

Best practice guidelines for handling Automotive External Aerodynamics with FLUENT

Version 1.2
(Feb 9th 2005)

Marco Lanfrit (mgl@fluent.de)
Fluent Deutschland GmbH
Birkenweg 14a
64295 Darmstadt/Germany

This document gives a description of a straightforward and reliable way to perform simulations in the field of automotive external aerodynamics using the CFD software package FLUENT. Items and approaches listed below do not claim to be complete nor optimized, they are just recommendations based on experience with recent comparable studies.

1. INTRODUCTION

External aerodynamics is one of the main applications in the automotive industry. Engineers in the field of aerodynamics have been using CFD for a long time. Traditionally, CFD is used to verify wind-tunnel experiments, optimize car shapes in terms of lift and drag, and study salient flow features. In the last few years the simulation of aero-acoustics has become another major application, which helps the aerodynamicist to obtain fast and reliable results for a specific configuration. To use CFD during the design cycle one needs a reliable and straightforward approach, which makes it possible to compare results for different vehicles and for several alternative vehicle modifications.

CFD is always subject to some constraints not only in the field of external aerodynamics. Among other constraints, allocatable RAM, total amount of CPU time and expected response time for results have to be taken into consideration. They determine the modeling approach used in terms of cell count and the complexity of numerical models. This influences the accuracy of the results as one can see for the simulation of the Ford Ka (Table 1). As a first step the existing best practice was applied, using a so-called coarse mesh with 5.5 million cells for a complete car model including a detailed underbody. In the next step, the same numerical models have been used with a finer mesh (11.5 million cells), while in the last step a higher order turbulence model was used in conjunction with the fine mesh. Table 1 shows the results of this set of simulations for drag coefficient c_D .

mesh size & turbulence model	c_D	Δc_D	CPUhs
wind-tunnel experiment	0.321	-	-
coarse mesh (5.5 M cells) realizable k- ϵ	0.336	4.7 %	450
fine mesh (11 M cells) realizable k- ϵ	0.328	2.1 %	750
fine mesh (11 M cells) RSM	0.322	0.3 %	1200

Table 1: Comparison of results for different modeling approaches [1]

The results clearly show the improvement of simulation accuracy in comparison to windtunnel results, when more effort is put into the modeling approach. In the next sections the most common modeling approaches for Geometry Cleanup, Meshing and Simulation are discussed.

2. GEOMETRY CLEANUP AND PREPARATION

2.1 General Remarks

Most of the car geometries that are analyzed today are close to the production shape, i.e., highly detailed within all areas. During the CFD-treatment the necessary simplifications should be as few as possible. Their creation is predominantly carried out by surface-based CAD-systems due to their complexity. Unfortunately these do not guarantee closed surfaces.

On the other hand, volume-based modeling systems have the advantage of the outer surfaces being inherently closed. The eventual goal of expanding the typical vehicle model so that it includes such features as wipers, aerials, detailed underbody and all under-hood components is a limitations of such systems.

During the preparation for a flow-simulation, a consistent definition of fully connected geometry has to be ensured. This first step comprises a cleaning-up of the CAD-model, and is totally independent of the subsequent method of simulation. Suitable tools for this process would include the original CAD-systems used for definition (CATIA, IDEAS, UNIGRAPHICS, ...) as well as FE-preprocessing programs (ANSA, CATIA- and IDEAS-Mesher, PATRAN, ...). For dealing with simple volume-based geometry definitions as well as for simple surface-based models Fluent's preprocessor GAMBIT can be used. To be successful with highly complex shapes the usage of an FE-preprocessor like ANSA might be necessary.

2.2 Vehicle Geometry

The surface mesh must be as smooth as possible to allow prism layers to be extruded from the surface of the car (except the underbody if it is too complex). Sharp angles (like spoilers) must be avoided; the creation of additional faces, which later will be treated as "interior", is a trick to close or smooth out those regions. TGrid can then be used to fill the gap between these additional "membrane-like" interior faces and the real surface of the car with tets. Cavities like air intakes should be closed in a similar way and treated as separate volumes during volume meshing (see Figure 1).

There will also be sharp angles between the wheels and the ground plane. This is a source for highly skewed cells. It is necessary to blunt this angle by introducing small faces connecting ground and tyres.

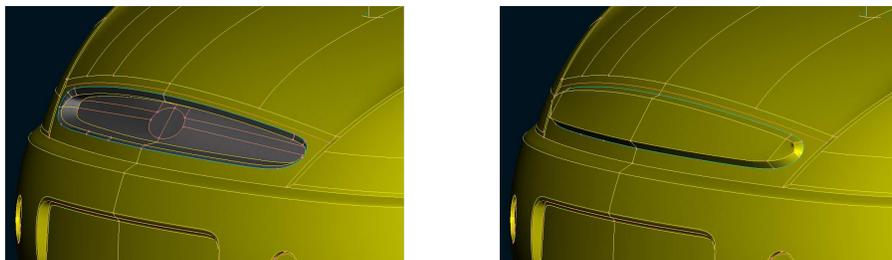


Figure 1: Additional Faces to improve quality of prism elements (right picture)

2.3 Computational Domain (Wind Tunnel)

The dimensions of the computational domain should be at least 3 car lengths in front of the car, and 5 lengths behind. The displacement of the car should be not greater than 1-1.5% of the total cross sectional area. This is not applicable if the domain boundaries represent the walls of a real wind-tunnel. In this case the simulation should take into account the related wall effects.

In order to control the volume mesh near the car, an "inner" box may be helpful. This box should extend about half a car length in front, to the sides and to the top, and about a car length in the wake. These boxes are used for building the domain in TGrid. This "inner-box" is not mandatory, due to the possibility of local refinement in TGrid. The user should think about the aimed volume meshing approach, prior to creating the domain. For Hexcore Meshing an inner box is not needed.

3 MESHING

3.1 Preliminaries

The faceted triangular surface resolution has to meet several requirements. For a typical car-like shape, pressure or form drag is dominant over skin friction, so the accuracy of the drag and lift predictions are largely determined by the accuracy of the predicted static pressure distribution on the body. This pressure distribution is strongly affected by the locations of flow separation and reattachment. Therefore it is important that the surface mesh resolves all relevant details of the geometry and satisfies the requirements of the physical models used in the simulation.

For high-Reynolds-number flows such as the flow around vehicles, it is a well recognized fact that resolving the near-wall region down to the viscous sub layer is not a practical option because the number of cells that must be allocated in near wall regions is prohibitively large. To overcome the well-known drawbacks of traditional wall-functions, FLUENT offers the possibility to use the so-called non-equilibrium wall functions (NWF's). NWF's are sensitized to pressure gradient effects. This feature is of great benefit to the prediction of ground-vehicle aerodynamics. Besides being sensitized to the pressure gradient, NWF's account for the effects of local variation in the thickness of the viscous sublayer, when computing the turbulent kinetic energy budget in wall adjacent cells.

A first step in the standard procedure is to calculate an average surface element size, by means of desired y^+ - values for near wall modeling. This size will later be assigned to all edges of the vehicle, to get an initial distribution for the surface mesh.

Figure 2 shows a graphical method for determining an average element length based on the free stream velocity and aimed degree of surface resolution.

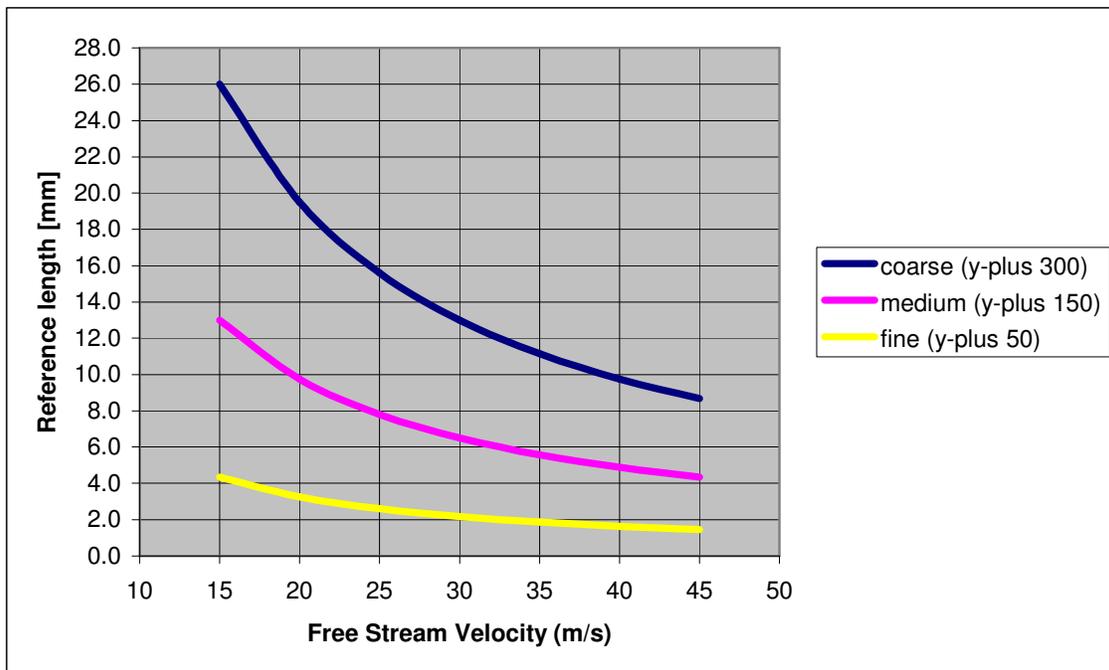


Figure 2: Estimating Surface Mesh Reference Length

The user should first decide which degree of resolution is needed for the simulation, and what resources are available. Choosing a coarse surface triangulation will lead to an initial mesh of approx. 2-5 million cells. A medium resolution, which currently is part of the standard approach, will lead to meshes that consist of approx. 5-10 million cells, while a fine resolution will correspond to meshes

beyond 10 million cells. The overall number of cells highly depends on complexity of the geometry and the settings for volume meshing.

3.2 Meshing Strategies

In general there are three different strategies to create the volume mesh for simulation in FLUENT 6.

- Strategy A (Adaption)
- Strategy B (Boxes)
- Strategy C (Controls)

In this section, the advantages and disadvantages of these three approaches are discussed.

3.2.1 Strategy A (Adaption)

This simulation strategy uses the adaption functionality in the FLUENT solver. A relatively coarse mesh is used as a starting point, and a first solution is calculated. To complete the simulation, several static pressure gradient adaptations are necessary. In each adaption cycle about 1-3% of the total number of cells should be refined using the hanging nodes adaption method in FLUENT 6. Then further iterations are needed until convergence of drag and lift coefficients is achieved. Overall 3 to 5 adaption cycles may be necessary to reach a state where the force coefficients, or any other important parameter, will no longer be subject to any significant changes.

In order to decrease the total number of cells, the cells to be adapted (marked) can be limited to a region not too far from the car. To do so, you must create one register using static pressure gradient and another grouping the cells inside a hexahedral region near the car. Then you can intersect them by changing the type of the hexahedral register from adapt to mask and combining the two registers.

Strategy A is the quickest meshing approach, but is not optimal due to several facts:

1. Since the FLUENT solver in Version 6.1 has no access to the grid's original geometry database, mesh adaption is not useful for improving the geometry resolution of the surface mesh.
2. By using the hanging nodes adaption functionality, numerical instability and maybe numerical diffusion is introduced by large size gradients of neighboring cells.
3. Adaption needs several manual interventions by the user.

3.2.2 Strategy B (Boxes)

This strategy is based on internal boxes created around the vehicle and in the wake region to explicitly control mesh size. This approach is more time consuming than strategy A, but very accurate. The boxes are typically created in the Preprocessing tool. A constant size of surface elements is applied to the box walls. The boxes are used in TGrid as meshing domains, in which cell size can be controlled in a very comfortable way. Another advantage of this method is the possibility to combine different meshing techniques in one model. Hence Hex meshes can be used for the outer region of the wind tunnel and can be kept for different simulations, while only the inner box is changed. The connection between those boxes can be either conformal using pyramids or non conformal by using the interface definition of FLUENT.

3.2.3 Strategy C (Controls)

In this strategy the internal boxes are replaced by virtual boxes (local refinement boxes) used for cell refinement in TGrid. Therefore the local refinement functionality in TGrid is used. This approach can either be used for general Tetrahedral or Hexcore Meshing. These methods will be described in the Volume Meshing section of this document (see Section 3.4.5). This approach is very accurate and avoids the creation of additional surfaces in prior steps. This strategy is recommended by Fluent.

3.3 Surface Meshing

The first step in the surface meshing procedure is the imposing of the estimated average element size on the whole vehicle geometry. Depending on the tool used for surface meshing this step gives the user the possibility to identify locations where geometry simplifications have to be applied or a higher degree of mesh resolution is needed to capture the geometric details.

To locate problems caused by geometric details, like narrow gaps, or sharp angles, the user should control the overall quality (skewness) of the surface mesh. Figure 3 illustrates a typical problem caused by sharp angles (upper left). Close to the origin of the sharp angle, highly skewed elements are created (upper right). This problem can be solved by either merging faces together and ignoring the detail (lower left), or by keeping certain areas of the detail and merging the faces that cause the sharp angle. This can either be done manually or by using automated tools, like the virtual clean-up tools in GAMBIT 2.1.

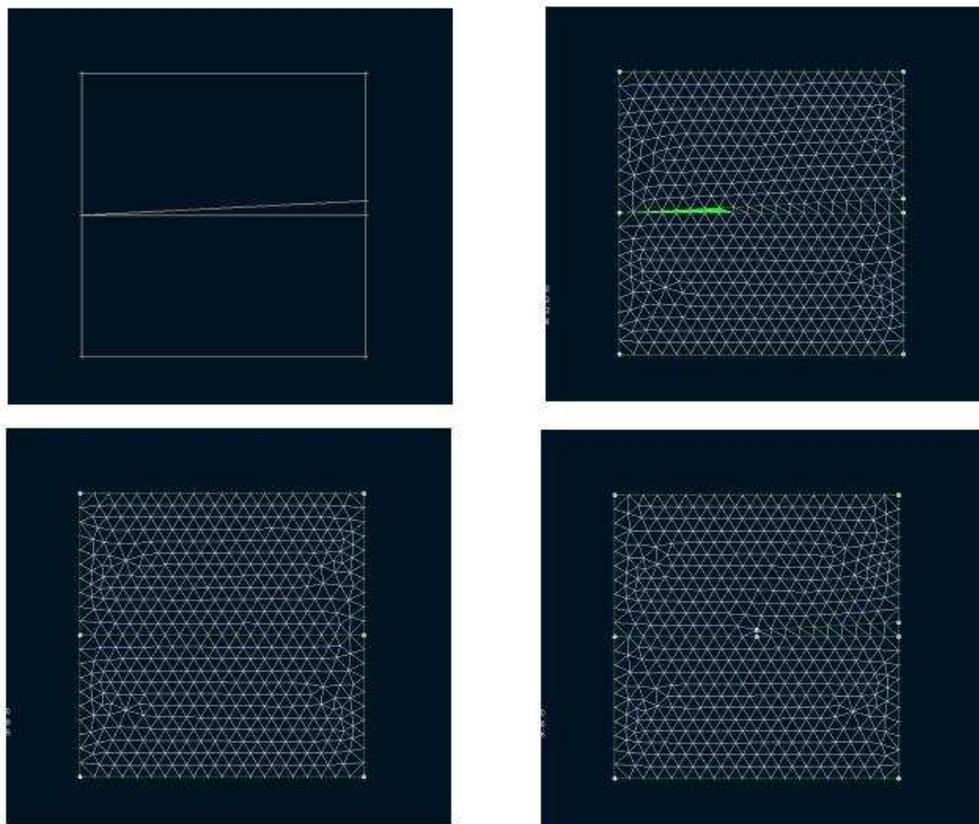


Figure 3: Sharp angle causing highly skewed elements

Figure 4 illustrates another typical problem caused by details during surface mesh creation. A narrow gap (upper left), which could be a typical detail like an eaves gutter on the roof of a car or the gap at the hatchback, hood or doors. One has to decide whether one wants to maintain this detail or ignore it. If the user wants to ignore it, the edges of the detail have to be removed by merging together the faces (lower left). If one has decided to keep the detail because its impact on the flow is important, one has to refine the mesh in this region and provide a good transition to the general mesh size (lower right). This transition should involve a size jump of no more than 20% from one element to the next. Common preprocessing tools provide functionalities to accomplish this, such as Sizing Functions (e.g., GAMBIT) or Spacing Functions (e.g., ANSA).

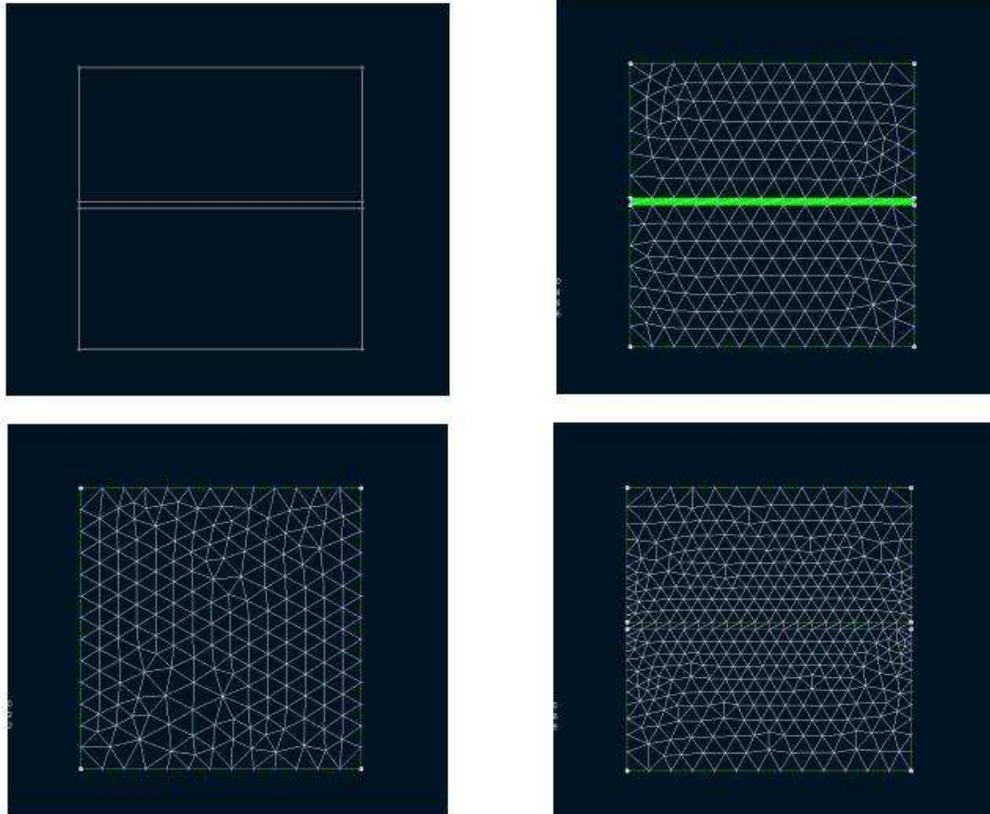


Figure 4 Geometrical Detail causing highly skewed elements

In the next step user has to take care that the mesh is able to represent the shape of the parts. This is essential for parts close to stagnation points (front bumper, wheels, sideview-mirrors) and at the back of the car (c-pillar, spoiler), to accurately describe separation. Figure 5 shows an example of the a sideview-mirror with two different mesh resolutions.

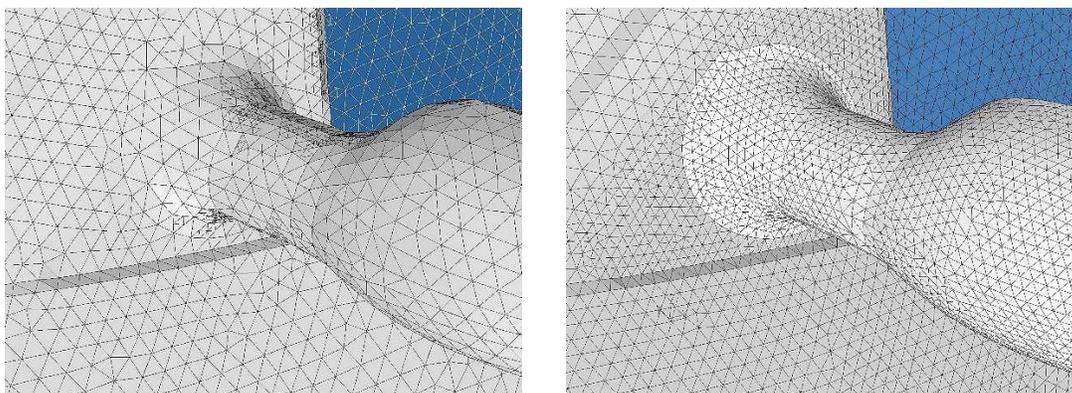


Figure 5 Mesh Resolution for sideview mirror

The surface meshing should result in a high quality, non-uniform triangular surface mesh that resolves all radii and geometrical details well.

Summary:

- Radii and sharp edges should be resolved very accurately, while planar faces can be meshed relatively “coarse”
- Maximum growth rate of surface elements should be 20%, even from radius to planar face and vice versa
- Skewness of the surface mesh should be as good as possible, ideally <0.45

3.4 Volume Meshing

Volume Meshing comprises of two main steps. For real industrial car shapes a hybrid meshing approach using all the available element types in FLUENT 6 is adequate. To ensure low skewness elements where viscous effects are large, prism elements are extruded from the car surface in a first step. Following this, the rest of the domain is filled either with hexahedral and/or tetrahedral elements.

3.4.1 Prismatic Layers

Layered elements provide good alignment with the flow near wall boundaries. This is beneficial for reducing numerical diffusion. For the creation of prism layers, Fluent’s preprocessing software TGrid is used. A basis for a high quality prismatic mesh in the highly affected viscous regions is a surface mesh using the quality criteria described in the above section. Prism Layers are extruded from the windtunnel floor and most of the car surfaces, like roof, side, back, engine-hood, and windscreen. Using prisms on a complete detailed underbody geometry is not possible, due to prism layer creation problems caused by undercuts and sharp angles. Nevertheless flat regions, especially on the front part of the underbody should be resolved using prismatic layers.

TGrid 3 allows prisms to be extruded for a specified value of cell aspect ratio. Extrusion by aspect ratio is recommended for all prism layers on a vehicle. This makes each prism’s height proportional to the size of its base triangle. For good prism characteristics with extrusion by aspect-ratio, the triangle size on each surface must vary smoothly.

Each succeeding prism layer should increase in height at a constant geometric rate. To ensure a mesh satisfying the numerical requirements, one must assure a good transition, in terms of cell-size-deviation at the interface between the prismatic layers and the tetrahedral region. To avoid large cell-size gradients that may lead to numerical diffusion, an optimal combination of settings for first aspect ratio, growth rate and number of layers has to be determined.

Figure 6 shows two different prism layers based on the same triangular surface mesh. The prisms on the left side were grown using a first aspect ratio of 5, a growth rate of 20% extruding 5 layers. The transition to the tetrahedral elements is smooth. On the right hand side a prism-layer is shown, using an aspect ratio of 10 for the first layers, maintaining the values given above for growth rate and number of layers. It can be clearly seen, that there is a huge volumetric size gradient at the interface between prismatic and tetrahedral elements.

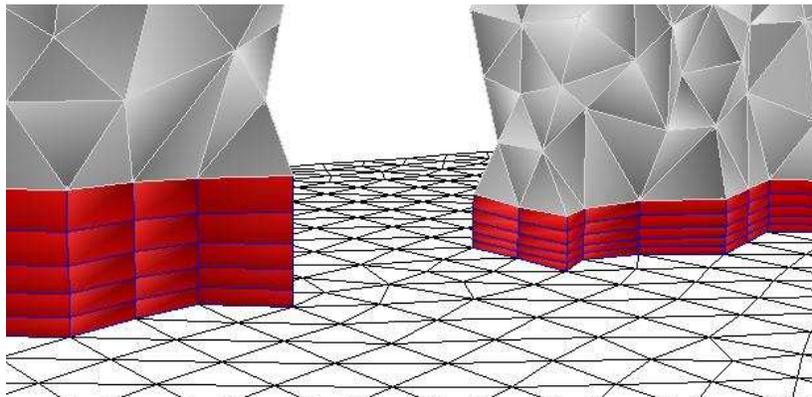


Figure 6 Prism Layer Growth

The recommendation for prism layer growth on the vehicle surfaces is First Aspect Ratio: 5, Geometric growth rate: 1.2, Number of Layers: 5 (also see Figure 7).

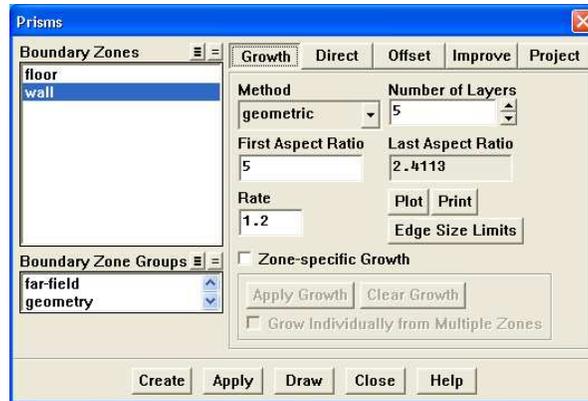


Figure 7 Prism Layer Growth - Panel

Growing prisms on the windtunnel-floor is done using a uniform growth method. Therefore a First Height, Growth Rate and Number of Layers has to be specified. The First Height is determined by the average surface element size of the floor under the car. To ensure the proper boundary layer resolution in this highly affected region, it is important to have a numerically fitted mesh in this particular region, just as on the car surface. Therefore first height is calculated using the aspect ratio approach given above. The first height should be a fifth of the average surface element size.

3.4.2 Hybrid Mesh Transition

Due to the complexity of some parts of the car, especially the underbody, it is not possible to cover the car completely with prismatic elements to create a closed O-Type Grid. In this case the prism-sides, consisting of quadrilateral elements, are present in the domain. There are two different ways of treating this issue. The first is using the automatic or semi-automatic creation of pyramid elements. This yields a so called conformal mesh, in which the pyramids' quadrilateral bases cover the quadrilateral side faces of the prism elements, and then each pyramid's four triangular side faces are connected to tetrahedral elements. This is automatically done while TGrid creates the tetrahedral mesh. Due to the size variation of the quadrilateral elements with respect to the connected surface triangles, this may result in highly skewed pyramids and/or connecting tetrahedral elements.

Therefore it is recommended to use the FLUENT 6 non-conformal interface feature to connect the prism-layer to the surrounding tetrahedral mesh. TGrid offers a comfortable way to create a copy of the prism-side surface and to automatically remesh the copy using triangular elements while maintaining the node distribution on the bounding edges (see Figure 8).

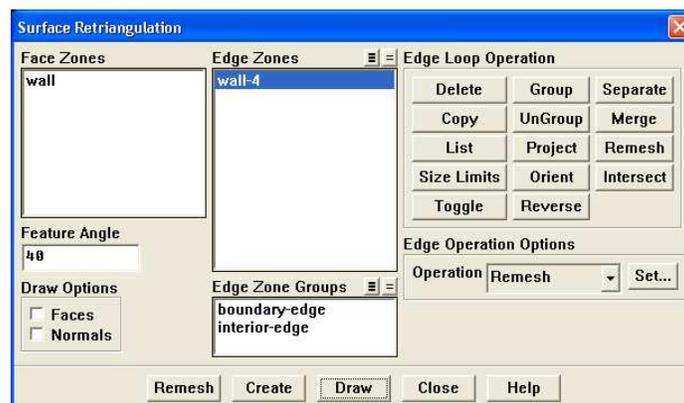


Figure 8 Surface Retriangulation – Panel

Figure 9 shows the application of non-conformal interfaces for the Ford Ka model. On the left hand side, the prism side is shown consisting of quadrilateral elements. The right hand side shows the retriangulated copy consisting of high quality triangular elements, which can be directly connected to the tetrahedral mesh.



Figure 9 Handling Prism Sides using Non-conformal Interfaces

3.4.3 Tetrahedral Mesh

There are two ways to fill the rest of the domain. In the past, tetrahedral elements were used to envelop all other cell types. Their completely unstructured nature allows them to grow to unlimited size, while maintaining conformal connectivity with their neighbors. Hence they efficiently fill the remaining volume of the domain. This facilitates calculations for domains with a very low level of solid blockage, as necessary for accurate external-aero predictions. TGrid offers a way to automatically create a high quality tetrahedral volume mesh by applying just a few important parameters (see Figure 10).

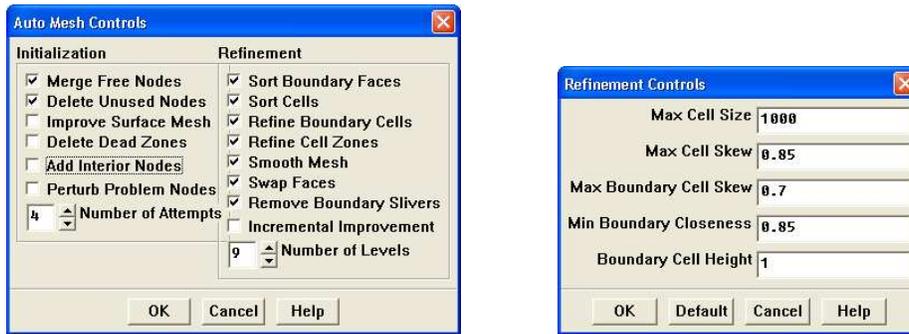


Figure 10 Tetrahedral Mesh Generation - Controls

In the Auto Mesh Controls Panel it is important to set the number of Refinement Levels to 9. This ensures a tetrahedral growth rate of about 20 % in the domain. Under Initialization, Improve Surface Mesh should be disabled.

To ensure a continuous growth of the tetrahedral elements to the boundaries of the domain, the Max Cell Size should be limited in the Refinement Controls. This value is given in terms of volume units. A recommendation for this value is based on the surface elements size on the bounding box (l_{bound}), which should be around 100 - 200 mm.

$$MaxCellSize = l_{bound}^3 / 5$$

3.4.4 Hexcore Mesh

TGrid 3.6 offers the possibility of Hexcore Meshing. Hexcore Meshing is a hybrid meshing scheme that generates cartesian cells inside the core of the domain and tetrahedral elements close to the boundaries. Hanging-node refinements on the Cartesian cells allow efficient cell size transition from boundary to the interior of the domain. This results in fewer cells with full automation and can handle complex geometries, internal walls and gaps.

The Hexcore meshing scheme is useful mainly for volumes with large internal or external regions.

Similar to the tetrahedral meshing, TGrid offers several commands to generate the Hexcore Mesh. A general description of how Hexcore works is given in the TGrid Users Guide section 7.3.

The main controls for Hexcore Meshing are Maximum Cell Length and Buffer Layers.

Maximum Cell Length

For Hexcore Meshing, the Maximum Cell Length is based on length units. If no Maximum Cell Length is specified, TGrid toggles automatic sizing of the domain, and calculates the maximum length using the domain extent and the surface mesh size. The recommendation is to either use Auto Size Domain, or limit the Maximum Cell Length to twice the element size on the bounding box.

Buffer Layers

When there is disparity in size distribution between the boundary mesh and the initial Cartesian cells, the transition from fine to coarser cells can be too rapid. To avoid this, TGrid marks additional layers of cells adjacent to those marked by the size requirement. Increasing the Buffer Layers will significantly increase the number of cells. Therefore the default value of 1 is recommended.

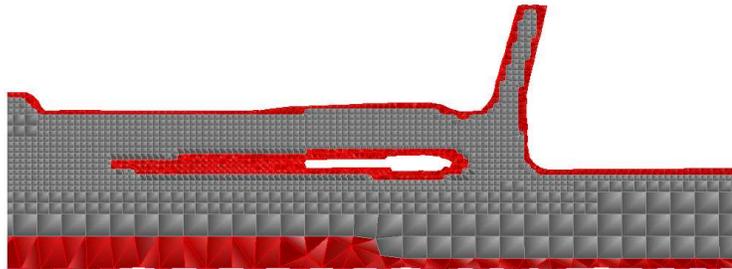


Figure 11 Hexcore Mesh Close to the Underbody

3.4.5 Local Refinement

For both methods, pure tetrahedral and hexcore meshing, local refinement boxes can be defined. It is strongly recommended to define such virtual refinement boxes, using Local Refine for Tets, or Region for Hexcore, prior to the main Meshing Process. This assures that the majority of elements will be located close to the vehicle and in the wake region. Figure 12 shows an example of the Ford Ka, with and without local refinement boxes.

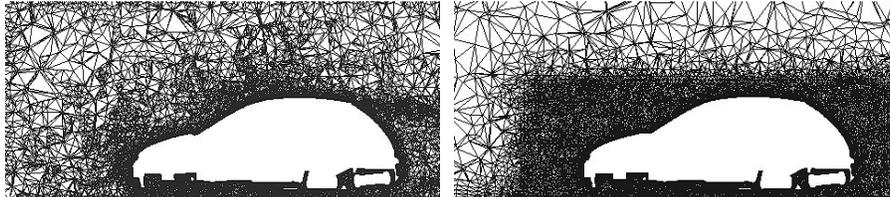


Figure 12 Impact of Local Refinement on Tetrahedral Mesh

3.4.6 Checking Quality

To achieve accurate and reliable results, it is important to fulfill the following requirements in terms of quality:

- Make sure that a particle coming from the inlet area and following a path to the front stagnation point of the car would have to pass through minimum 100 cells. If this region is under-resolved, the pressure coefficient will go far beyond 1 and spoil the overall solution.
- Check that the largest cells within the flow domain are smaller than those attached to the bounding box (for pure tetrahedral meshes)
- Check that Quality of cells is below 0.9 for the whole domain. It is recommended to have a quality of below 0.85 for prism elements on the car surface. Use TGrid's Mesh Repair functionalities to fix local quality problems.

4. SOLVING

4.1 Boundary Conditions

The specification of boundary conditions should be geared as close as possible to the measurement conditions in the wind tunnel. In the majority of cases, flow velocity and turbulent intensity of the wind-tunnel are known. Therefore a velocity-inlet boundary condition is used to model the incoming flow. Velocity Magnitude and the Flow direction are specified, completed with Turbulent Values at the inlet. The user can choose between several options to specify those.

Effects like rotating wheels and moving belt can be modeled using the Moving/Rotating Wall Boundary Condition. This adds tangential velocity to the selected walls.

In cases where boundary layer suction is present, the relevant areas have to be defined as separate wall zones and shear stresses have to be set equal to zero.

Specify a Symmetry Boundary Condition for the rest of the domain (e.g., the windtunnel's top and sides).

4.2 Turbulence Modeling

The fidelity of CFD predictions for turbulent flow is highly dependent upon the quality of the turbulence modeling. This is even more important for the flow around ground vehicles, whose salient flow features include three-dimensional boundary layers with strong streamline curvature, separation and strong vortices. These features require turbulence models that can properly account for non-equilibrium effects and anisotropy.

Based on experience, Fluent recommends two types of turbulence models for external aerodynamic studies.

Realizable k-epsilon Model

The Realizable k-epsilon Model proposed by Shih et al. [5] was intended to address the well-known deficiencies of traditional k-epsilon Models by adopting the following:

- A new eddy-viscosity formula involving a variable for C_{μ} originally proposed by Reynolds
- A new model equation for dissipation based on the dynamic equation of the mean-square vorticity fluctuation

Industrial applications of this model show that it is possible to achieve good results in terms of integral values (e.g., drag coefficient), which are within 2-5 %. Due to its implementation it is very stable and fast converging. Therefore it is perfectly suited for automated calculation processes, allowing a huge number of calculations in a relatively small time frame (see Table 1).

Reynolds Stress Model (RSM)

In the last few years, the employment of second-moment closure models in which transport equations for the individual Reynolds Stresses are solved, has become more and more widespread. RSM rigorously accounts for anisotropy of turbulence and the transport of all Reynolds Stresses. But these advantages come at the price of a higher cost in computational time (+40%) and RAM resources (+20%). The RSM Model in FLUENT 6 is based on the models of Gibson and Launder [6] and, more recently, Speziale et al. [7], and it is implemented in the framework of an unstructured mesh. RSM is recommended for high-fidelity simulations, in which salient flow features and structures are studied and compared.

If RSM is applied, it is recommended to use the following settings:

- Initialize the Flow Field (do not use flow field calculated using a 2-equation Model)
- Set Under Relaxation Factors to:
 - 0.65 for Pressure,
 - 0.35 for Momentum and
 - 0.50 for k and epsilon
- Change formulation for the epsilon-equation for external aerodynamics with RSM, by setting the following SCHEME Commands:
 - (rpsetvar 'drsm/coupling-alt-average? #t)
 - (rpsetvar 'realizable-epsilon? #t)
- Request 50 Iterations with FIRST ORDER DISCRETIZATION, then switch to SECOND ORDER

- If convergence problems occur right at the beginning of the calculation, set Under Relaxation Factors for k and ϵ to 0.2 for 50 iterations, then switch to 0.5 and after that switch to SECOND ORDER DISCRETIZATION

Near Wall Modeling

For high Reynolds Number Flows, such as flows around ground vehicles, it is a well-recognized fact that resolving the near-wall region down to the wall is not practical. Therefore semi-empirical Wall functions are used. To overcome the well-known drawbacks of traditional wall-functions, FLUENT 6 offers non-equilibrium wall-functions (NWF's). Besides being sensitized to the effects of pressure gradients, NWF's account for the effects of local variation in the thickness of the viscous sublayer, when computing the turbulent kinetic energy budget in wall-adjacent cells.

It is strongly recommended that NWF's be selected for external-aero simulations. Compared to the traditional wall functions, NWF's provide more realistic predictions of the behavior of turbulent boundary layers, including flow separation, and they do so without a significant increase in either CPU-time or dynamic memory.

4.3 Steady State Calculation

Use the following procedure to start the simulation:

- Make sure that all boundary conditions are accurately defined
- Define Non-Conformal Interfaces (if existent)
- Enable Second Order Upwind for Momentum, Turb. Kinetic Energy and Turb. Dissipation Rate
- Initialize the flow field with zero velocity and turbulence values from the inlet
- Activate Monitors for Lift and Drag of the vehicle (make sure that reference values are set properly)
- Define Monitor points for Pressure or Velocity at a particular point in the wake region
- Request Iterations
- Carry on calculation until all monitors show constancy.
- Check plausibility by plotting C_p , Compare C_d and C_l to windtunnel results
- Check for y^+ -values (30-300)

4.4 Transient Calculation

Especially for specific kinds of car-shapes, such as cars with a hatchback configuration like the Ford Ka, Mercedes A-Class or Audi A2, the flow field shows huge transient effects especially in the wake. This is mainly caused by the absence of the counter rotating vortex pair known from notchback configurations like the SAE Reference Body, Ahmed Body or cars with distinct c-pillars. These strong vortices stabilize the flow field in the wake region and make it more or less "steady-state". For Hatchback configurations an oscillation of this wake can be detected [1].

During the steady-state solution process, time-dependence of the configuration can be detected by several criteria:

- Residuals especially for Reynolds Stresses don't "come down"
- Monitors for Drag and Lift are oscillating around a constant value
- Solution process takes a long time in terms of number of iterations
- Plot of Velocity Vectors on a plane in the wake of the car, shows variance during the iterative process, even if residuals are nearly constant (vectors can be displayed via the Animate Panel in FLUENT 6, and one might display them every 250 iterations)
- Cells can be marked by the behavior of the residuals, via ADAPT → ISO-VALUE → RESIDUALS. For Iso-Max set Maximum Value, for Iso-Min set a value of 1% from Maximum Value.
In most of the cases Residuals of the Reynolds Stresses are higher than the rest. These residuals can be accessed via **solve/set/expert** in the Text User Interface, since by default only Mass Imbalance is accessible.

Therefore it might be useful, but time-consuming, to set up a transient calculation. To estimate a time-step size the following approach is recommended:

- Calculate the frequency of the expected transient effects based on Strouhal Number

$$St = \frac{l \cdot f}{U} \Rightarrow f = \frac{St \cdot U}{l}$$

St: Strouhal Number = 0.25, l : characteristic length = sqrt (projected front area), f : Frequency,
U: Velocity Magnitude

The Frequency gives the number of periodical variations per second. Each variation should be resolved by at least 30 time steps, although 50 would be better.. This gives the time-step for the calculation. The correctness of the time-step is verified if the continuity residual drops about 2 orders of magnitude from the beginning of the time-step within 20 iterations. To obtain a meaningful solution for time averaging a periodical behavior of the flow field has to appear. Therefore at least 10 (20 would be better) periods have to be calculated.

Besides the transient parameters, the model set-up is the same as for steady-state simulations.

5. CONCLUSION

The document gives an overview of the current capabilities and recommendations for modeling external aerodynamics with FLUENT, covering geometry, mesh and case set-up for steady-state and transient simulations. It is proposing a practical approach that is geared to produce CFD results that are reliable and can be achieved in a reasonable turnaround time.

REFERENCES

- [1] **A Best- Practice for High Resolution Aerodynamic Simulation around a Production Car Shape**
4th MIRA International Vehicle Aerodynamics Conference in Warwick, UK, October 16-17, 2002
by W. Seibert, M. Lanfrit, B. Hupertz and L. Krüger
- [2] **Advances in External-aero Simulation of Ground Vehicles Using the Steady RANS Equations**
SAE 2000 World Congress, Detroit; SAE Technical Paper No. 2000-01-0484
by F. Makowski and S.E. Kim
- [3] **CFD in Aerodynamic Design Process of Road and Race Cars**
European Automotive Congress, Bratislava, Slovakia, June 18-20, 2001
by W. Seibert
- [4] **Some Remarks and Techniques for handling Autom. Ext. Aerodynamic Set-up and Simulation**
A technical Note by W. Seibert
- [5] **A New $k-\epsilon$ Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation**
T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu
Computers Fluids, 24(3):227-238, 1995.
- [6] **Ground Effects on Pressure Fluctuations in the Atmospheric Boundary Layer**
M. M. Gibson and B. E. Launder
J. Fluid Mech., 86:491-511, 1978.
- [7] **Modelling the Pressure-Strain Correlation of Turbulence: An Invariant Dynamical Systems Approach**
C. G. Speziale, S. Sarkar, and T. B. Gatski
J. Fluid Mech., 227:245-272, 1991.