



Aniruddha Hemantkumar Wadekar

464438

Assignment-2 Grid Generaton

School of Aerospace, Transport and Manufacturing
Computational Fluid Dynamics

MSc

Academic Year: 2024–2025

Supervisor: Dr. Tom-Robin Teschner
December 2024

© Cranfield University 2024. All rights reserved. No part of this publication may be reproduced without the written permission of the copyright owner.

Academic Integrity Declaration

I declare that:

- the thesis submitted has been written by me alone.
- the thesis submitted has not been previously submitted to this university or any other.
- that all content, including primary and/or secondary data, is true to the best of my knowledge.
- that all quotations and references have been duly acknowledged according to the requirements of academic research.

I understand that knowingly submit work in violation of the above statement will be considered by examiners as academic misconduct.

Abstract

This study evaluates the meshing capabilities of two commercial grid generation software tools—ANSYS Fluent Meshing and Cadence Fidelity Pointwise—in generating high-quality computational grids for realistic automotive models, specifically the DrivAer-Fastback configuration. The tools are compared based on several parameters, including mesh quality, density, ease of operation, compatibility with third-party tools, user experience, and their ability to accurately capture intricate geometric details and flow features. The study begins with literature review for understanding RANS model, y^+ considerations and domain sizing required to capture necessary flow conditions. CAD cleanup of the geometry is performed in ANSYS Spaceclaim followed by meshing the clean geometry separately in Fluent and Pointwise. The grids were generated in consideration with the wall modeled approach. Mesh refinement is performed on all grids using methods like 'body of influence' in Fluent and 'sources' in Pointwise. The generated grids are compared based on qualitative and quantitative metrics, including cell skewness, aspect ratio, volume orthogonality, inflation layer quality, and the ability to resolve boundary layer flows effectively. The findings revealed the unique strengths and limitations of each software, providing a detailed assessment of their suitability for complex automotive geometries and external aerodynamics simulations.

Keywords:

Grid Generation, RANS, DrivAer-Fastback, ANSYS Fluent, Cadence Fidelity Pointwise,

Table of Contents

Academic Integrity Declaration	i
Abstract	ii
Table of Contents	iii
List of Figures	iv
List of Tables	v
Nomenclature	vi
1 Introduction	1
1.1 Assumptions and given Data	2
2 Literature Review	3
3 Methodology	5
3.1 CAD/Geometry Cleanup - ANSYS Spaceclaim	5
3.2 Calculation for Number of Inflation Layers	7
3.3 ANSYS Fluent Meshing	8
3.4 Cadence Fidelity Pointwise	9
3.5 Mesh Quality Matrices	9
3.5.1 Skewness	9
3.5.2 Aspect Ratio	9
3.5.3 Orthogonality	9
4 Results and Discussion	11
4.1 Comparison of Surface Mesh	11
4.2 Fluent Tetrahedral v/s Fluent Poly-Hexcore	12
4.3 Pointwise T-Rex v/s Pointwise Voxel	14
4.4 Complexity - Which software is easier to use and why?	15
4.5 Accuracy - Which software is producing a better grid and why?	16
4.6 Recommendations. Which software would you recommend and why?	16
5 Conclusion	18
References	19

List of Figures

3.1	Original DrivAer model provided	5
3.2	Clean Geometry	6
3.3	Geometry with Enclosure and Body of Influence	6
4.1	Fluent-Surface Mesh	11
4.2	Pointwise - surface mesh	11
4.3	Fluent Tetrahedral - Inflation layer resolution	12
4.4	Fluent Poly-hexcore - inflation layer resolution	12
4.5	Fluent Tet mesh - Skewness	13
4.6	Fluent Poly-hex - Skewness	13
4.7	Fluent Tet mesh - Orthogonality	13
4.8	Fluent Poly-hex - Orthogonality	13
4.9	Fluent Tet mesh - Aspect Ratio	14
4.10	Fluent Poly-hex - Aspect Ratio	14
4.11	Pointwise T-Rex mesh - Inflation Layers	14
4.12	Pointwise Voxel - Inflation Layers	14
4.13	Pointwise T-Rex mesh - Skewness	15
4.14	Pointwise Voxel - skewness	15
4.15	Pointwise T-Rex mesh - Orthogonality	15
4.16	Pointwise Voxel - Orthogonality	15
4.17	Pointwise T-Rex mesh - Farfield	16
4.18	Pointwise Voxel - Farfield	16

List of Tables

4.1	Surface Mesh Comparison	11
4.2	Fluent Tetrahedral v/s Poly-hexcore	12
4.3	Pointwise T-Rex v/s Voxel	14
4.4	Matrix for selecting Grid Generation Tool	17

Nomenclature

1. CFD – Computational Fluid Dynamics
2. RANS – Reynolds-averaged Navier–Stokes equations
3. ρ – Fluid Density (kg/m^3)
4. L – Vehicle Length (m)
5. H – Vehicle Height (m)
6. W – Vehicle Width (m)
7. μ – Dynamic Viscosity ($Pa.s$)
8. ν – Kinematic Viscosity m^2/s
9. R_e – Reynolds number
10. $\delta(x)_{turb}$ – Boundary layer height
11. u_* – friction velocity
12. U_∞ – Free stream velocity
13. C_f – Skin Friction Coefficient
14. τ_w – Wall Shear Stress
15. y – Height of first cell
16. y^+ – Dimensionless co-ordinate
17. N – Number of Inflation Layers
18. G – Growth rate
19. s_e – Element Skewness
20. a_e – Element Aspect Ratio
21. o_e – Element Orthogonality
22. tet mesh – Tetrahedral volume mesh
23. poly-hex – Poly-Hexcore volume mesh

Chapter 1

Introduction

In the modern era, the research in automotive aerodynamics can be summarised by a three-word quote: "need for speed." This singular demand has driven the ever-increasing need for innovations in vehicle design, particularly in reducing drag, optimising flow characteristics, and improving overall performance. As vehicles become more advanced, so too must the tools used to analyse and optimise their aerodynamic properties. Computational Fluid Dynamics (CFD) plays a pivotal role in this evolution, allowing for detailed simulations of airflow over complex geometries and facilitating the design of more efficient and aerodynamic vehicles (refeences)

Historically, models such as the "Ahmed body" or the "SAE body" were used in CFD simulation to study flow separation and drag(refer tum website). However, these do not effectively capture all the flow characteristics. In 2011 the "DrivAer model" was introduced. The DrivAer model's modular structure allows for the simulation of various vehicle components, including tyres, underbody configurations, and rear-end designs, enabling researchers to understand and optimise flow features critical to vehicle performance (reference).

CFD simulations, particularly those based on Reynolds-Averaged Navier-Stokes (RANS) equations for subsonic flows, are crucial in modern vehicle aerodynamics. Key parameters such as y^+ , inflation layers, and fluid domain size must be carefully considered to ensure accurate boundary layer resolution and reliable predictions of aerodynamic forces. High-quality meshing is critical for accurately resolving complex flow features, especially around challenging regions such as tyres and underbody flows, which are crucial for drag reduction and fuel efficiency (refer). This study focuses on evaluating the meshing capabilities of two prominent commercial CFD tools—ANSYS Fluent Meshing and Cadence Fidelity Pointwise—and compares their effectiveness in generating high-quality computational grids for automotive models like DrivAer (refer).

The methodology incorporates best practices from the literature for y^+ estimation, fluid domain sizing, and mesh quality optimisation. CAD cleanup is performed using ANSYS SpaceClaim to prepare the geometry for meshing, followed by grid generation in Fluent and Pointwise using consistent parameters. Refinement techniques, such as body of influence in Fluent and source-based methods in Pointwise, are applied to improve resolution in critical flow regions. The generated grids are then evaluated using common quality metrics such as skewness, aspect ratio, and orthogonality, to assess their effectiveness in capturing flow structures and resolving the boundary layer accurately.

By comparing the strengths and limitations of each meshing tool, this article aims to selecting the optimal tool for automotive aerodynamics simulations, ultimately contributing to the precision and reliability of CFD predictions in vehicle design.

1.1 Assumptions and given Data

For the purpose of this exercise, some assumptions about the fluid flow have been made. They are listed as follows

1. The external fluid domain is considered to at Standard Temperature and Pressure(STP)
2. The processes are carried out assuming that ANSYS Fluent will be used as a solver.
3. RANS model will be used for subsonic flows
4. $R_e = 4640000$
5. $\mu = 1.8110^{-5} \text{ kg/ms}$
6. $\rho = 1.204 \text{ kg/m}^3$
7. $L = 4.61265 \text{ m}$
8. $H = 1.41404 \text{ m}$
9. $W = 2.01744 \text{ m}$

Chapter 2

Literature Review

The development of aerodynamic models for automotive applications, particularly the DrivAer model, has been a key area of research in computational fluid dynamics (CFD). The DrivAer model, representing a standard reference for vehicle aerodynamics, has been extensively studied to understand the impact of mesh resolution, turbulence models, and grid refinement on drag predictions and flow characteristics. Various studies, including (14), have demonstrated the critical role of grid refinement in predicting accurate drag values. It was found that higher mesh resolution improves drag accuracy, particularly when turbulence models are carefully selected. Similarly, (5) revealed that hybrid simulations benefit significantly from high-quality meshes, which improve boundary layer resolution and lead to more reliable aerodynamic predictions.

Mesh generation, a key aspect of CFD, balances computational efficiency with the need for technical precision. As (6) explains, while modern tools automate much of the process, achieving optimal results still relies on expert judgment to tailor meshes for specific geometries and flow regimes. This observation aligns with findings from (8), which emphasized optimizing mesh quality to balance accuracy and computational expense. It was shown that careful mesh refinement in critical flow regions significantly enhances the accuracy of flow prediction fidelity while maintaining computational feasibility.

The broader implications of mesh schemes and turbulence models are discussed in (11). Although focused on stirred tank flows, the paper provides valuable insights into the influence of meshing strategies and turbulence models on prediction accuracy. It highlights the universal importance of selecting appropriate mesh configurations to resolve complex flow structures effectively, principles that are directly applicable to external aerodynamics simulations of the DrivAer model.

The use of Reynolds-Averaged Navier-Stokes (RANS) models for automotive aerodynamics is well-established, as demonstrated in (4). This paper evaluates the suitability of RANS models for external aerodynamics, particularly for configurations like the DrivAer model, and outlines the influence of boundary layer modeling on prediction accuracy. RANS models, though widely used, are sensitive to mesh resolution and require careful attention to wall y^+ values, directly influencing boundary layer resolution and flow predictions.

The importance of tire geometry in automotive aerodynamics was explored in (15). The study revealed how tire attributes significantly affect aerodynamic drag and wake structures, underscoring the need for accurate modeling in CFD simulations. Similarly,

(2) showed that considering wheel-body interactions and adopting structured meshes improves prediction fidelity.

The study (17) conducted an in-depth analysis of different DrivAer configurations, focusing on the evolution of aerodynamic wakes. It highlighted the critical role of turbulence modelling and mesh resolution in accurately capturing far-field aerodynamic characteristics.

These studies collectively stress that mesh generation, though increasingly automated, remains an intricate task where human insight complements technological advancements. Achieving accurate, cost-effective simulations for automotive aerodynamics requires careful selection of meshing strategies, numerical parameters, and turbulence models, with the DrivAer model serving as a valuable benchmark for evaluating CFD methodologies.

Chapter 3

Methodology

3.1 CAD/Geometry Cleanup - ANSYS Spaceclaim

The DrivAer - Fastback model provided is as shown in the figure 3.1. It is composed of open surfaces, solids, inexact edges and even missing faces. This can be termed as a dirty geometry. ANSYS Spaceclaim was used to obtain an optimally clean geometry that would be easy to mesh.

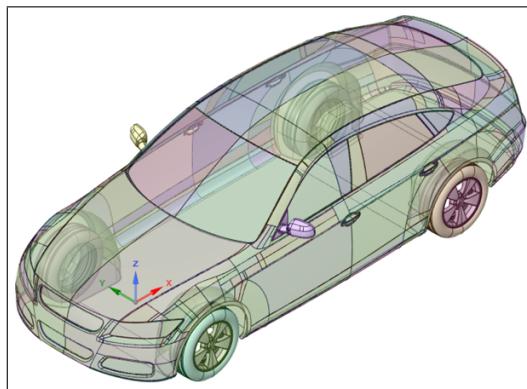


Figure 3.1: Original DrivAer model provided

Step-1 The obtained model was first saved in the native '.scdoc' format.

Step-2 Within the model, all surfaces were first moved to one component and then stitched using the 'Stitch' Command.

Step-3 The 'Missing Faces' command was used to search for missing surfaces. Once created, a solid geometry obtained. This was done because it was easier to work with the solid geometry rather than the surfaces

Step-4 Post solidification, the mirror bodies present were moved in to the solid chassis body and then merged together using boolean commands.

Step-5 On the chassis, with the help of the 'Merge Faces' command, different surfaces were strategically merged so as to create a geometry that would be easy to mesh.

Care was taken to ensure that the geometry was maintained as it is. This step is done to ensure that irregular/small faces that might cause issues in meshing are merged into larger ones in order to obtain relatively clean grids.

Step-6 The treads on the tyres were deleted. Additionally, the tyres were cut from the bottom at a height of 3 mm so as to simulate the contact between the ground and the tyre(similar method was adopted in (15)). The wheel rim was converted to a complete cylindrical block and then merged with the tyre to obtain one single wheel geometry.

Step-7 The fluid domain is created using the 'Enclosure' command. The dimensions of the enclosure/fluid domain are derived from (8)

Step-8 As the overall body is symmetric, it is assumed that the flow characteristics remain symmetric about the X-Y plane. Hence in order to reduce computational time and complexity, the domain is split about the X-Y Plane and the LHS side is carried forward in the process.

Step-9 Bodies of influences are added as required.

Step-10 Using named selections, define the important boundaries like inlet and outlet along with the other surface entities.

Step-11 Finally the geometry was not exported to the Fluent meshing.

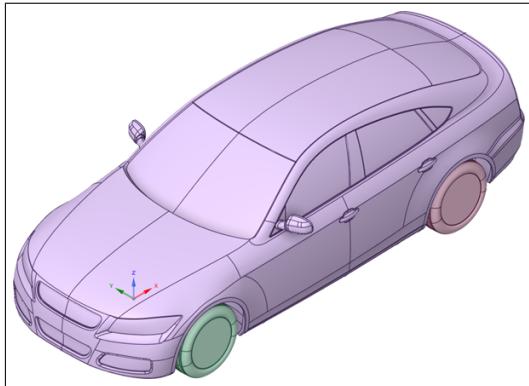


Figure 3.2: Clean Geometry

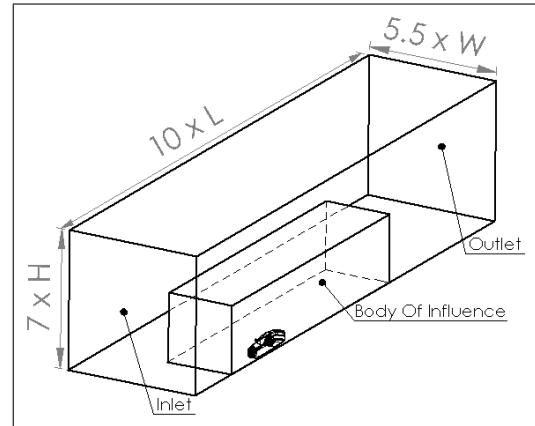


Figure 3.3: Geometry with Enclosure and Body of Influence

Note that for Pointwise, the fluid domain was not exported. Only the vehicle(Half) geometry was exported in a '.STEP' format

3.2 Calculation for Number of Inflation Layers

The RANS model requires good and accurate modeling of the boundary layer to precisely simulate the flow features. The boundary layer thickness varies with the flow conditions that is the Reynolds Number (R_e), distance from leading edge,

The total height of the Turbulence layer can be determined by

$$\delta(x)_{turb} = \frac{0.37 * L_e}{R_e^{0.2}} \quad (3.1)$$

For the current process, The $R_e = 4640000$, $L_e = L = 4.61255$ m

Thus The total height of the turbulence layer is

$$\delta(x)_{turb} = 0.079222516m \quad (3.2)$$

The free stream velocity U_∞ can be obtained fro the equation of the Reynolds number (9)(12)

$$R_e = \frac{U_\infty L}{v} \quad (3.3)$$

Therefore the free stream velocity of the flow

$$U_\infty = 15.2476832m/s \quad (3.4)$$

The approximate skin friction coefficient is calculated as

$$C_f = \frac{0.026}{R_e^{1/7}} \quad (3.5)$$

$$C_f = 0.00290144 \quad (3.6)$$

The approximate wall shear stress is given by

$$\tau_w = 0.5C_f R_e U_\infty^2 \quad (3.7)$$

$$\tau_w = 0.406085842Pa \quad (3.8)$$

Similarly the approximate friction velocity

$$u_* = \sqrt{\tau_w / \rho} \quad (3.9)$$

$$u_* = 0.580758641m/s \quad (3.10)$$

The y^+ is a non dimensional co-ordinate whereas the distance normal to the surface is y typically, the typical y^+ value is between 30 to 70((10), (13) and (13)). for the scope of this task, the y^+ is assumed as 30. This will imply more inflation layers within the the boundary layer. Thus it is expeced to give godd resolution of the flow in the boundary. Hence,

$$y^+ = 30 \quad (3.11)$$

The height of the first cell within the boundary layer(adjacent to the wall) can be calculated is

$$y = \Delta y = \frac{y^+ v}{u^*} \quad (3.12)$$

$$y = 0.000783m \quad (3.13)$$

In order to correctly capture the boundary layer, the subsequent cell heights are increased. This is done with the help of a 'Growth Rate' G and number of inflation layers N . In general, the subsequent cell heights can be considered as a geometric progression until the summation of all cell heights is equal to the height of the boundary layer. Thus, the relationship between the initial cell height y , growth rate G , number of inflation layers N and the total height of the boundary layer $\delta(x)_{turb}$ is given by (7), (9), (12)

$$\delta(x)_{turb} = y \frac{1 - G^N}{1 - G} \quad (3.14)$$

In most commercial CFD tools, the Growth rate is preset to 1.2, Hence keeping that constant, and solving the eq. 3.14 for N

$$N = 16.75984098 \quad (3.15)$$

$$N \approx 17 \quad (3.16)$$

3.3 ANSYS Fluent Meshing

The clean geometry along with the fluid domain and body of influence can be directly imported from Spaceclaim. The process and the structure/tree are straightforward and well defined. It employs the waterfall approach which is easy to understand and use.

Ansys Fluent Meshing employs a top down approach for meshing. This implies that, the outermost surfaces will be meshed first and progressively the lower surface boundaries. Hence Local 'Face sizing' was applied to the farfield and symmetry surfaces. The body of influence is marked as 'Body of Influence'. This will help define the mesh near the vehicle surface and the regions of interest where wake of the flow might be observed. The remaining surfaces namely the vehicle chassis and the wheels are assigned the 'Curvature' sizing. The 'Curvature' sizing function examines the curvatures on edges and faces to control the element size. Thus it is useful to create a refined mesh near the curved geometries. Finally the surface mesh was created and was passed through multiple quality metrics.

The default volume meshing in Ansys Fluent uses 'Unstructured Mesh Type'. Although efficient, it is not the best approach. A 'Hybrid Mesh' type can be obtained by applying mesh refinements in areas of interests. This approach is particularly effective in balancing computational efficiency and accuracy. The Hybrid volume mesh is obtained by adding advanced features like inflation layers near areas of interest. Under the 'Add Boundary Layer' sections values for first cell height (eq.3.13), Number of Inflation layers(3.16). The default Growth rate remains unchanged. As the inflation layers are required near the vehicle surfaces, the 'Walls' are selected as the regions.

Lastly two different mesh were obtained. a Tetrahedral mesh and a Poly-Hexcore mesh. Note that both the meshes were scrutinized for mesh quality.

3.4 Cadence Fidelity Pointwise

Pointwise employs bottom up approach for meshing. It starts descritizing the smallest entities first and then moves to outer layers. This is the very reason as to why Only the Vehicle cut section and the wheel bodies were imported as opposed to ANSYS Fluent where the fluid domain and bodies of influence were imported at the same time. Structured, Unstructured and Hybrid mesh can be generated in Pointwise. For the scope of this article, Hybrid meshing stratergy is adopted. The imported geometry although clean had to be modified so as to make it easier for Pointwise algorithm to mesh. The 'Assemble Quilts' command was used for he same. Note that the 'Auto Surface Mesh' command was not used to generate the surface mesh. This is because it resulted to fatal errors like system crash or at times loss of geometry. As a result a roundabout way was used to generate the surface mesh. Here Pointwise gives an option to choose between Structured or Unstructured mesh. However as the current geometry is irregular and has many curves the unstructured isotropic approach with all 'Triangle' elements was selected. care was taken to ensure that the sizing parameters were kept the same as in Ansys fluent. Quality checks were perfomed to ensure that the overall mesh quality was acceptable.

To Generate the Volume mesh, the 'Automatic Volume mesh' was used. This toll can create the fluid domain as required and subsequently mesh it. Noote that The complete Volume mesh was not generated at this stage. Instead 'Source' objects which are equivalent to 'Bodies of Influences' are added and then meshed to obtain the final volume mesh. Note that 2 meshes were generated 1st a Tetrahedral Mesh with T-Rex refinement and 2nd using the 'Voxel' approach

Note that Care was taken to ensure that all meshes are conformal nature. also, the CAE Solver type is changed to ANSYS Fluent before starting the meshing process.

3.5 Mesh Quality Matrices

3.5.1 Skewness

Skewness is the default mesh quality criterion in ANSYS Fluent. The range of skewness is between $0 < s_e < 1$. It gives an idea as to how close the element is to the ideal element. In ANSYS Fluent, the good element should have a skewness close to '0'. For Poinwise, the scale is reversed, that is the ideal element skweness should be close to '1'(1)

3.5.2 Aspect Ratio

It refers to the ratio of the maximum edge to the minimum edge of an element. For example, an ideal quad element will have an aspect ratio of 1 (square shape) the rage is in between $1 < a_e < \infty$

3.5.3 Orthogonality

Orthogonality takes into account the neighbouring elements to decide on the qialty. This can be usefull in determinning the overall mesh quality of the grid. For a pair of elements,

orthgonality is the angle formed by the normal of the face that these cells share and the vector that connects two nearby cell centers. It varies between the range of $0 < o_e < 1$

Chapter 4

Results and Discussion

4.1 Comparison of Surface Mesh

The Surface Mesh produced in both the tools is satisfactory. In Ansys, the farfield and the vehicle surfaces are meshed at the same time where as in Pointwise only the car surface and wheels are meshed. Time taken is less in ANSYS compared to that in Pointwise.

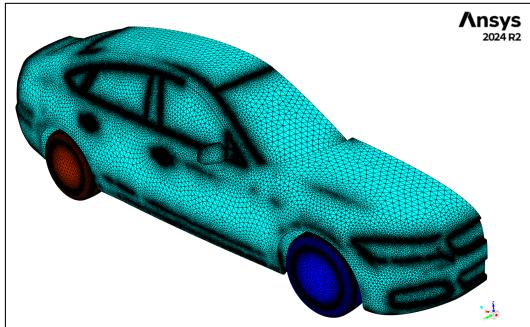


Figure 4.1: Fluent-Surface Mesh

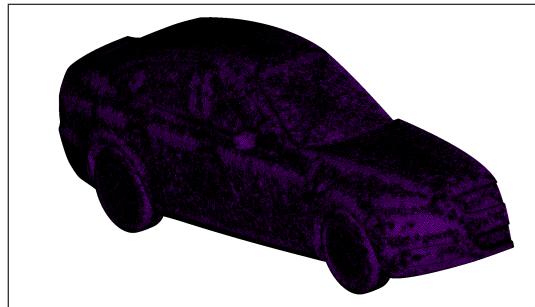


Figure 4.2: Pointwise - surface mesh

Table 4.1: Surface Mesh Comparison

Surface mesh Matrices	Ansys Fluent Meshing	Cadence Fidelity Pointwise
Cell Type	Triangles	Triangles
Total Cells	379262	368021
Time	3.25 mins	12 mins
Cells with Worst skewness (%)	0.105731658	0.029889599
Cells with worst Aspect Ratio (%)	0.117596806	0.000271724

The elements with bad skewness and aspect ratio are observed in the bottom portion of the geometry in both cases. Specifically along the symmetry boundary plane. These elements can cause delays in terms of computing time and volume mesh generation. This can be solved by locally adding proximity sizing near the edges.

4.2 Fluent Tetrahedral v/s Fluent Poly-Hexcore

The tet mesh has over 10 million cells where as the poly-hex has less than 4.5 million which is less than half.

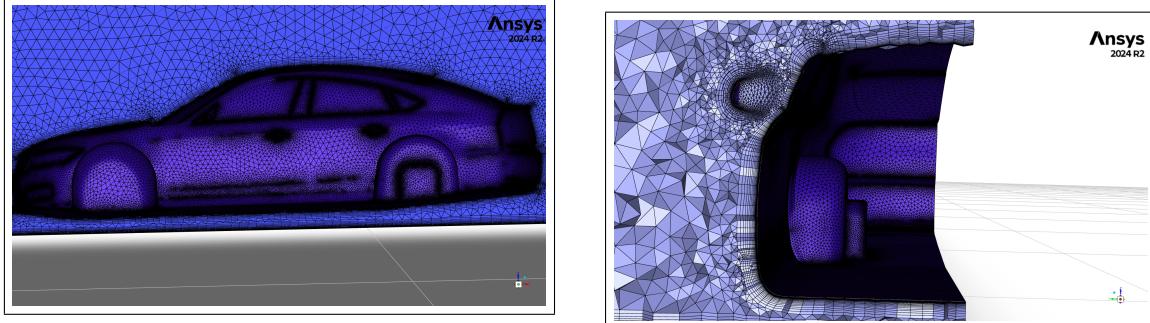


Figure 4.3: Fluent Tetrahedral - Inflation layer resolution

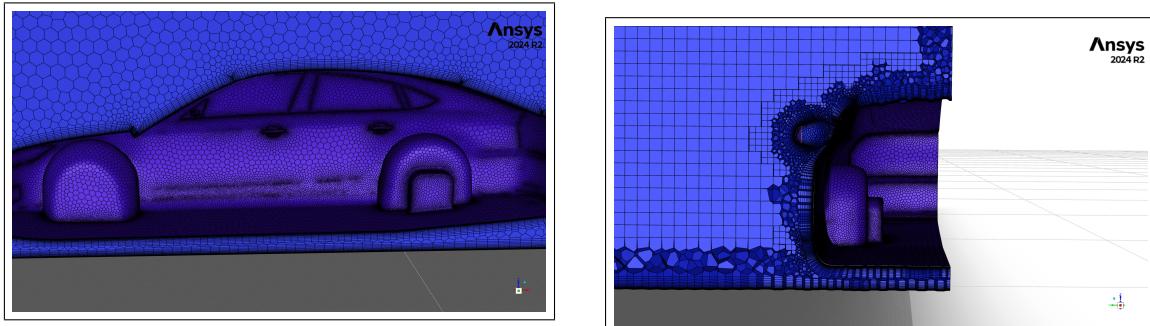


Figure 4.4: Fluent Poly-hexcore - inflation layer resolution

As seen in the figures above it is observed that the inflation layers are well defined in the poly-hex mes as compared to the tet mesh. The reason is that the poly-hex mesh employees 'peel layers' between the inflation layers and the rest of the volume mesh. This creates a relatively smooth transition from. As a result the overall quality of the volume mesh improves drastically.

Table 4.2: Fluent Tetrahedral v/s Poly-hexcore

Parameter	Tetrahedral	Poly-Hexcore
Cell Type	Tetrahedrons, prisms, quads	hexahedral, quad
Total Cells	10365043	4413017
Time (mins)	5	3.48
Worst skewness cells(%)	0.00014	0.00033
Worst orthogonal cells(%)	0.00014	0.01495
Worst Aspect Ratio cells(%)	0.00041	0.00022

The table4.2 shows the summary for the two meshes. The time taken to generate the poly-hex core mesh is relatively less as compared to the tet mesh. This can be attributed to the overall number of cells.

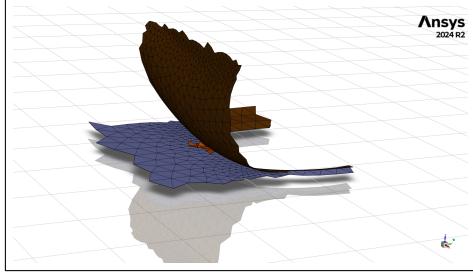


Figure 4.5: Fluent Tet mesh - Skewness

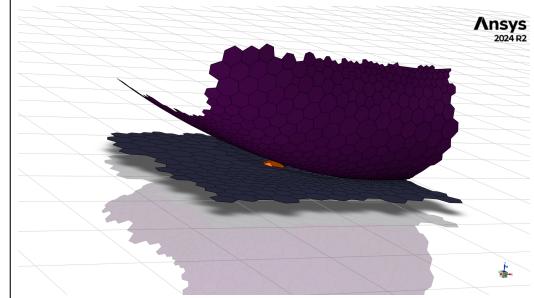


Figure 4.6: Fluent Poly-hex - Skewness

As observed in the figures 4.5, 4.6, the worst quality elements(skewness) are found at the nearly the same location. These elements are ones close to the region where the vehicle tyre meets the bottom boundary(road). Here the geometry becomes almost flat thus resulting in elements with high skewness. These elements can cause bad flux(errors) to be induced in the model thus reducing overall accuracy. This can be overcome by adding a vertical step in the model. A similar observation can be made if orthogonality is used as a quality metric. For aspect ratio, varying observations can be made. The

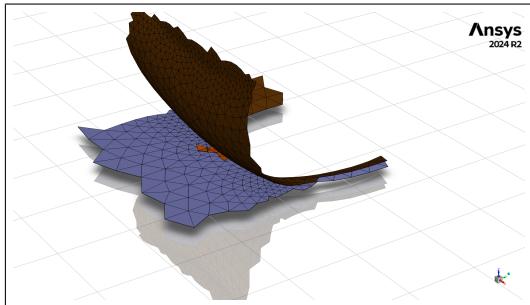


Figure 4.7: Fluent Tet mesh - Orthogonality

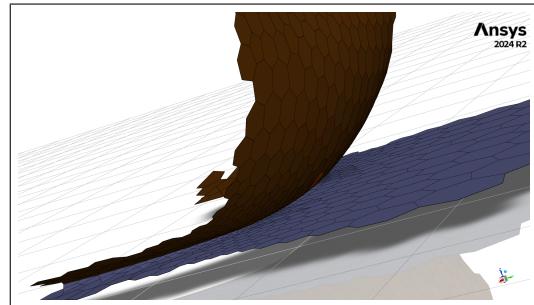


Figure 4.8: Fluent Poly-hex - Orthogonality

worst aspect ration for the tet mesh is observed near the door handles. Here the geometry changes abruptly from convex to concave. Thus the cell quality is reduced. This observation exactly matches with the description provided in (18). Boundary layer separation is expected to occur in this region but due to the bad aspect ratio elements there might be come irregularities in the results On the other hand, in case of poly-hexcore, the elements with the worst aspect ratio are observed far away fro the vehicle and infact in front of the geometry. Thus these might not have adverse effects on the CFD results.

Overall, it was observed that the poly-hex mesh was relatively well structured. It has fewer number of elements but is capable of capturing all minute geometric details without issues. As per a white paper published by ANSYS Inc (3) the poly-hexcore(also called as mosaic mesh method) is an effective in cutting down time and cost of the simulation with little to no effect on the result accuracy. This is supported by arguments in (8). Hence it can be concluded that the Poly-Hexcore mesh should be preferred over the tetrahedral mesh in ANSYS Fluent.

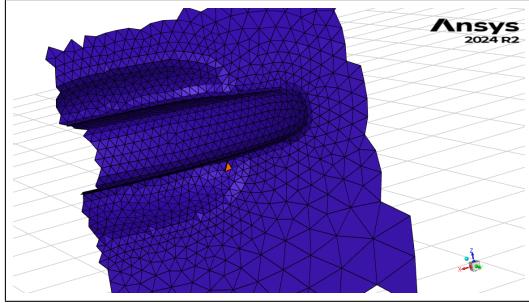


Figure 4.9: Fluent Tet mesh - Aspect Ratio

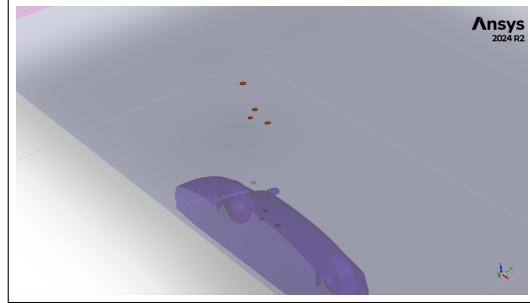


Figure 4.10: Fluent Poly-hex - Aspect Ratio

4.3 Pointwise T-Rex v/s Pointwise Voxel

Table 4.3: Pointwise T-Rex v/s Voxel
Tetrahedral

Parameter	Tetrahedral	Poly-Hexcore
Cell Type	Tetrahedrons, prisms, quads	tetrahedrons, prisms, quads
Total Cells	13514184	37140891
Time (mins)	5	53
Worst skewness cells (%)	0.06439	0.00431
Worst orthogonal cells (%)	0.02585	0.02157
Worst Aspect Ratio cells(%)	7.3E-06	2.6E-06

In Pointwise, both T-Rex and Voxel are used to produce hybrid grids. As observed from the figures 4.11, 4.12, the Voxel mesh looks more refined as compared to the T-rex mesh. However both the approaches fail to properly capture the inflation layers around small curvatures in the front section of the vehicle. As seen in the image, the primary

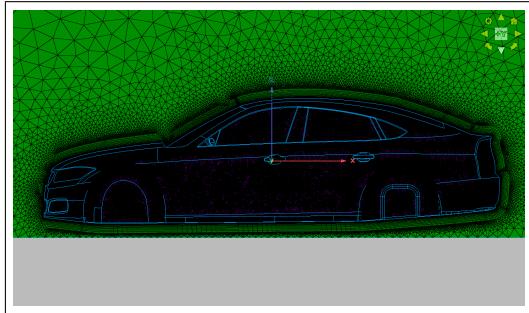


Figure 4.11: Pointwise T-Rex mesh - Inflation Layers

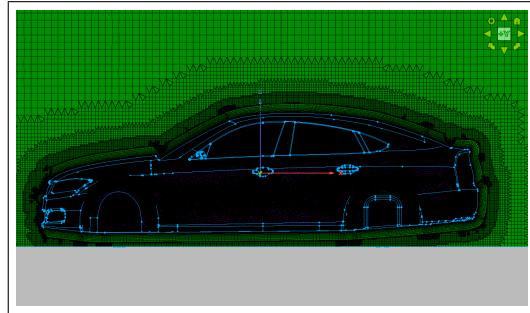


Figure 4.12: Pointwise Voxel - Inflation Layers

element in T-Rex is a tetrahedral, however in voxel, the quad element appears to be dominant. The voxel block typically contains aligned cartesian grids. This might be because of the way the voxel mesh is generated. It is like the dendrites of a neuron or the root of a plant, it will keep branching into smaller and smaller sections till it reaches the end. This enables the software to produce extremely dense and structured mesh. Overall the voxel has the best element quality of all the meshes generated so far. However, the number of

elements (above 37 million) and the overall size of the mesh are too high as compared to its peers. This can give rise to long computation times and the overall simulation cost might increase. In terms of mesh quality, there are barely elements with bad aspect ratio in both meshes. However, elements with bad skewness and orthogonality were observed near the mirror (figures 4.13, 4.14, 4.15, 4.16). These can cause the flow field to diverge and might not capture the boundary layer separation that is expected after the mirror. Note that it is difficult to locate bad elements in Pointwise.

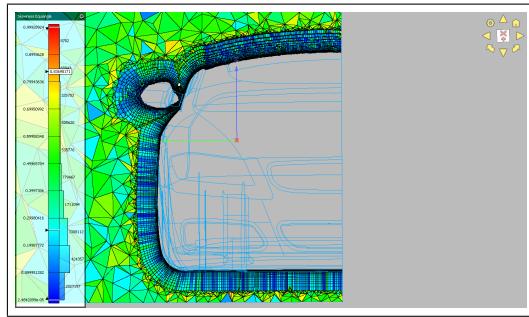


Figure 4.13: Pointwise T-Rex mesh - Skewness

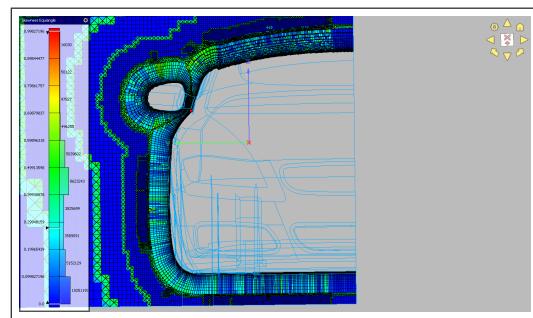


Figure 4.14: Pointwise Voxel - skewness

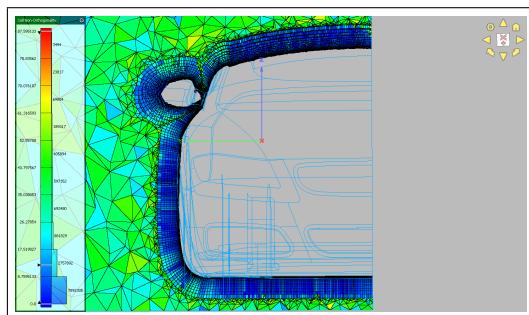


Figure 4.15: Pointwise T-Rex mesh - Orthogonality

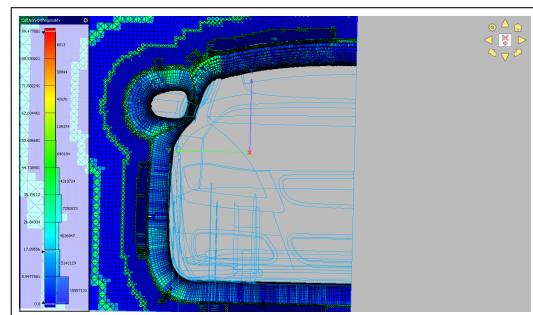


Figure 4.16: Pointwise Voxel - Orthogonality

It can be observed that the voxel mesh has a very uniform mesh quality. The layered structure enables it to maintain good mesh resolution in the regions of interest. Another point to note here is the mesh resolution between the T-rex model and the voxel model specifically in the farfield region. Refer figure 4.17, 4.18. The element size increase drastically towards the extreme ends of the farfield in case of T-rex. This might cause divergence/dispersion of fluid flow in the wake region.

Overall the voxel mesh offers a better chance to accurately capture flow features.

4.4 Complexity - Which software is easier to use and why?

The user experience in ANSYS Fluent is far better than Cadence Fidelity Pointwise. This can be attributed to the following points.

- Workflow - Fluent follows a well defined workflow and structure which is easy to understand and use at the same time.

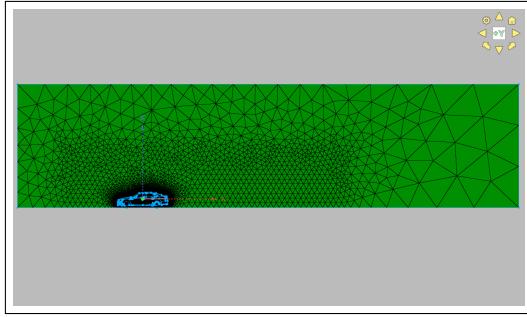


Figure 4.17: Pointwise T-Rex mesh - Farfield

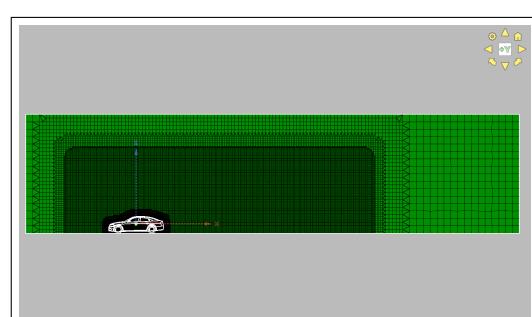


Figure 4.18: Pointwise Voxel - Farfield

- User Interface - All tools are easily accessible in fluent than in pointwise. User does not need to search for commands to execute.
- Unit System - Pointwise does not have a proper unit system. Hence it is confusing to modify parameters or values.
- Descriptions/help - Hovering over a command shows a detail use case in Ansys Fluent. This makes it much more easier to use.
- Cross Platform Compatibility - Cross platform compatibility for CAD and other files is better in Fluent. It is not good in Pointwise. for example, when importing a cleaned geometry from Spaceclaim, random bizarre curves are produced with gaps and irregularities to original surfaces. This adds to time required to re-clean the geometry.
- Fatal Error/crash - Pointwise tends to crash more than Fluent(6 such instances were recorded during this task)

4.5 Accuracy - Which software is producing a better grid and why?

Accuracy and quality of the grids in both the tools is equally good. In Pointwise, the Voxel approach yields a mesh closest to a structured mesh. However the overall size and the time required for computation are very high .On the other hand, the poly-hexcore mesh produced in Fluent demonstrated good mesh quality and can be used for quick analysis without any loss in accuracy.

4.6 Recommendations. Which software would you recommend and why?

Pointwise and Fluent both have their merits and demerits. Both the tools are capable of producing good quality meshes for complex automotive geometries. The choice of the

Table 4.4: Matrix for selecting Grid Generation Tool

Parameter	ANSYS Fluent Meshing	Cadence Fidelity Pointwise
User Interface	Good	Acceptable
Workflow/Structure	Well Defined	Complicated
Unit System	Well Defined	Not defined
Grid Types supported	Unstructured , Hybrid	Structured, Unstructured
Mesh Quality	Good	Better
Control over mesh	Not much	Very good
CrossPlatform Compatibility	Good	Bad
Output file storage size	medium	Very High
fatal error/crash	0	6
Catered towards	Novice and expert users	expert user

tool is subjective and based on requirement. Table 4.4 provides a chart for deciding the meshing tool.

Pointwise provides far more control over the mesh than Fluent. However it is marred by the unintuitive and complex process. If user needs a structured mesh, then Pointwise should be the go to option. Incase a mesh with less computation time but fairly accurate results is required, poly-hexcore mesh in Fluent should be used.

For a novice user, Ansys Fluent is like a drawing book with a stencil printed on it. With less time, fairly good quality mesh can be produced. On the other hand Pointwise could be termed as an open canvas where an experienced maestro can create the most detailed grids for an exhaustive simulation.

Chapter 5

Conclusion

This report discussed the process of generating mesh for complex automotive geometries like the DrivAer - Fastback model using 2 different commercial Grid generation tools namely Ansys Fluent Meshing and Cadence Fidelity Pointwise. The obtained meshes were scrutinized with the help of mesh quality parameters like skewness, orthogonality and aspect ratio. The poly-hexcore mesh is far superior than the standard unstructured mesh. It requires less computational time and hardware and produces fairly accurate results. Incase of a detailed mesh requirement, the voxel mesh generated using pointwise is recommended. For a novice user, Ansys Fluent Meshing is a good starting point, but Pointwise is capable of producing better quality mesh if the user is an expert in the tool.

References

1. Determining mesh quality and accuracy parameters. <https://resources.system-analysis.cadence.com/blog/msa2022-determining-mesh-quality-and-accuracy-parameters>, n.d.
2. N. E. Ahmad, E. F. Abo-Serie, and A. Gaylard. Mesh optimization for ground vehicle aerodynamics. *CFD Letters*, 2(1), 2010. URL https://www.researchgate.net/publication/330383629_Mesh_Optimization_for_Ground_Vehicle_Aerodynamics.
3. Ansys. Ansys Mosaic Technology Combines Disparate Meshes. <https://www.ansys.com/en-gb/resource-center/white-paper/ansys-fluent-mosaic-technology>, 2021. Accessed: 2024-12-14.
4. N. Ashton, A. West, S. Lardeau, and A. Revell. Assessment of rans and des methods for realistic automotive models. *Computers & Fluids*, 128:1–15, 2016. doi: 10.1016/j.compfluid.2016.01.008.
5. N. Ashton, P. Unterlechner, and T. Blacha. Assessing the sensitivity of hybrid rans-les simulations to mesh resolution, numerical schemes and turbulence modelling within an industrial cfd process. *SAE Technical Paper Series*, 2018. doi: 10.4271/2018-01-0709.
6. T. J. Baker. Mesh generation: Art or science? *Progress in Aerospace Sciences*, 41 (1):29–63, 2005. doi: 10.1016/j.paerosci.2005.02.002.
7. [CFD]. Inflation layers / prism layers in cfd. <https://www.youtube.com/watch?v=1gSHN99I7L4>, 2021. Retrieved December 11, 2021.
8. G. Felekos, E. Douvi, and D. Margaris. Cost-effective numerical analysis of the drivaer fastback model aerodynamics. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 2022. doi: 10.1177/09544070221135908.
9. Fluid Mechanics 101. [cfd] what is the difference between $y+$ and y^* ? https://www.youtube.com/watch?v=nSdVaF3JnI0&list=PLnJ8lIgfDbkrhQTjGZ_22z7NoktJTrTF1&index=6, 2019.
10. A. I. Heft, T. Indinger, and N. A. Adams. Experimental and numerical investigation of the drivaer model. *Volume 1: Symposia, Parts a and B*, 2012. doi: 10.1115/fedsm2012-72272.

11. M. Karimi, Guven Akdogan, and S. M. Bradshaw. Effects of different mesh schemes and turbulence models in cfd modelling of stirred tanks. *Physicochemical Problems of Mineral Processing*, 48(2):513–531, 2012.
12. B. R. Munson, D. F. Young, and T. H. Okiishi. *Fundamentals Of Fluid Mechanics*. John Wiley & Sons, 2007.
13. P. Qin, A. Ricci, and B. Blocken. Cfd simulation of aerodynamic forces on the draivaer car model: Impact of computational parameters. *Journal of Wind Engineering and Industrial Aerodynamics*, 248:105711–105711, 2024. doi: 10.1016/j.jweia.2024.105711.
14. Mats Ramnefors, Rikard Bensryd, E. Holmberg, and Sven Perzon. Accuracy of drag predictions on cars using cfd - effect of grid refinement and turbulence models. *SAE Technical Paper Series*, 1996. doi: 10.4271/960681.
15. S. Rath and A. Untaroiu. Effects of tire attributes on the aerodynamic performance of a generic car-tire assembly. *Journal of Fluids Engineering*, 147(1), 2024. doi: 10.1115/1.4066192.
16. P. M. Rousseau, Azzeddine Soulaimani, and M. Sabourin. Comparison between structured hexahedral and hybrid tetrahedral meshes generated by commercial software for cfd hydraulic turbine analysis. 2013.
17. Renan Francisco Soares, K. P. Garry, and J. Holt. Comparison of the far-field aerodynamic wake development for three draivaer model configurations using a cost-effective rans simulation. 2017. doi: 10.4271/2017-01-1514.
18. Tom Robin Teschner. *Lecture PPT's from the course of Grid Generation - CAD Lecture Notes (Static)*.
19. F. M. White. *Fluid Mechanics (8th ed.)*. McGraw-Hill Education, 2016.
20. C. Zhang, C. P. Bounds, L. Foster, and M. Uddin. Turbulence modeling effects on the cfd predictions of flow over a detailed full-scale sedan vehicle. *Fluids*, 4(3):148, 2019. doi: 10.3390/fluids4030148.