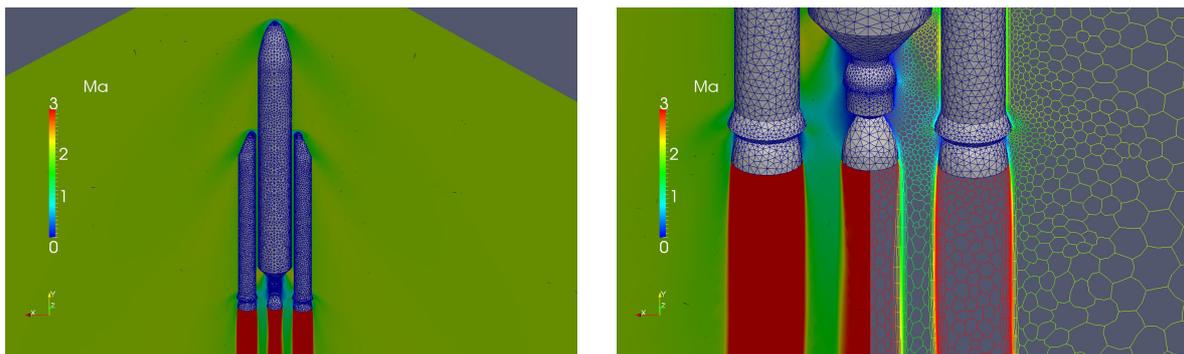


Figure 1: Ariane 5 at maximum dynamic pressure ( $Ma=2.05$ )

**Key words:** enGrid, mesh generation, GUI, pre-processing

### References

- [1] enGrid: Open Source Mesh Generation, <http://engits.com/engrid>
- [2] Blender, <http://www.blender.org>
- [3] NETGEN/NGSolve, <http://www.mathcces.rwth-aachen.de/netgen>