

## 3D EXHAUST SYSTEM (MUFFLER) DESIGN TOOL FOR 1D CFD SIMULATION PURPOSES

D. Tonković, N. Putz and F. Juretić

*Keywords: muffler, exhaust system, CAD system, mesh generation*

### 1. Introduction

These days the environmental topics are becoming extremely relevant for the automotive industry. Most big cities have problems with pollution, mostly caused by engine exhaust gas. The most dangerous is the air pollution, which is more-and-more regulated by laws and regulations. However, engine noise is also a serious problem that shall not be neglected. In addition, engine's sound plays an important role in car's identity, in some cases even as important as the car's visual design.

The internal combustion engines exhaust system has three major functions. The first one is to channel out the waste products out of the engine. The second function of the exhaust system is to clean up emissions that are harmful to the environment. This is achieved with elements like catalytic converters and diesel particulate filters. The third function is to reduce noise generated by the engine. Muffler, the exhaust system element that is used to reduce and modify the noise, is the main target for the tool presented in this paper.

In the world of computational fluid dynamics (CFD) simulations, 3D simulations are typically very complex, thereby giving the engineers detailed information about the fluid flow, heat transfer and pressure fields in geometrically complex systems. However, 1D CFD simulations are much faster, and still give a good overview of the system under investigation. By being fast, they are also more suitable to run in optimisation loops, which is a desirable feature in the area of exhaust system design. Since mufflers are becoming more and more complex, and no detailed information about the muffler geometry is known and/or required in the early design stage, coupling 1D and 3D CFD is shown to achieve the optimal results in some applications [Montenegro 2011]. However, 1D method still requires a set of 3D cells, which shall be generated automatically. Aftosmis [Aftosmis 1997] has developed a method for cutting Cartesian cells with the geometry of the model in order to perform the mesh generation automatically. Recently, the cut cell approach has been extended for mesh generation of internal combustion engines [Senecal 2007]. Juretić [Juretić 2005] has implemented a method for generation of meshes consisting of arbitrary polyhedral by cutting off the parts of the polyhedral mesh template outside of the geometry surface.

This paper presents BOOST 3D Cells, a product capable of both designing mufflers, and transferring their 3D geometry information to a network of 1D cells. It is integrated in AVL BOOST, an environment that can provide necessary inputs to the system, run the simulations, analyse the data and use automated optimisation loops. The tool is designed to be user friendly and interactive, as the targeted engineers are not used to working in complex 3D systems. BOOST 3D Cells was developed by using Spatial's 3D ACIS modeller [Coreney 2002].

The workflow transferring the 3D geometry to a 1D network of cells is achieved in three steps:

1. Computer aided design (CAD) stage
2. Mesh generation
3. Polyhedral cell mesh visualization / post processing

The muffler CAD design stage is elaborated in section 2. As an introduction, a short description of muffler design principles is included, as this is a basis for the implemented CAD functionalities. Section 3 presents the meshing approach. Section 4 elaborates polyhedral cell visualization and transfer to 1D mesh elements. Conclusions and summary are given in section 5.

## 2. 3D CAD sketch tool optimized for muffler design

The main motivation for using CFD simulation tools in the area of automotive power train design is to test the geometry of specific powertrain elements. Good and accurate geometry information is thereby essential in order to get satisfactory simulation results.

When creating this simulation tool, two options were considered. The first option assumed that the user can generate geometry in an arbitrary CAD system, export this geometry as a file and use it for simulation. However, this approach suffers of multiple problems. Pre-processing is often the most complex and the most time consuming stage in a simulation workflow, and the user input is often essential in order to create good result. Therefore, the geometry shall contain certain amount of user data, like element classification and parameters. In general, CAD systems are not optimized to handle this kind of specialized data input, especially from usability point of view. The user would not only have to learn an often very complex general purpose CAD system, but also have to learn how to attach data useful for meshing to CAD entities. Additionally, CAD input shall satisfy additional requirements, like quality, connection between objects, water tightness, etc.

After analysis, the second option was proven to be much more appealing. Thus, if the simulation tool has its own specialized CAD system, it can both guide the user into giving correct information for the meshing and simulation process, and ensure that the geometry is of high quality. Also, the learning phase is much shorter and simpler than in the case of a general CAD system. In addition, the motivation behind the concept of integrating the CAD modeller into the simulation tool is to speed up the design - simulation cycle. By integrating them both to a single software environment, the user does not need to use various software environments in order to optimize the geometry, which is often associated with data loss and degradation of its quality.

The application workflow consists out of three steps. The initial CAD stage results in the geometry definition according to user's requirements which is used to define a background template mesh. The second stage is the meshing process resulting in a polyhedral mesh. The final stage exports the mesh which is then used in AVL BOOST for acoustic simulation.

### 2.1 Muffler geometry

Muffler geometries are quite complex, compared to the rest of the internal combustion engine system. However, by implementing the knowledge of experienced engineers, the geometry of the model can be optimized in a way not to include unnecessary features and still keep the ones that influence the results.



**Figure 1. Reactive and dissipative muffler**

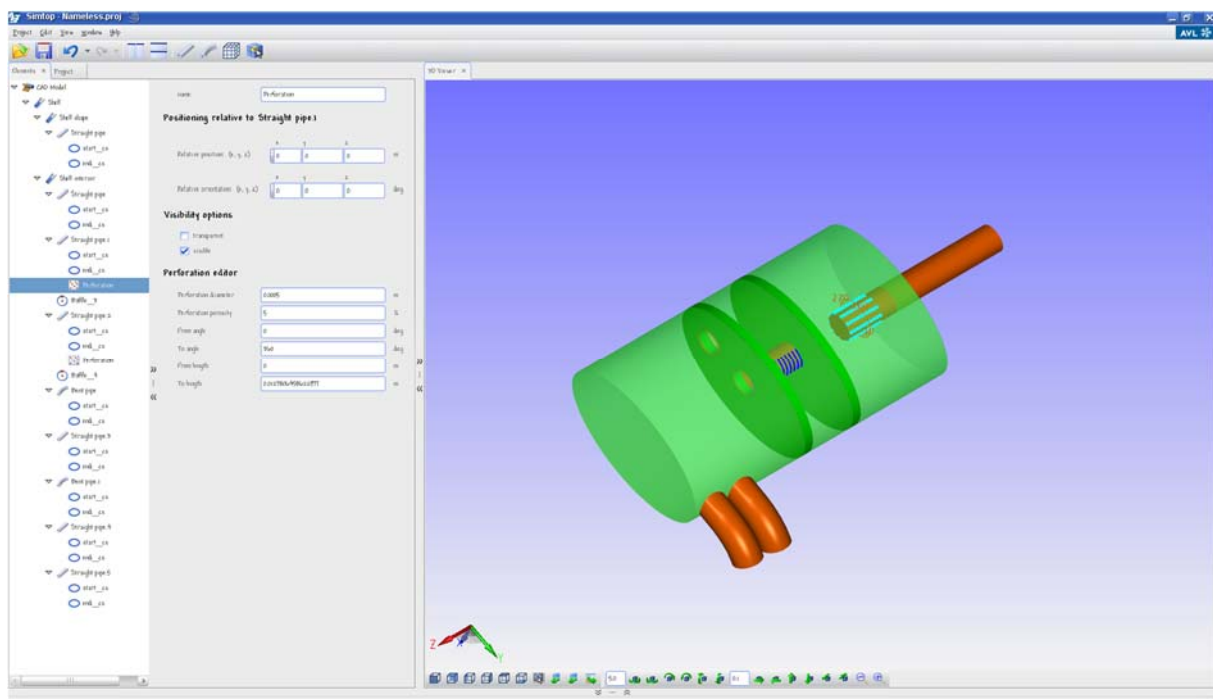
There are two commonly used types of mufflers, namely a reactive (also called reflective) muffler or dissipative (also called absorptive) muffler. Reactive mufflers are usually implemented as cavities

attached to a pipe. The pipes are often perforated in order to allow the sound pulses to scatter out in numerous directions. Cavity geometry is designed for specific frequency, so in order to target a frequency range, it is important to correctly distribute the cavities. Dissipative mufflers use sound absorptive material, like steel or glass wool in order to take the energy out of a wave. Its usual implementation involves a perforated pipe wrapped with a layer of sound absorptive material and housing. The reactive muffler is shown in the left part of Figure 1 and the dissipative one in the right part.

In order not to waste space and material, the elements of a muffler are grouped together and placed inside a single shell. Chambers are divided by barriers typically referred to as baffles which can also have different holes and perforations. Furthermore, chambers can be filled with absorptive material, thereby creating a dissipative muffler.

## 2.2 Muffler CAD tool

As shown in previous section, muffler's design is based on standard components like straight and bent pipes, baffles and perforations. Since the modeller is also required to be simple to learn, a template based combination of direct and parametric modelling approach was used. An example of the application window layout can be seen in Figure 2.

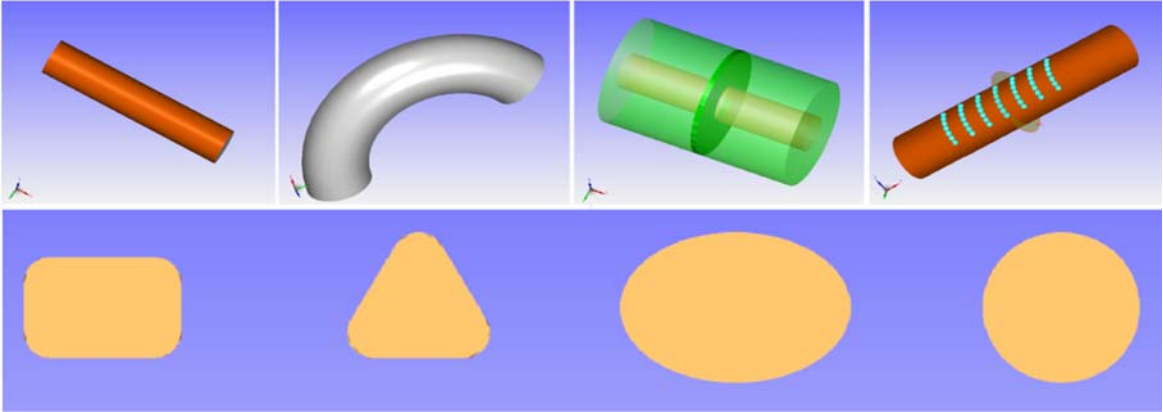


**Figure 2. Application window layout**

A muffler consists of different elements in simple relations to one another, thus the template based model has been considered the best choice. In addition, template components like straight or bent pipes make the parameterisation quite simple and straightforward, therefore also making the parametric model simple. This makes both constraints and the possibilities of the modeller immediately clear for the user. Templates used in the muffler designer are pipes, bent pipes, baffles, perforations and cross sections. Different templates can be seen in Figure 3.

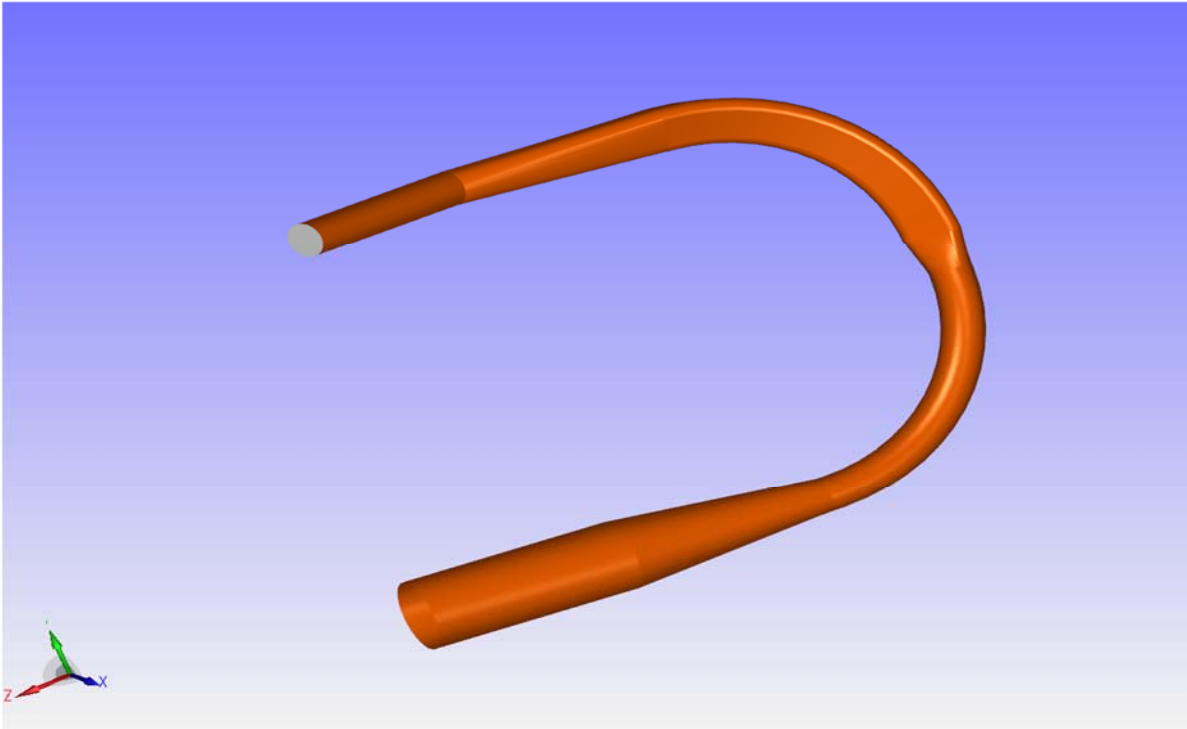
Figure 2 shows the two essential Graphical User Interface (GUI) elements. The first is the element tree with the parameter editor and the interactive viewer window. It is important to emphasize that both viewer and element tree are completely independent but synchronized. When an element is modified in the element tree the modification is visible in the viewer and vice versa. Furthermore, when an element is selected in the element tree it is selected in the viewer as well. By having these possibilities, the user can combine both GUI elements to achieve the desired design. The tool supports a direct modelling approach to make it easy to use. In addition, it allows easy creation and quick modification

of geometries on the fly, by using viewer interactions. However, there is a significant weakness of this approach for such applications, because it makes running a CAD modeller in an optimisation loop impossible since automatic optimisation of the geometry to fit the desired simulation result is not achievable without user interaction. In order to define and vary parameters a parametric model is implemented, regardless if the parameters will be optimised automatic by some optimisation algorithms, or manually by the user. Since design variations are an important use case, a hybrid approach was chosen. This hybrid approach includes a parametric model as a base, a direct modelling approach and a set of hard-coded rules mandatory for muffler geometries.



**Figure 3. Templates used for modelling**

The viewer interaction system is an essential part the application and is mostly based on moving the objects. With this functionality, it is possible to influence the pipe’s length by moving the starting or end cross section, change its opening angle by rotating the cross section, or change the pipe’s cross section diameter. The application is designed to allow the user to create an approximate sketch of a complete muffler using only viewer interactions, and then to modify the parametric information in order to get the exact and precise model.

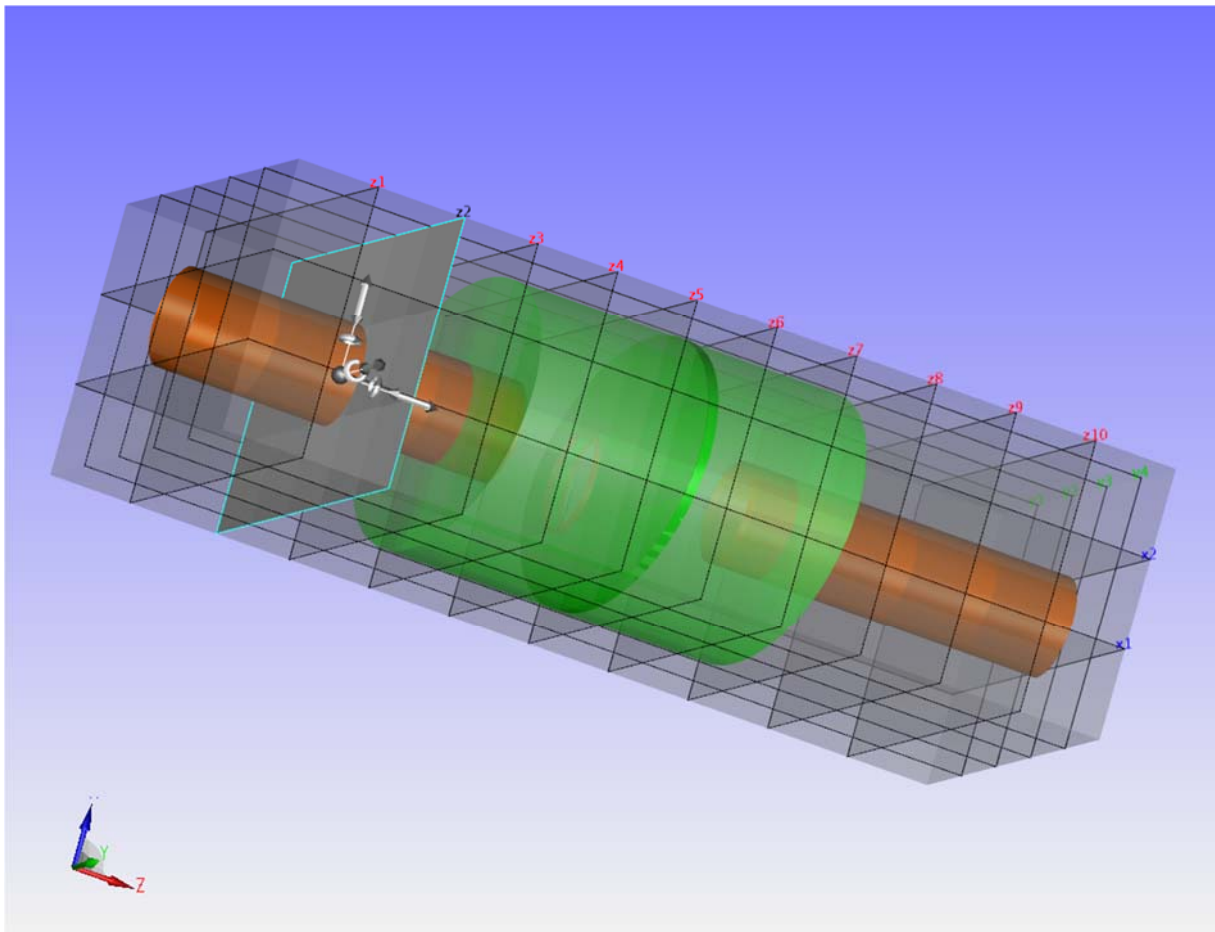


**Figure 4. A piping system**

Since mufflers consist mainly of pipes, regardless if the pipes are inside the shell or they connect multiple shells, the modeller includes piping CAD system which takes the above geometry constraints into consideration. This is achieved by simple pipe concatenation. The relationship between pipes is set by simply selecting a pipe's start or end and adding another pipe to that cross section, either in a context menu, or by using a toolbar. By setting this relationship the pipes are permanently connected, so the model can again be roughly sketched using viewer interactions, and modified using the parametric model. The model shown in Figure 4 was created by using less than ten mouse clicks and moves.

### 3. Mesh generation

Acoustic simulations usually do not require a fine mesh. For that purpose an orthogonal grid describing a meshing template is used (Figure 5). A smaller number of cells in the mesh provide an opportunity for user interventions, which are not possible in case of complex 3D CFD simulations, often consisting of millions of cells. As seen in Figure 5, the discretization can be interactively modified, by adapting the background mesh to the actual geometry, based on the user's experience. The focus of the meshing approach is to achieve robustness, while being straightforward for the user.



**Figure 5. Template mesh**

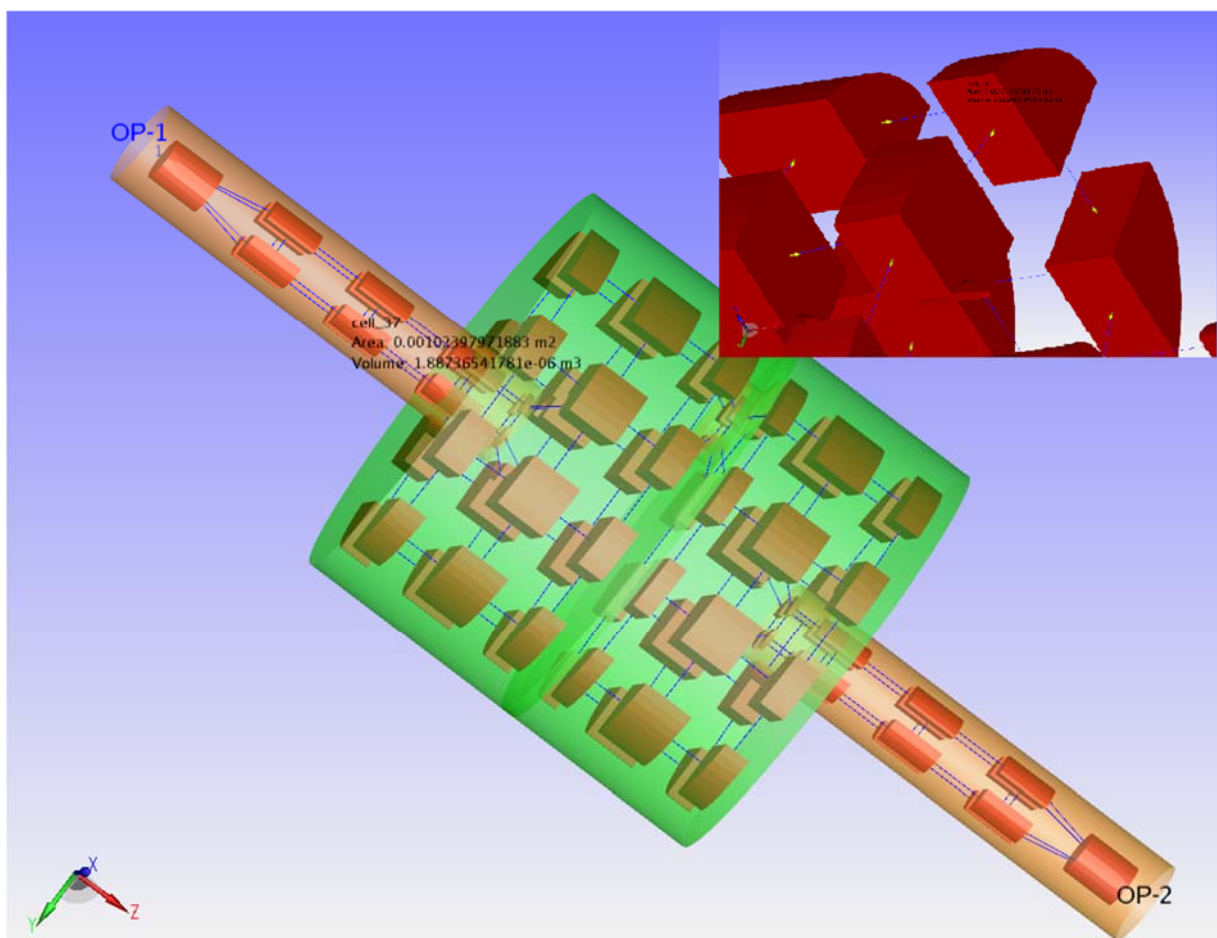
Polyhedral meshes are used at all stages in the meshing process. The motivation for using polyhedral meshing approach is primary the small number of polyhedral cells that can describe the geometry. Polyhedral cells are also capable of describing virtually any kind of geometry, so they guarantee that the meshing will always be possible and that it will always provide a mesh. In addition, we have used the polyhedral meshes, because of their ability to exactly describe the geometry, with a minimum error introduced.

The meshing procedure had to satisfy the same criterion described above, so the chosen meshing procedure was the cut-cell approach [Aftosmis 1997]. Though not simple to implement, the cut-cell approach satisfied the requirement to preserve specific geometric properties needed for the simulation, like areas, volumes and lengths in specific directions. In addition, this approach simplified the extraction of the above mentioned properties.

#### 4. Visualization and post-processing of generated meshes

The visualization of coupled 1D and 3D system includes visualization of both 1D elements and 3D elements in 3D. In addition, 1D elements are represented as icons, and 3D elements as polyhedral cells, as they have to give the user information about their shape.

The visualization of 3D meshes has always been a challenging task, since usually only the elements on the surface are visible, hiding the structure behind. Therefore, the presented application uses shrinking for visualisation of 3D polyhedra. A scaling transformation matrix is applied to each polyhedral cell. An example is shown in Figure 6.



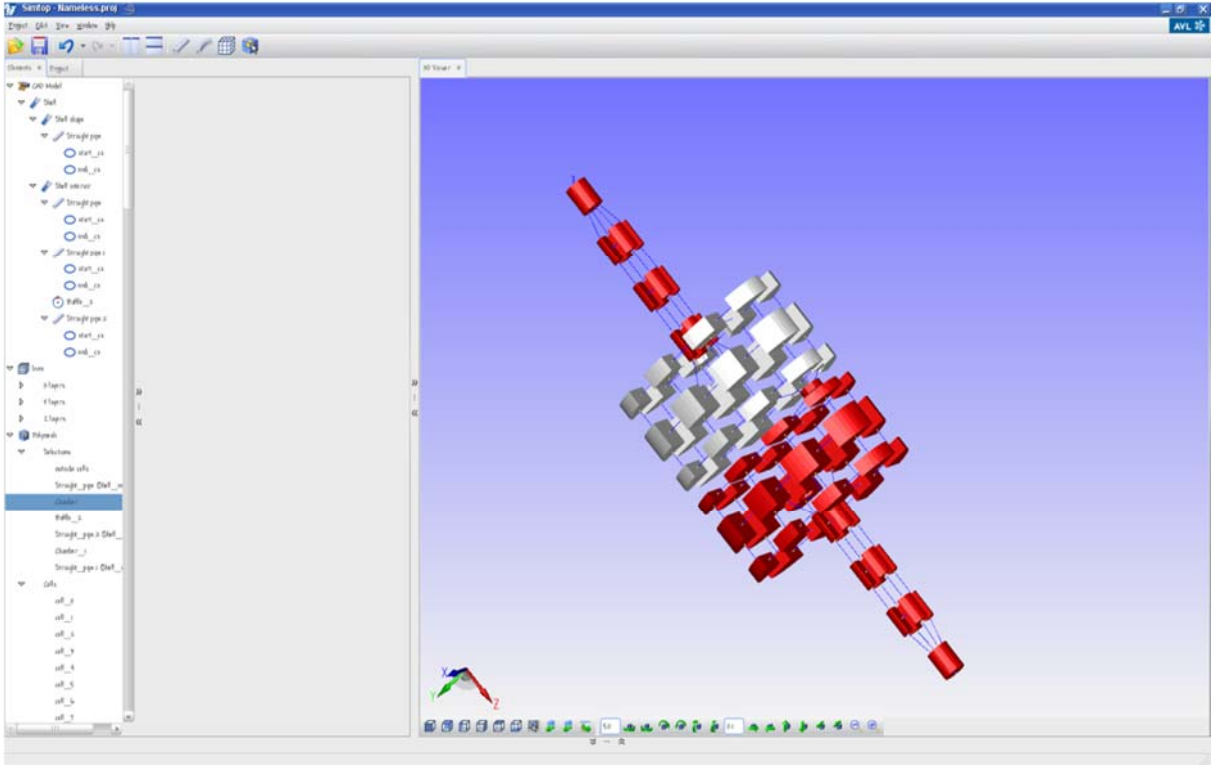
**Figure 6. Polyhedral mesh**

The goal of the meshing procedure described in this paper is to generate the network of 1D cells describing the 3D polyhedra cell system. The most important information needed to visually inspect the mesh quality is the volume and the shape of the polyhedra.

In addition, there are two more informations needed for the simulation procedure to run. The first one is the connection between cells. In our visualization approach the connections are visualized by using simple connecting lines. The existence of the line between two cells means that there is no physical obstacle between them, so the flow is possible between these two cells. Absence of the connection line indicates that there is a solid barrier between these cells, and a dashed line indicates a perforated connection.

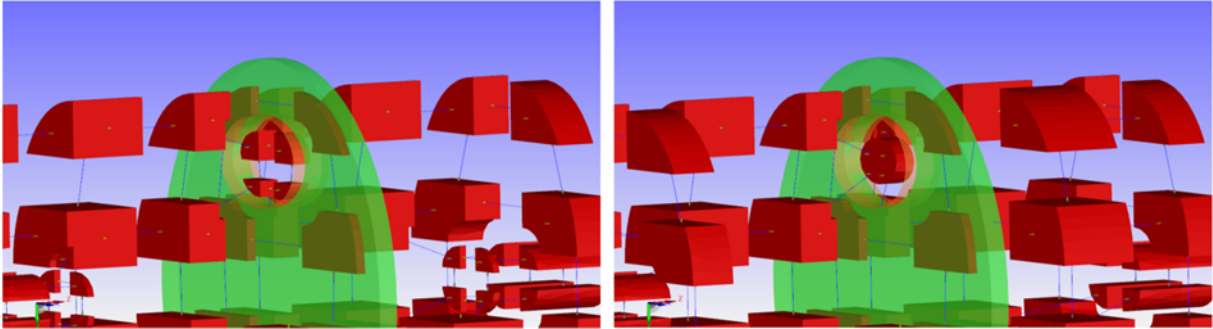
The second information visualized in this system is port information. Ports are positions on 3D cells where the connection lines are connected to. Information attached to ports is the area, the direction of the flow and the length of the cell in that direction. They are visualized as arrows positioned on the surface of the cell, pointing in the actual flow direction.

By using a region growing algorithm [Agathos 2007], we can make the segmentation of the polyhedral mesh into smaller regions. While performing the region growing algorithm, and while maintaining a rule not to cross the surface of the muffler, a mapping is established. The map contains all the muffler surface elements touching that region. This information is then used to distinguish the outer from the inner cells, recognising the cells inside a pipe, or cells inside a baffle. A mesh split into regions is shown in Figure 7.



**Figure 7. Polyhedral mesh separated to regions**

The structure of the mesh allows the automated post processing steps, like deleting or merging cells. For instance, an important post-processing step is to attach cells with small volume to the neighbouring cells, or the small flow length cells to the cells in the same flow direction. Figure 8 shows the topology where the small volume cells are merged to their neighbours.



**Figure 8. Cell merging**

## 5. Summary and conclusions

This paper presents a new and innovative method for mesh generation designed to support 1D simulation process. The tool is designed to be used in early stages of the design process, by engineers with little experience in CAD and 3D CFD. The presented CAD approach focuses on interactivity and usability, by using a parametric model for geometry information and for simulation parameters. A general meshing approach using polyhedral cells was presented. The approach is robust and applicable to complex geometries. In addition, a new method of polyhedral mesh visualization is introduced. This approach provides an easy and a fast way of mesh inspection, also presents the flow connectivity information to the user.

The presented method is based on a workflow concept with well-defined steps and interfaces between them. The CAD system can be extended with a 2D sketching tool which would give the freedom to the user to sketch objects of complex shape and embed them in the muffler. The method can be further improved by allowing more complex operations in the process of template mesh generation which would improve the quality of generated polyhedral meshes.

## References

- Alexander Agathos, IoannisPratikakis, Stavros Perantonis, NikolaosSapidis and Philip Azariadis, "3D Mesh Segmentation Methodologies for CAD applications", *Computer-Aided Design & Applications*, Vol. 4, No. 6, 2007, pp 827-841
- F. Juretic, "Error Analysis in Finite Volume CFD", PhD Thesis, Imperial College, London, 2005.
- G. Montenegro, A. Della Torre, A. Onorati, R. Fairbrother, A. Dolinar, "Development and Application of 3D Generic Cells to the Acoustic Modelling of Exhaust Systems", paper number 2011-01-1526, SAE 2011 Noise and Vibration Conference and Exhibition, Grand Rapids, Michigan.
- G. Montenegro, A. Onorati, A. Della Torre, A. J. Torregrosa, "The 3Dcell Approach for the Acoustic Modelling of After-Treatment Devices", *SAE Int. J. Engines* August 2011 4:2519-2530
- G. Montenegro, R. Faribrother, A. Onorati, A. Della Torre, "A General 3D Cell Method for the Acoustic Modelling of Perforates with Sound Absorbing Material for I.C. Engine Exhaust Systems", *ICSV 17, 17th International Congress on Sound & Vibration*, 2010, Cairo, Egypt.
- J. Corney, T. Lim, "3D Modeling with ACIS", 2<sup>nd</sup> edition, Saxe-Coburg Publications, Kipen Stirling, UK, 2002.
- M. J. Aftosmis, M. J. Berger, J. E. Melton, "Robust and Efficient Cartesian Mesh Generation for Component-Based Geometry", *35th ALAA Aerospace Sciences Meeting*, Reno NV Jan, 1997.
- P. K. Senecal, K. J. Richards, P. Pomraning, T. Yang, M.Z. Dai, R.M. McDavid, M.A. Patterson, S. Hou and T. Shethaji, "A new parallel cut-cell cartesian CFD code for rapid grid generation applied to in-cylinder Diesel engine simulations, In proceedings, SAE International, SAE paper 2007-01-0159, 2007.

Danijel Tonković, dipl. ing.  
Software Development – Fluid Dynamics  
AVL-AST d.o.o.  
Avenija Dubrovnik 10, 10000 Zagreb, Croatia  
Telephone: +38517775037  
Telefax: +38517775123  
Email: danijel.tonkovic@avl.com