Different Mesh Types in MSC CRADLE CFD

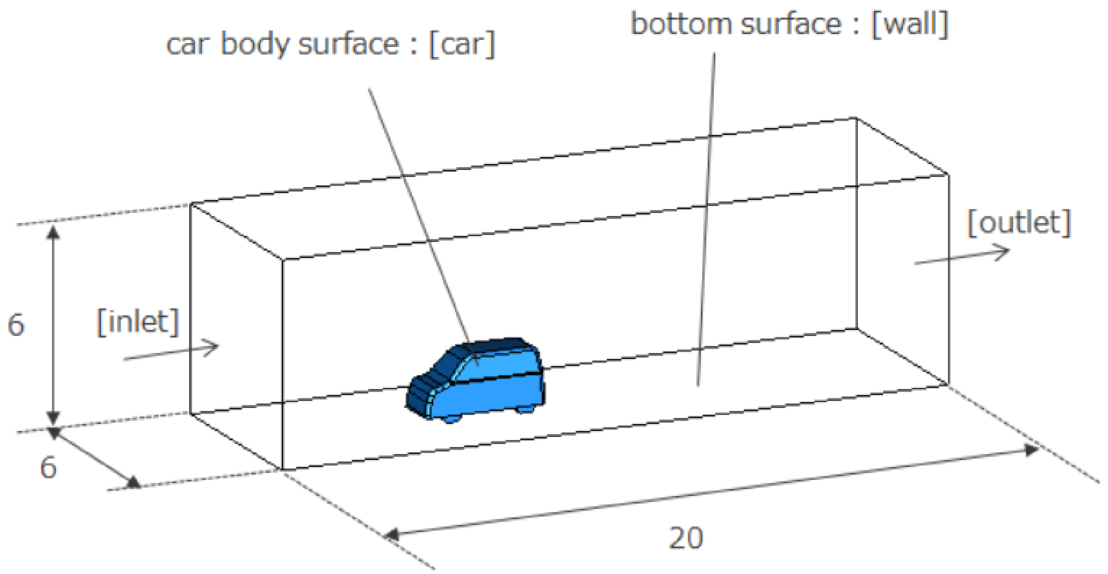
PREPARED/EDITED BY	CHECKED BY
SERHAN TARIK MEMUK INTERN ENGINEER	M.AHMET KOZANOGLU LEADER CFD ENGINEER

Abstract

In Computational Fluid Dynamics, many governing equations help us to solve even the most complex cases. However, these equations cannot be solved directly for the given CAD file. These geometries must be divided into mathematically defined finite elements. Mesh elements help us to apply these equations to the given problem. Therefore, having an appropriate mesh is one of the key factors that affect the computation directly. The mesh affects the accuracy, convergence, and computation time. In this paper, three different mesh type is studied over the same CFD Model.

CFD Model

The air flows into the domain from the inlet with a velocity of 10 m/s, turbulent flow is analyzed. Momentum equations, Mass conservation equation and k-EPS equations are solved. Computational Domain is incompressible air at $20°C$. Drag Forces are calculated at the end of the simulation.



t

Figure 1 Computational Domain

Mesh Information

Same octree parameters are used for all meshes. 0.35 is set for min/max octant size. In the region near the car surface, the octants are set to 0.05 in the range of 2.

Polyhedral Mesh, Voxel Fitted Mesh and Tetrahedral Mesh is applied on the same model. For all meshes, near wall and car surface, mesh variation rate of thickness is set to 1.1 and number of layers are 2.

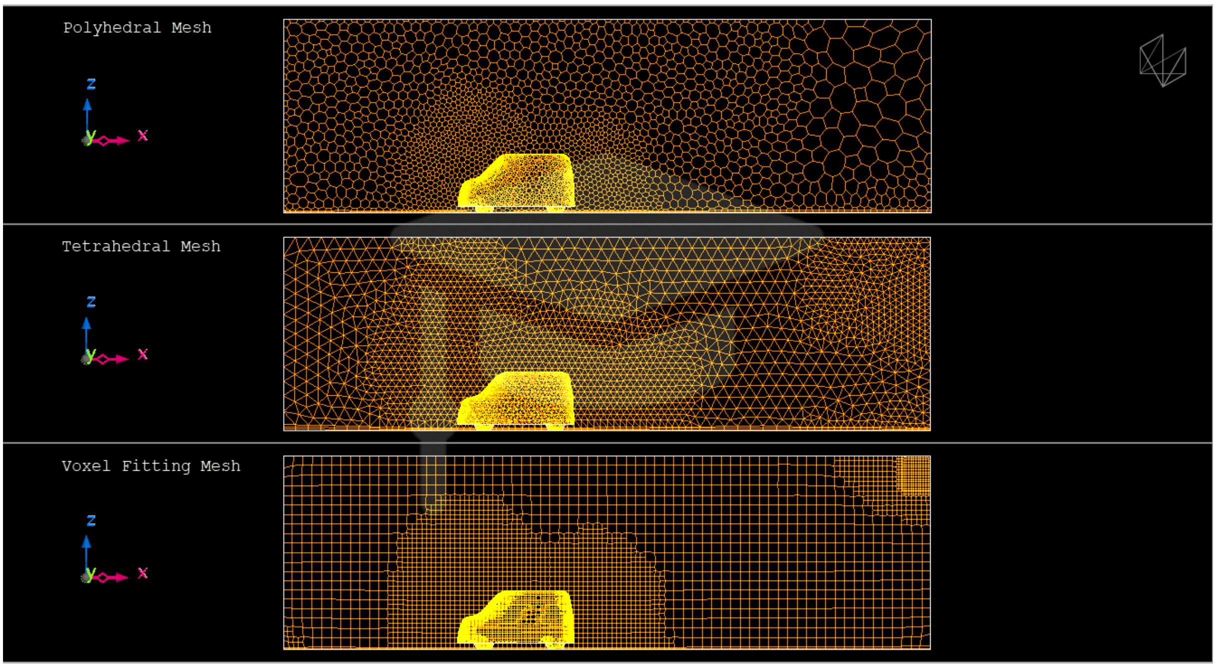


Figure 2 Types of Mesh

Mesh Adaptation

Mesh adaptation is used in this study. After each analysis, the domain is remeshing by emphasizing pressure gradient and concentrating on mesh division where velocity gradient is large.

	# of Repetition	Ratio of # octants first to last analysis	# of Cycles
Polyhedral	5	5	107>121>133>151>164
Tetrahedral	5	3	184>305>322
Voxel Fitting	5	5	400>162>149>175>194

Table 1

Results

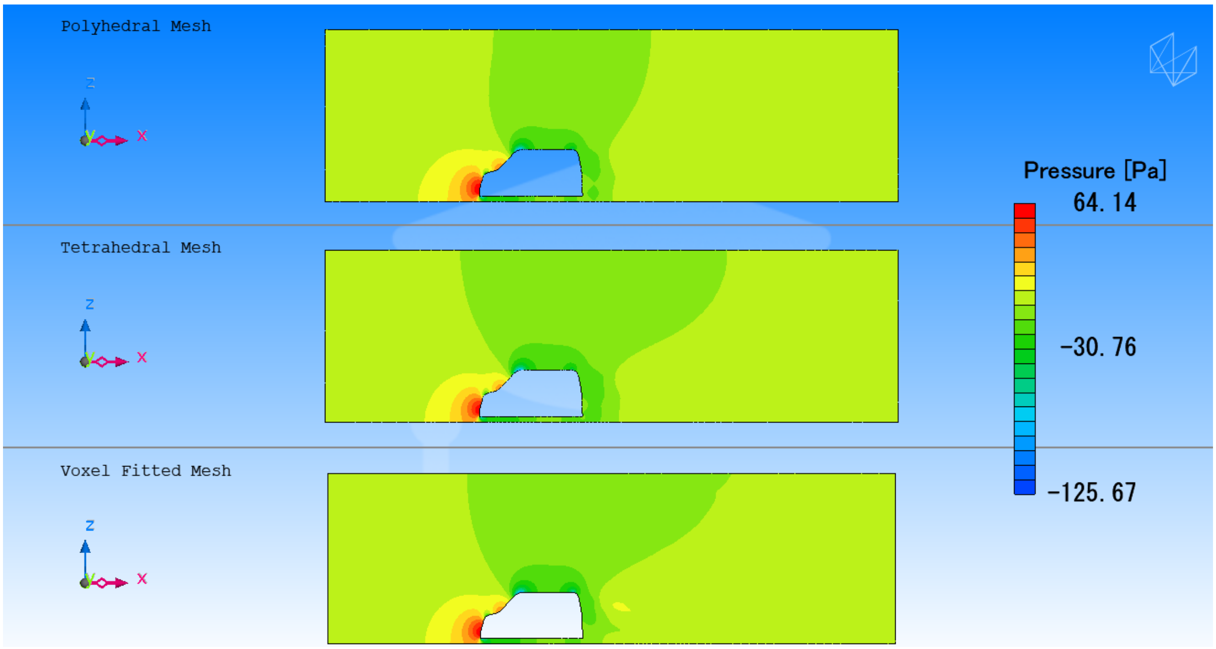


Figure 3 Pressure Contours

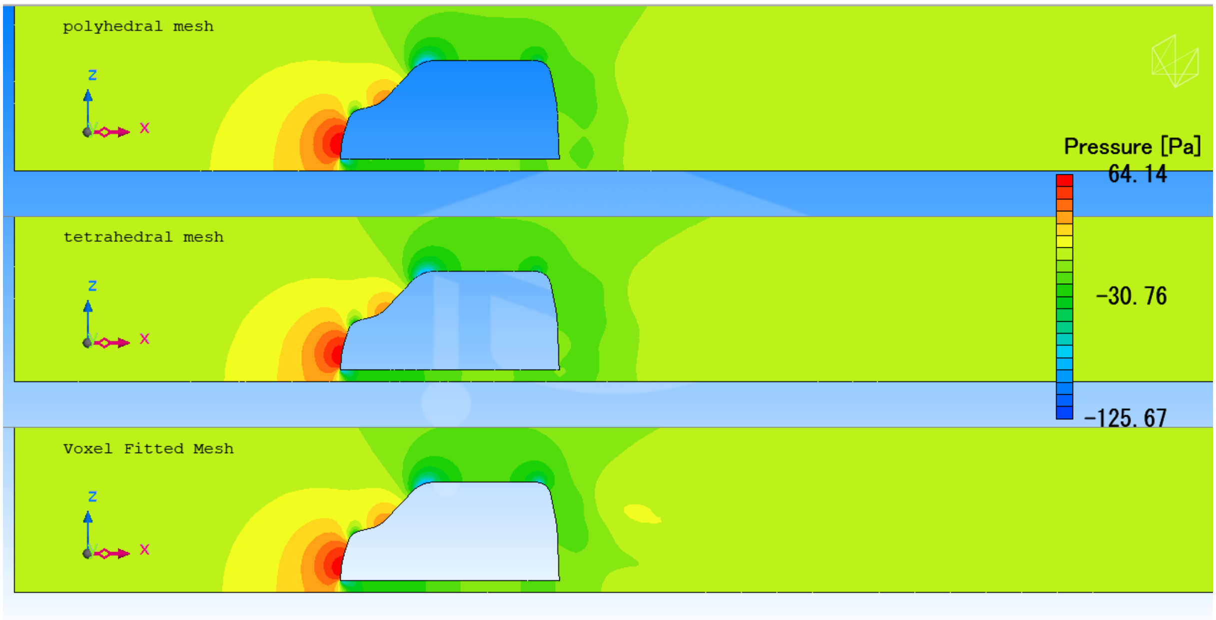


Figure 4 Pressure Contour Detailed View

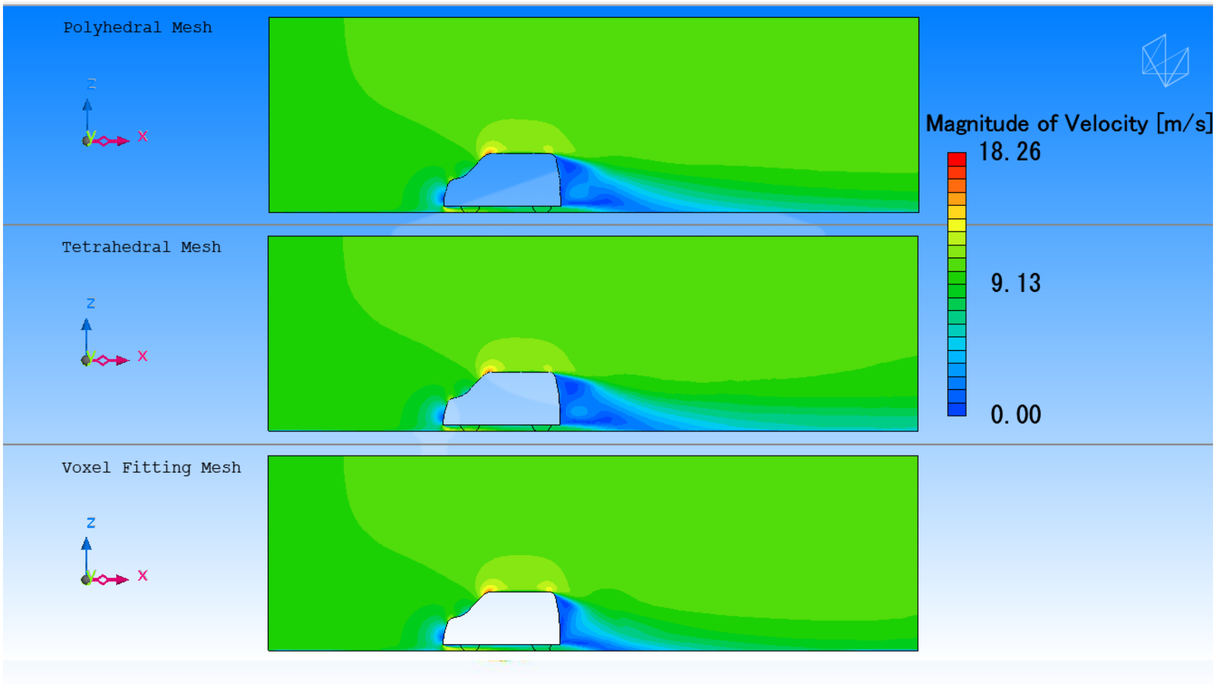


Figure 5 Velocity Contours

Drag Forces

Polyhedral Mesh	48.61 N
Tetrahedral Mesh	47.36 N
Voxel Fitting Mesh	54.41 N

Discussion

According to pressure contour figures, the pressure distribution seems similar. Maximum pressures occur nearly in the same coordinate in each case, and there are around 3% differences between maximum pressures. However, looking at the upper part of the car body, the fluid domain looks insufficient, and further enlargement is needed to optimize the analysis.

According to Magnitude of Velocity contour figures, Polyhedral and Tetrahedral mesh configurations gave similar distributions. However, In the VF mesh, the wake region is relatively different. However, the maximum velocity occurs almost in the same coordinate, and the error between maximum velocities is nearly 2%.

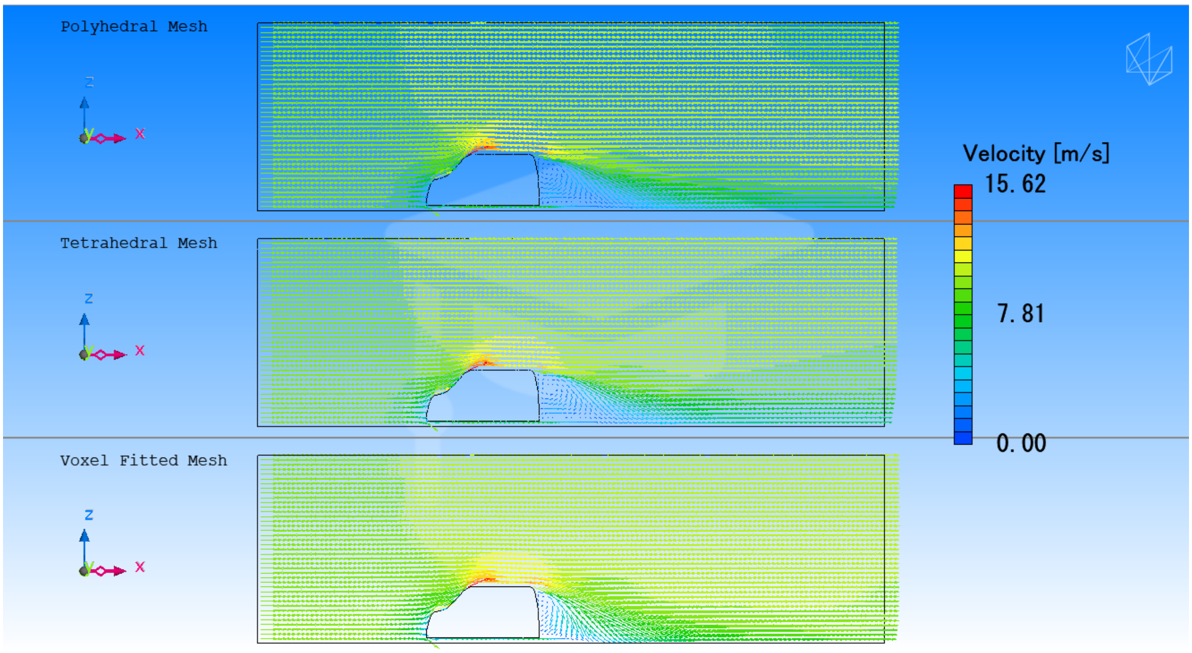


Figure 6 Velocity Vectors

Finally, comparing the Drag Forces on three different configurations, the error between Polyhedral Mesh and Tetrahedral Mesh is around 2.5%. On the other hand, there is more than 10% error between Voxel Fitting Mesh and the others.

Overall, Polyhedral and Tetrahedral mesh configurations gave similar results in this study, while Voxel Fitted mesh configurations gave relatively different results. Since the shapes of the meshes are entirely different, this might lead to result deviations in the analysis, even if the octree parameters were completely the same.