Development of Advanced Numerical techniques to Reduce Grid Dependency in Industrial CFD Applications

Hans Peter Blahowsky

AVL LIST GmbH
Advanced Simulation Technologies
Hans List Platz 1
A - 8020 Graz, Austria
Phone: +43(316)787-1222, Fax: +43(316)787-1462

ABSTRACT

Automatic mesh generation procedures applied to industrial flow problems lead to complex mesh topologies where usually no special considerations to mesh resolution are taken. In the present study a fast and flexible solution algorithm in combination with generalized higher order discretization schemes is presented and its application to intake port calculation is demonstrated.

Keywords: Advanced Numerical Methods, Intake Ports, Higher Order Schemes

Introduction

The continuous optimization of the product development process within the automotive industry requires a re-designed of the applied numerical simulation methodologies.

The total project turnaround time, based on the time needed to produce a computational mesh, to define the right boundary conditions and to obtain a converged solution has to be optimized in all aspect without compromising result accuracy or predictive capability. For a wide range of problems the generation of the computational model is the main bottleneck. Therefore automatic mesh generation tools have been developed with different concepts being now available on the commercial software market.

Additionally automatically generated meshes show in general higher numbers of mesh cells and complex mesh topologies. This requires that the numerical solution procedure has to be optimized with respect to flexibility and solution speed.

Meshing Methodology

The most commonly used automatic mesh generation methodologies are based upon either tetrahedral cell types or Cartesian grids which are not boundary fitted. Even though tetrahedral elements allow excellent meshing flexibility the draw back of reduced numerical accuracy and weakened solution stability is considerable. Cartesian auto-meshing concepts without proper treatment of the wall boundaries are as well of limited value.

In the presented investigation an automatic hexahedral meshing software, AVL-FAME, was applied. The methodology is based on hexahedral start topologies of arbitrary complexity and a high quality boundary layer modeling. With this the requirements of high quality cell shapes and accurate resolution of the boundary is fulfilled.

The meshing procedure can be summarized as follows:

1. Import of 3D surface descriptions
2. Closing of possible gaps in the surface data
3. Definition of a start topology which is in the simplest case Cartesian
4. Calculation of mesh-surface intersections
5. Interactive mesh refinement were necessary
6. Calculation of cell layers at wall
7. 3D smoothing of resulting mesh

In Figure 1 a simple example mesh is given. The start topology is in this case a Cartesian cube. The boundary fitted wall layers are clearly visible.

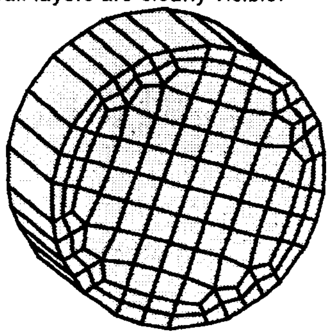

Figure 1: Example Mesh generated automatically by use of AVL-FAME

The application of this methodology allows a drastic reduction of meshing time for a wide range of flow analysis problems.

Numerical Scheme

AVL is introducing now new multi-purpose software CFD package, namely SWIFT. SWIFT is based on the finite volume technology where the governing transport equations are discretized on a generalized control volume, thus:

$$\underbrace{\frac{d}{dt}\int_V \rho B_\phi dV}_{\text{Rate of change}} + \underbrace{\sum_{j=1}^{n_f}\int_{S_j}\rho\phi(\mathbf{v}-\mathbf{v}_s)\cdot d\mathbf{s}}_{\text{Convection}} = \underbrace{\sum_{j=1}^{n_f}\int_{S_j}\Gamma_\phi grad\phi\cdot d\mathbf{s}}_{\text{Diffusion}} + \underbrace{\sum_{j=1}^{n_f}\int_{S_j}\mathbf{q}_{\phi S}\cdot d\mathbf{s} + \int_V q_{\phi V} dV.}_{\text{Sources}}$$

The main difference of the present numerical scheme to existing solution algorithms is that in many existing codes the data structure is limited to cell shapes with a maximum of six cell faces n_f while in SWIFT a new face connectivity has been introduced which allows cell shapes with arbitrary number of faces. As such, SWIFT can calculate any possible computational grid. For the sake of simplicity, the values of the faces are obtained from linear interpolation of cell centers. The gradients at the center of volume are calculated before interpolated on cell faces. For that purpose, Gauss theorem or similar practice can be used, thus:

$$\int_V grad\psi dV = \int_S \psi d\mathbf{s} \Rightarrow (grad\psi)_{P_0} \approx \frac{1}{V_{P_0}}\sum_{j=1}^{n_f}\psi_j \mathbf{s}_j$$

For the diffusive flux through an internal cell face, the following approximation is used:

$$D_j = \int_{S_j}\Gamma_\phi grad\phi\cdot d\mathbf{s} \approx \Gamma_\phi (grad\phi)_j^* \cdot \mathbf{s}_j$$

In this case *(grad ϕ_j)* is treated in a special manner which allows separation of this term in implicit and explicit part. Further details are found *in (Muzaferija and Gosman)*

AVL-SWIFT was used for calculations presented here. The most important aspect of present calculations is the successful application of higher-order differencing schemes which significantly decreases numerical error.

Differencing schemes for convective fluxes, implemented in SWIFT and used in the present case study are:

- first order upwind
- central differencing or
- MINMOD differencing scheme *(Sweby, 1984)* is a combination of a central and linear upwind differencing schemes while
- SMART *(Gaskel and Lau)* is a variant of a bounded QUICK differencing scheme.

Both, MINMOD and SMART, are implemented in flux-blended fashion with the extension to generalized shapes of the control volume.

The solution procedure of the obtained set of algebraic equations is of SIMPLE type.

The Examples

In the present study the computational models of two tangential intake ports, commonly found in high-speed diesel engines, have been selected for evaluating the mesh generation and numerical solution procedure.

In order to compare the methodologies the same geometry was meshed manually using the AVL-FIRE multi-block pre-processor and automatically applying the AVL-FAME auto-meshing tool. The calculation set-up in respect to boundary conditions and steady state flow solution represents the experimental configuration found on a typical flow bench. In Figure 2 *Mesh A* is shown which has been obtained using the multi-block meshing methodology whereas in Figure 3 the automatically generated mesh, *Mesh B* is outlined.

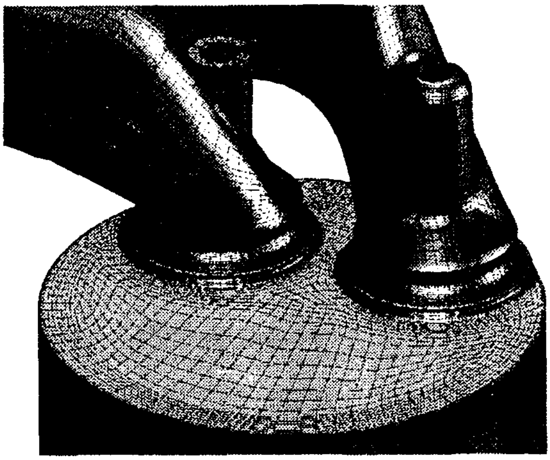

Figure 2: Computational Mesh A generated manually by use of AVL-FIRE preprocessor

Mesh A contains 220.000 mesh cells. Special care was taken in the areas of possible flow separation and at the cylinder wall during the meshing procedure. The meshing time was approximately 30 hours.

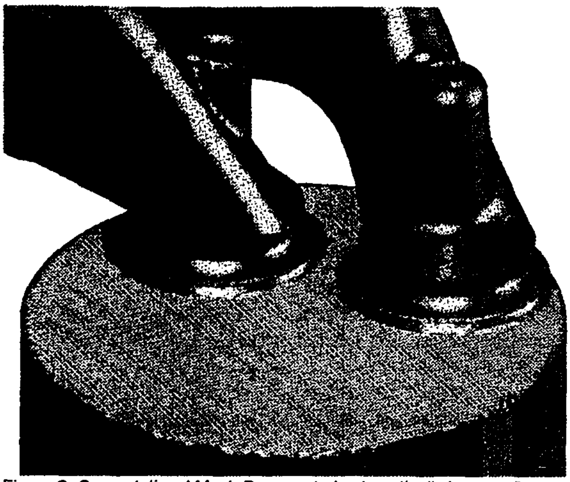

Figure 3: Computational Mesh B generated automatically by use of AVL-FAME

Mesh B contains 550.000 cells. The geometric area of the valve seat was refined by one level in order to capture geometric details accurately but no specific mesh optimization was performed. The meshing time in this case is 1.5 hours on a SGI R10000 Indigo2 workstation.

For the present investigation the following boundary conditions where applied:

- intake: total pressure
- cylinder end: static pressure

These settings correspond to the physical conditions typically found in flow bench test beds.

Results

In Figure 4, the velocity field obtained with LDA measurements is shown in a plane normal to the cylinder axis at the axial location of 1.75 x D_{cyl}. In order to assess the flow structure the velocity vectors for Mesh B is plotted in the same plane in Figure 5. The velocity vectors have been interpolated from cell positions to the exact position of LDA measurement locations.

No obvious difference between both velocity fields can be observed. A more meaningful assessment is the comparison of integral flow properties which represent the overall three dimensional flow structure. For the analysis of intake ports for HSDI engines two values are of significance:

1. the massflow through the intake port resulting from the settings of inlet and outlet pressure
2. and the swirl number

The mass flow and mean axial velocity, v_{Ax_mean}, is obtained by integration over the cylinder cross section. The following equation applies:

$$v_{Ax\_mean} = f(\rho, \bar{v}) = \frac{\int \rho v_{Ax} dA}{\int \rho dA} \qquad [m/s]$$

$$\dot{m} = f(\rho, \bar{v}) = \int \rho v_{Ax} dA \qquad [kg/s]$$

The swirling characteristic of the flow was accessed by means of paddle wheel and torque meter measurements.

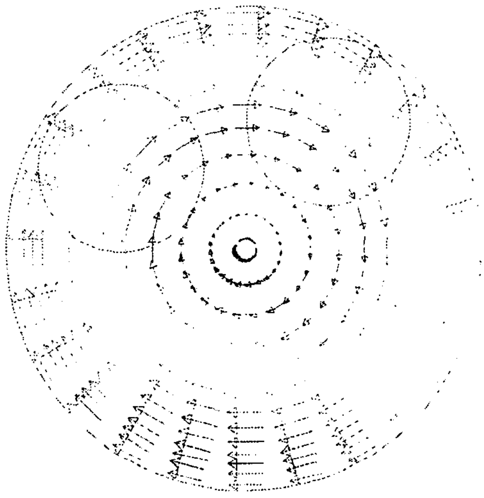

Figure 4: Velocity field obtained by LDA measurements

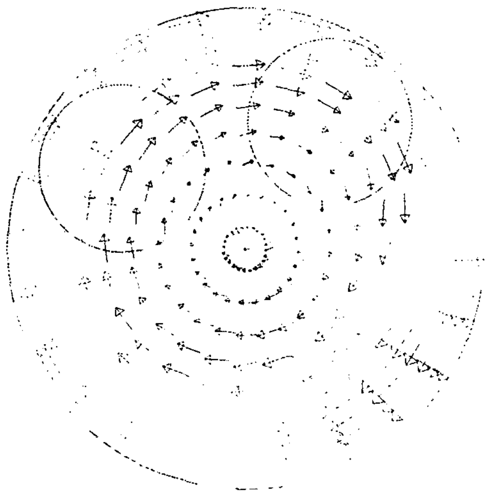

Figure 5: Computational Mesh B generated automatically by use of AVL-FAME

In order to compare the computational results with the measured torque, the following torque will be computed over the total cylinder cross section in a distance of 1.75*D from the bottom of the cylinder head (this computation will be done relating to the cylinder centre):

$$M_t = \Sigma p_i(\vec{v}_i x \vec{r}_i) \cdot \vec{n} \cdot v_{Ax} \cdot A_i \qquad [Nm]$$

$\vec{v}$ .... [m/s] Velocity

$\vec{r}_i$ .... [m] Radius

$\vec{n}$ .... [m] Normal Vector

v_{Ax} .... [m/s] Axial Velocity

A_i .... [m^2] Area Element

Furthermore, a speed is computed from this torque:

$$n_{Mt} = \frac{30}{\pi} \cdot \frac{M_t}{\Sigma p_i \cdot r_i^2 \cdot v_{iAx} \cdot A_i} \qquad [min^{-1}]$$

This speed will be compared with an engine speed,

$$n_{Eng} = \frac{30}{\pi} \cdot \frac{v_{Ax\_mean}\pi}{D} \qquad [min^{-1}]$$

v_{Ax_mean} ..... [m/s] Mean Axial Velocity
D .... [m] Bore Diameter

Which yields the following dimensionless reduced swirl number.

$$SN = \frac{n_{Mt}}{n_{Eng}}$$

Applying the above method the experimental result is

- Mass flow = 0.03594 kg/s
- Swirl Number = 1.65

In Table 1 the calculation results are summarized for several calculation runs is given:

Table 1: Comparison of integrated quantities obtained from numerical calculations

	Mesh A	Mesh B
CD	m = 0.03626 kg/s V_{ax} = 7.9 m/s SN = 1.87	m = 0.039 kg/s V_{ax} = 7.9 m/s SN = 1.28
MINMOD	m = 0.03506²kg/s V_{ax} = 6.727 m/s SN = 1.51	m = 0.03787²kg/s V_{ax} = 7.0195 m/s SN = 1.72
SMART	-	m = 0.03649²kg/s V_{ax} = 6.676 m/s SN = 1.76

Conclusion

From Table 1 one can see that the multi-block mesh shows a strong variation in result accuracy and in can be concluded that a mesh independent solution is not yet obtained. Therefor further manual or automatic mesh adoption is necessary in critical flow areas.

The results for the automatically generated mesh show that with increased complexity of the second order scheme both the flow rate and the swirl number are within engineering accuracy limits.

The application of advanced higher order schemes on automatic generated meshes is mandatory in order to reach accurate results.

References:

Gaskell, P.H; Lau, A.K.C.; 1988, "Curvature Compensated Convective Transport: SMART a new Boundedness Preserving Transport Algorithm" Int. J. for Num. Methods in Fluids, Vol. 8, pp. 617-641.

Sweby, P.K. 1984 " High Resolution Schemes Using Flux Limiters for Hyperbolic Conservation Laws", SIAM J. Num. Anal., Vol. 21, pp. 995-1011.

Muzaferija S., Gosman D., 1997 " Finite Volume CFD Procedure and Adaptive Error Control Strategy for Grids of Arbitrary Topology", J. of Comp. Physics, Vol. 128, pp. 766-787.