INTERNATIONAL DESIGN CONFERENCE - DESIGN 2008
Dubrovnik - Croatia, May 19 - 22, 2008.

DESIGN 2008

RECENT ADVANCES IN AVL'S CFD MESH GENERATION SOFTWARE – ESE-TOOLS

F. Juretić, W. Moser and G. Rainer

Keywords: CFD, mesh generation, simulation-oriented tools

1. Introduction

The design cycle in the automotive industry becomes shorter every day, and the norms products must fulfil become more and more strict. This motivates engineers to use CFD simulations to get information about the performance of their product early in the development cycle, in order to eliminate any possible deficiencies as soon as possible.

On the other hand, CFD simulations are difficult to perform at the pace required in the industrial environment due to current limitations of the commercially available pre-processing tools [White 2000]. The main deficiency is that these tools are difficult to use for engineers with no CFD background, and require a substantial amount of manual work to correct errors in geometrically-complex regions, such that the user can start the simulation and expect useful results as soon as possible. In addition, detailed CFD simulations require big models with long running times and large computational resources, which is not always feasible during the pre-design stage.

The problems mentioned above have motivated AVL to start developing application-specific tools, which aim at solving problems in the automotive industry, and therefore allow engineers to gather valuable information in a short time even on a home PC. Additionally, they are designed for user-friendliness, and are easy to learn, even for engineers with no CFD experience. This paper presents three such tools, integrated in AVL FIRE's Engine Simulation Environment (ESE), which are:

1. ESE Diesel – Diesel engines (segment models).
2. ESE Aftertreatment – Exhaust line and exhaust gas aftertreatment systems.
3. ESE 3DEngine – Internal combustion engines.

Section 2 presents the main features of ESE Diesel. The features available in ESE Aftertreatment and their impact on the productivity are given in section 3. The ideas behind the ESE 3DEngine and the current status of the project are described in section 4. The summary and conclusions for out future work is given in section 5.

2. ESE Diesel – Diesel engines (segment models)

ESE Diesel is a simulation environment for Diesel engines with enhanced pre and post-processing, and is developed under the assumption that Diesel-spray and Diesel-combustion happen under axial-symmetric conditions during the so called 'high-pressure-cycle' (from intake valve closing to exhaust valve opening). Therefore, only a slice of the cylinder, generated from a 2D sketch, can be used for modelling the phenomena. This assumption reduces complexity of the meshing requirements without any significant influence on the reliability of the results. However, this assumption reduces meshing complexity from a 3D problem down to the 2D for which the reliable meshing techniques are already available.

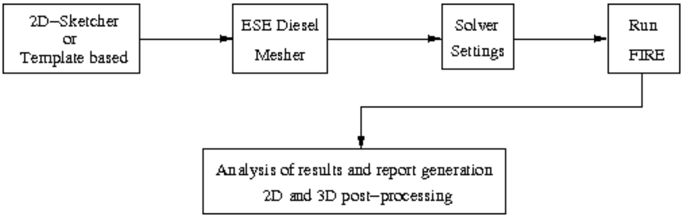

Figure 1. ESE Diesel workflow

The procedure is designed to lead the user through the workflow consisting of a few steps, Figure 1. At the initial stage, the user needs to provide a 2D sketch which contains the cylinder bowl, squish volume and the injector. In order to simplify the work at this initial stage, many pre-defined geometries are on disposal to the user. The geometry can be loaded into a newly developed tool, so-called Sketcher, which allow the user adjusting the template to the actual geometry under consideration. The Sketcher is based on the DCM library [UGS PLM Software 2007] that is used for constraint handling and parametrized sketching, and also allows for generation of the geometry from scratch. The piston movement is also specified at this stage.

Once the geometry sketch is finished, the procedure proceeds to the meshing step, which is performed by using paving [Blacker 1991] and transfinite interpolation [Thompson 1999] as the backbone algorithms. An example is presented in Figure 2.

After the mesh is finished, it is still remains to set up boundary conditions and solver parameters in the same fashion as in the other established AVL CFD tools.

ESE Diesel contains a set of application specific post-processing features (2D graphs of engine specific data, comparison with measurement data, automated report generation …) which increase its attractiveness for design engineers.

It is worth pointing out that ESE Diesel can readily be run in batch mode (off-screen mode), since all user-relevant functions (project setup, meshing, solver start …) are python-wrapped and are therefore accessible for external optimization tools, either commercial ones or in-house tools of AVL's customers.

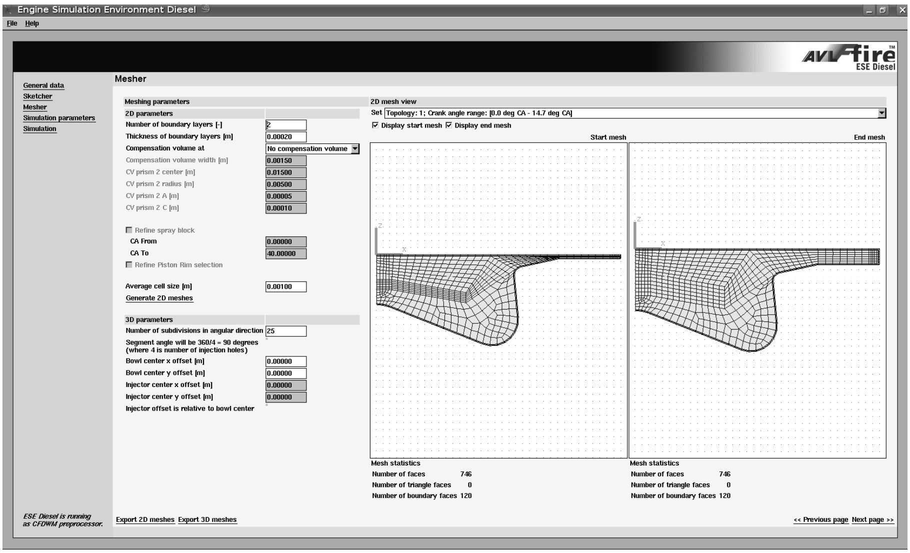

Figure 2. Meshes generated by ESE Diesel

3. ESE Aftertreatment – Exhaust line and exhaust gas after-treatment systems

ESE Aftertreatment is a mesh generation and result analysis environment for after-treatment devices. The after-treatment devices mainly consist of pipes which can be bent or straight, silencers, mufflers, catalysts and Diesel particle filters (DPF). This approach is based on the rationale that most of such components are similar and therefore can be meshed by using the pre-defined mesh topologies.

The chemical processes happening in catalysts are very complex and challenging to model, making the quality requirements for a computational grid are very strict. They are also demanding from the computational point of view because the length scales of the solution features are not uniform, and therefore require flow-aligned, high aspect-ratio cells to reduce the discretisation error in the solution.

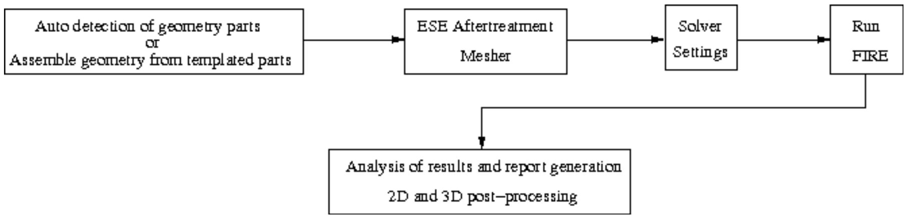

Figure 3. ESE Aftertreatment workflow

The aim of the project is to provide engineers with the set of pre-defined topologies and the advanced algorithms which can generate high-quality meshes in complex geometries where the pre-defined topologies do not provide satisfactory results.

ESE Aftertreatment leads the user through a pre-defined workflow shown in Figure 3. The mesh generation procedure is the initial stage of the process and it consists of two steps. At the first step, the user defines the geometry of his/her system from a set of pre-defined geometries, see Figure 4., or from its own (triangulated) CAD data. In the latter case, various components will be auto-recognized. If the auto-recognition does not correctly identify all parts, the user can interactively interfere and provide the missing information to the software.

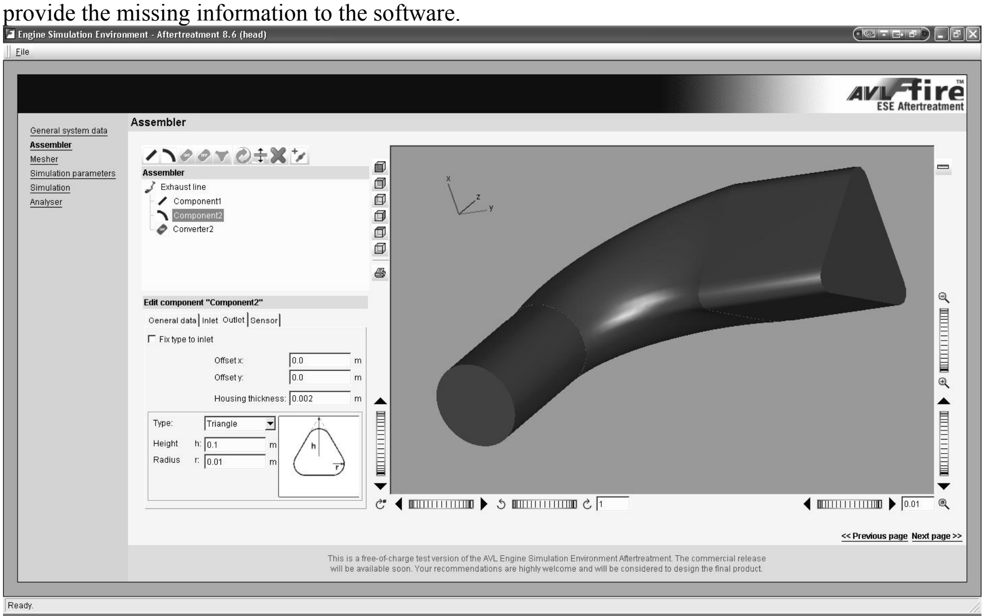

Figure 4. Geometry generation in ESE Aftertreatment

The second step is the generation of the mesh for the defined geometry. This stage is fully automatic, and is performed by using paving [Blacker 1991], sweeping [White 2000] and revolving [Thompson 1999] as the main meshing algorithms. The same techniques are used to generate well-suited start topologies (i.e.: good initial approximations of the original shapes) for complex parts (waste-gate flap, manifolds, pipes with sensors …), which are subsequently ‘sculptured’ to match the real geometry of the component under consideration. An example of the mesh generated is shown in Figure 5.

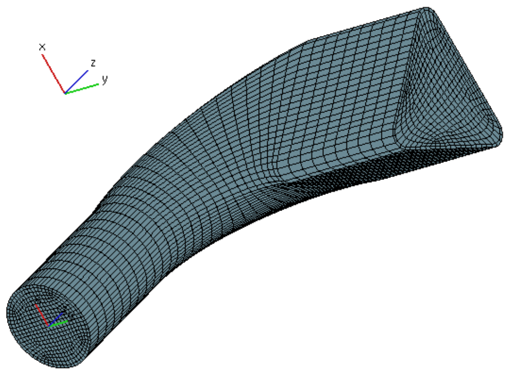

Figure 5. A mesh generated with ESE Aftertreatment

After the mesh is finished, it can be loaded into other AVL CFD Tools which are used for running the model and extracting the information of interest from the simulation.
Moreover, ESE Aftertreatment, similar to ESE Diesel, is also equipped with application specific post-processing features intended to simplify and speed up the design process of after-treatment systems.

4. ESE 3DEngine – Internal combustion engines

ESE 3DEngine is an ongoing software development project in AVL Advanced Simulation Technologies. It is intended to be used in cases for which the assumptions made for ESE Diesel are too restrictive. The aim of the ESE 3DEngine is to provide engineers in the automotive industry with an automatic tool for generation of high-quality flow-aligned 3D engine meshes which are needed for detailed CFD calculations. In order to make the tool robust, efficient and to ensure short time-to-market, the development strategy is to combine the best meshing approaches currently available and develop new ones where necessary.
A particular challenge in the field of CFD simulations for internal combustion engines is the fact that moving parts, ie. valves and the piston, are involved in the simulation. Strictly speaking, for each calculation time step, a new mesh has to be provided. This can either be achieved by deforming an already existing mesh, or by completely re-meshing the altered geometry. The first approach is preferred, since the loss of accuracy during the mapping from one mesh to the next is less in the case of deformed meshes with the same cell topology than in the case when the completely different mesh topologies are used. The degree to which a computational mesh can be deformed without suffering an unacceptable loss of cell quality depends on its alignment in space and the cell structure in areas which undergo the major share of deformation (cells near valves or near the piston top). The change between the various mesh topologies, where both meshes cover the same geometrical domain, is commonly called a ‘rezone’.
Please note that the combustion models required for simulating internal-combustion engines are very complex and their accuracy is closely related to the quality of the computational grid, especially in regions of high velocities, turbulence … , like in valve gap or around injectors or spark plugs. An example of the valve mesh generated with this tool is shown in Figure 6, and its quality is better than our traditional approach shown in Figure 7. The new tool has some advantages over the traditional

one. For example, the number of cells is reduced by almost half when compared to the traditional approach resulting in shorter calculation times. Additionally, the new approach produces flow-aligned meshes which result in better convergence and lower discretisation errors than the traditional approach.

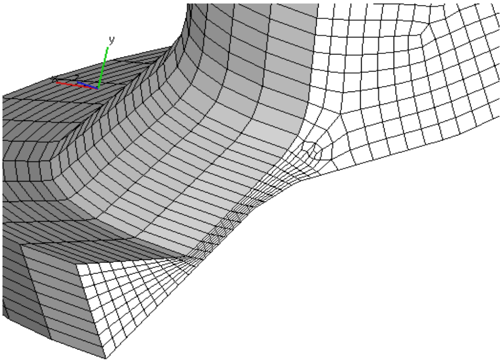

Figure 6. Valve mesh generated with ESE-Engine 3D

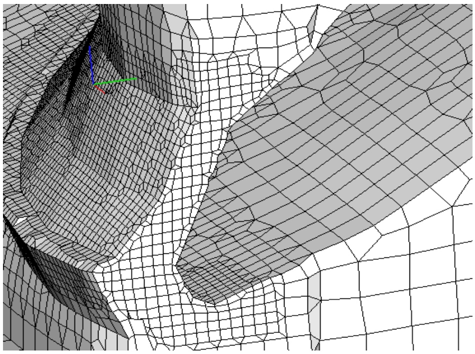

Figure 7. Valve mesh generated with FAME

From the aforementioned requirements, the development objectives of the ESE 3DEngine project are straightforward to derive. The aim is to develop a versatile, engine-specific pre-processing tool producing meshes of overall good quality, which meets extreme quality requirements in determined parts of the problem domain (e.g. in the valve gap). These meshes shall be constructed such that the number of rezones necessary to simulate the movement of the piston and the valves is minimized.
Another major focus will be on the methods for straightforward generation and manipulation of geometry variants, i.e. exchange of geometry parts like piston bowls, valves, intake / exhaust ports, spark plugs, injectors, pre-chambers, etc. without re-meshing the whole geometry. This will be enabled through a modular workflow which will enable the user to combine pre-defined mesh templates or choose different meshing algorithms for various parts of the engine under consideration.
ESE 3DEngine will also be equipped with tools for the analysis of computational results and the automated generation of reports.

5. Summary and Conclusions

This paper presents three design-oriented tools developed – or currently being developed - by AVL-AST in order to increase the usage of CFD at the early stages of the design process. The tools are designed for user-friendliness and are easy-to-use even for engineers with little CFD background.

References

Blacker, T. D., Stephenson, M. B., "Paving: A new approach to Automated Quadrilateral Mesh Generation", International Journal for Numerical Methods in Engineering, Vol.32, 1991, pp 811-847.

UGS PLM Software, "D-Cubed 2D DCM Manual Version 52.0", D-Cubed Ltd, UK, 2007.

Knupp, P. M., "Applications of smoothing: copy, morph and sweep on unstructured quadrilateral meshes", Vol.45, 1999, pp 37-45.

Thompson, J. F., Soni, B. K., Weatherill, N. P., "Handbook of Grid Generation", CRC Press, UK, 1999.

White, D. R., Tautges, T. J., "Automatic scheme selection for toolkit hex meshing", International Journal for Numerical Methods in Engineering, Vol.49, 2000, pp 127-144.

dr. Franjo Juretić
Software Development – Fluid Dynamics
AVL-AST d. o. o.
Avenija Dubrovnik 10, 10000 Zagreb, Croatia
Tel.: +38516598653
Fax.: +38516598666
Email: franjo.juretic@avl.com