



MARMARA UNIVERSITY
FACULTY OF ENGINEERING



INVESTIGATION OF GRID GENERATION STRATEGIES FOR THE ANALYSIS OF INTAKE MANIFOLDS

Arda Çağlar Pulat, Alaattin Kerem Soğancı

GRADUATION PROJECT REPORT

Department of Mechanical Engineering

Supervisor

Prof. Dr. Mehmet Zafer GÜL

ISTANBUL, 2020



MARMARA UNIVERSITY
FACULTY OF ENGINEERING



Investigation of Grid Generation Strategies for the Analysis of Intake Manifolds

by

Arda Çağlar Pulat, Alaattin Kerem Soğancı

(150416036)

(150415024)

July 5, 2020, İstanbul

**SUBMITTED TO THE DEPARTMENT OF MECHANICAL
ENGINEERING IN PARTIAL FULFILLMENT OF THE
REQUIREMENTS FOR THE DEGREE**

OF

BACHELOR SCIENCE

AT

MARMARA UNIVERSITY

The author(s) hereby grant(s) to Marmara University permission to reproduce and to distribute publicly paper and electronic copies of this document in whole or in part and declare that the prepared document does not in anyway include copying of previous work on the subject or the use of ideas, concepts, words, or structures regarding the subject without appropriate acknowledgement of the source material.

Signature of Author(s)..... Alaattin Kerem Soğancı *A. K. Soğancı* Arda Çağlar Pulat *D. Pulat*

Department of Mechanical Engineering

Certified By.....

Project Supervisor, Department of Mechanical Engineering

Accepted By.....

Head of the Department of Mechanical Engineering

ACKNOWLEDGEMENT

First of all, I would like to thank my supervisor Prof. Dr. Mehmet Zafer Güл, for the valuable guidance and advice on preparing this thesis and giving me moral and material support.

July, 2020

Arda Çağlar Pulat, Alaattin Kerem Soğancı

CONTENTS

ACKNOWLEDGEMENT	ii
ABSTRACT	vi
ABBREVIATIONS	vii
LIST OF FIGURES	viii
LIST OF TABLES	xi
1. INTRODUCTION.....	1
1.1. Aim of the Research	1
1.2. Literature Survey	2
1.2.1. Introduction	2
1.2.2. Methods for grid generation for CFD analyses	2
1.2.3. Conclusion.....	4
2. MESH TYPES AND QUALITY	4
2.1. Two-Dimensional Mesh Types	4
2.2. Three-Dimensional Mesh Types	4
2.3. Classification of Grids	5
2.3.1. Structured grids.....	5
2.3.2. Unstructured grids.....	5
2.3.3. Hybrid grids	6
2.4. Choosing the correct mesh	7
2.4.1. Setup time.....	8
2.4.2. Computational expense	8

2.4.3.	Numerical diffusion	8
2.5.	Decision Process.....	8
2.6.	Mesh Quality	9
3.	CAD EXCHANGE DATA FORMATS.....	10
3.1.	Neutral Cad File Format	11
3.1.1.	STL - Stereolithography or “Standard Tessellation Language”	11
3.1.2.	STP - ISO 10303 or STEP.....	12
3.1.3.	IGES - Initial graphics exchange specification	12
3.2.	Native CAD File Format.....	13
3.2.1.	DXF and DWG (AutoCAD).....	13
4.	REPAIRING PROCESS	15
4.1.	ANSYS SpaceClaim Repair Section Tools	15
4.1.1.	Solidify section	16
4.1.2.	Fix section.....	16
4.1.3.	Fix curves section	16
4.1.4.	Adjust section	17
5.	IMPORTING CAD FILE AND REPAIRING SECTION	17
6.	CFD SOFTWARE	20
6.1.	Open-Source CFD Software:	20
6.1.1.	Pre-Processing	21
6.1.2.	Mesh generation.....	21
6.1.2.1.	BlockMesh.....	21

6.1.2.2.	SnappyHexMesh	21
6.1.3.	Mesh conversion:	21
6.1.4.	Solver:	22
6.1.5.	Post-Processing	22
6.2.	Gmsh.....	22
6.3.	SALOME	23
6.4.	ANSYS Fluent.....	23
6.4.1.	Modelling turbulence in ANSYS Fluent.....	24
6.4.2.	Solvers in ANSYS Fluent.....	25
6.4.3.	Segregated algorithms in ANSYS Fluent.....	26
7.	GRID GENERATION IN ANSYS FLUENT	26
7.1.	Fully Automated Unstructured Tetra Mesh	27
7.2.	Manually Improved Unstructured Tetra Mesh	29
8.	RESULTS AND DISCUSSION	44
8.1.	Results for Fully Automated Unstructured Tetra Mesh:	44
8.4.	Results for Improved Unstructured Pave Grid	50
8.5.	Comparing Open-Source and Proprietary Software.....	52
9.	CONCLUSION.....	53
10.	REFERENCES:	55

ABSTRACT

The investigation of grid generation strategies for the Computational Fluid Dynamics (CFD) analysis of intake manifolds is discussed in this paper. Mesh structures and different mesh generation strategies are investigated in detail. Geometry formats and problems in the Computer Aided Design (CAD) files are discussed. Element quality of different grid types are inspected. The commercial computer program ANSYS Fluent is used for obtaining results. The used geometry is an intake manifold designed by BMC. Different mesh structures were employed in the flow domain. Obtained results are compared in the scope of computational expense, time cost and accuracy. Turbulent air flow inside the intake manifold is simulated with the use of k- ϵ model and pressure-based Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm of ANSYS Fluent. Results showed that the accuracy of the results are highly dependent on the mesh structure and quality of the mesh elements. Best element quality was achieved with Butterfly grid, however, this grid was not applicable to the intake manifold because of complexity of the geometry. Boundary layers are tested with unstructured tetrahedral meshes. Results were improved with the improvement of mesh. Best results are obtained with a hybrid grid, where hexahedral elements are dominant in number. Higher element density in hexahedral dominant mesh resulted in a greater time to build the mesh and a greater time to obtain a solution, but a better solution. Results also showed that very fast convergence can be achieved with relatively coarser hexahedral dominant meshing, but it takes more time to build the mesh compared to tetrahedral meshes and results might not be accurate enough.

ABBREVIATIONS

CAD	:	Computer Aided Design
CAE	:	Computer Aided Engineering
CAM	:	Computer Aided Manufacturing
CFD	:	Computational Fluid Dynamics
FSM	:	Fractional Step

LIST OF FIGURES

	PAGE
Figure 2.1: Visualisation of Hexahedral Meshing (left) and Tetrahedral Meshing with Prism Layers (right)	5
Figure 2.2: Structured Grid and Unstructured Grid Around the Nose of an Airplane ...	6
Figure 2.3: Cell types and corresponding OpenFoam keywords.....	7
Figure 2.4: Schematics of CAD data exchange.....	11
Figure 3.1: Comparison of IGES and STEP file format.	13
Figure 3.2: Precise (left) and Tessellated (right) models	14
Figure 3.3: Comparison of CAD, STL CAD file formats.....	14
Figure 4.1: Repair panel of ANSYS SpaceClaim.....	15
Figure 5.1: Before removing unnecessary parts.	18
Figure 5.2: After removing unnecessary parts.	18
Figure 5.3: Small Faces issue in intake manifold	19
Figure 5.4: Split Edges issue in intake manifold	19
Figure 5.5: Fluid Domain after repairing.	20
Figure 6.1: Recombine meshes on Gmsh.....	23
Figure 7.1: Fully Automated Mesh with Tetrahedrons.....	27
Figure 7.2: Whole Elements of the Flow Domain Coloured According To Element Quality.....	28
Figure 7.3: Number of Elements versus Skewness Graph in the Flow Domain	28
Figure 7.4: Elements with the highest Skewness value	29
Figure 7.5: Improved Tetra Mesh of the Manifold.....	29
Figure 7.6: Inflation (Boundary Layers) in the mesh	30
Figure 7.7: Whole Elements Coloured According to Their Skewness.	30

Figure 7.8: Number of Elements versus Skewness Graph in the Flow Domain	31
Figure 7.9: Unstructured Pave Grid	32
Figure 7.10: Elements coloured according to their skewness in the cross-section of the manifold	32
Figure 7.11: Highly skewed elements.....	33
Figure 7.12: Number of elements versus their Skewness value	33
Figure 7.13. Flow Volume of the Air with Hexahedral Dominant Meshing Inside the Intake Manifold	34
Figure 7.14. Whole Elements Coloured According To Their Quality.....	34
Figure 7.15: Rectangular Grid at the Circular Area of the Flow Volume	35
Figure 7.16: Elements Coloured According To Their Skewness	35
Figure 7.17: Number of Elements versus Skewness.....	36
Figure 7.18: Tetrahedral and Hexahedral elements with Skewness value close to 1	36
Figure 7.19: General view of model Butterfly mesh.	37
Figure 7.21: Lateral section view of Butterfly mesh	38
Figure 7.22: Element quality of Butterfly mesh.	39
Figure 7.23: Element quality of inside of Butterfly mesh.....	39
Figure 7.24: Distribution of Butterfly mesh element quality	40
Figure 7.25: Aspect ratio of Butterfly mesh.....	40
Figure 7.26: Distribution of Aspect Ratio in the Butterfly Mesh.....	41
Figure 7.27: Skewness of Butterfly mesh.	42
Figure 7.28: Lateral view Skewness for Butterfly mesh.....	42
Figure 7.29: Distribution of Skewness ratio.....	43
Figure 7.30: Butterfly mesh in Salome	43
Figure 7.31: Elements coloured according to their aspect ratio	44

Figure 8.1: Velocity Contour of the air	45
Figure 8.2: Velocity Streamlines of the air	45
Figure 8.3: Pressure Contour of the air	46
Figure 8.4: Velocity Contour of the air	47
Figure 8.5: Velocity Streamlines of the air	47
Figure 8.6: Pressure Contour of the air	48
Figure 8.7: Velocity Contour of the air	49
Figure 8.8: Velocity Streamlines of the air	49
Figure 8.9: Velocity Streamlines of the air	51

LIST OF TABLES

	PAGE
Table 4.1. Comparison between the grid.....	54

1. INTRODUCTION

1.1. Aim of the Research

It is known that CFD analyses are highly affected by the grids that are employed. For that reason, BMC asked us for a grid generation strategy for the CFD analysis of intake manifolds. When the literature is reviewed, it is seen that a general strategy for grid generation for such geometries is not given enough place. This study is dedicated to investigate different mesh types, qualities of grids, obtain accurate results and minimize the effort of the CFD modeller, and to come up with a grid generation strategy for circular geometries and intake manifolds.

Intake manifolds can be designed in many ways using a variety of CAD programs. While transferring data from one program to another, in most of the cases there will be some data lost or changed during the conversion of files from one format to another. This study investigates certain CAD data formats (STEP, STL, IGES). Another problem in grid generation process is the problems in the CAD data. These problems may occur because of many reasons, such as inexact faces, small faces, curve gaps etc. In order to obtain a fine grid, these problems should be solved before meshing process. This study investigates the repair process of geometries with such problems. Process of automated and manual repairing is investigated using ANSYS SpaceClaim.

Many different types of grids are used in CFD applications. Each grid type has its own advantages and disadvantages, and some might be more suitable for circular pipes or intake manifolds. A research was made on the effect of meshes on CFD analyses for circular pipes by Hernandez-Perez et al. According to their findings, accurate results can be obtained with the use of unstructured pave grid, and the best results were obtained with the use of a grid named Butterfly grid. Our study investigates the use of these grids in an intake manifold geometry designed by BMC. Because of the complexity of the geometry, these meshes cannot be directly applied, however, a strategy can be developed to use their certain advantages in hybrid meshes. Besides those grids, tetrahedral meshes were also investigated due to their easy and fast application capabilities in complex geometries.

This study aims to present solutions for BMC's problems in grid generation, reducing the effort of the CFD modeller and to contribute to CFD science in terms of investigation of grid generation strategies for intake manifolds.

1.2. Literature Survey

1.2.1. Introduction

Grid generation for CFD analysis of a CAD data can take serious time and effort. A general strategy is needed for optimal grid generation. Although there are many ways to generate grid for a geometry, there are not many generalized strategies in the literature, especially for intake manifolds. This review aims to summarize the researches done on this topic, and discuss them according to their methodology.

1.2.2. Methods for grid generation for CFD analyses

“For complex configurations, the surface grid generation step requires significant user expertise, is highly labour intensive, and typically consumes at least 80% of the total grid generation time.” [1] This problem makes it inevitable to work on a methodology which speeds up the grid generation process and make it less labour intensive. A research was done by William M. Chan regarding this problem. According to *Chan, 2017*, mesh resolution needs be higher in curving areas compared to flat surfaces. In the translation of a CAD data to STEP data, mismatches in the end points of CAD edge curves can occur due to mismatches in translation tolerances between neighbouring surfaces of a solid. This is currently repaired manually. *Chan, 2017* suggests a three-step procedure to automate the overset structured grid generation, consisting of an algebraic step, a hyperbolic step and a gap-filling step. Focusing on the algebraic step, first the connectivity between the curves is established. The next step is to apply grid point distribution on the curves, which can be done in two ways. Uniform grid point distribution is applied to short and turning curves and stretched non-uniform grid point distribution is applied to remaining curves. Groups of four-curve domains that can be concatenated together are then identified. Grid point count is adjusted on the curves subsequently in each group in the direction of concatenation. Algebraic meshes are generated inside each

4 curve domain followed by concatenation in each group. “Significant savings in manual effort and time are achieved with just the algebraic step of the automation scheme.”
(Chan, 2017)

Another research addressing the problem is done by V. Hernandez-Perez et al. According to their research, flow behaviour in a pipe is highly dependent on the type of mesh employed. They obtained the best results with the grid as known as butterfly grid and they had misleading results using cylindrical mesh. Another finding in their research is unstructured paved grid in a 3D pipe results in a memory and CPU overhead because in this grid the grid points are not connected with a regular topology and cannot be described by a function between the number of cell and the number of its neighbours. According to their observations, a difference occurs in the form of velocity vectors in different meshes. V. Hernandez-Perez et al. states “For the case of the cylindrical O-grid, the parabolic velocity profile is not clearly attained; instead, the velocity profile is almost flat, resembling turbulent flow. As shown in the results, a misleading behaviour of the flow is achieved when this mesh is used.”[2]

On the other hand, they obtained similar velocity vectors with rectangular, butterfly and paved grids. V. Hernandez-Perez et al. conducted an experiment and they found that results obtained with butterfly grid matches well with the results of experiment.

The research also showed that the mesh density also affects the performance of the mesh topology. With higher mesh density, they realized their solutions with a smaller time step, but the results are no different. According to their findings, for the rectangular H-grid, the mesh quality is reduced significantly when the mesh density is increased. They obtained better results using pave grid or butterfly grid. V. Hernandez-Perez et al. state that perhaps the major drawback of the butterfly grid is the time and expertise required for it to be built. In conclusion, they found that the polar O-grid does not success while the rectangular H-grid is suitable for low mesh densities. Both pave grid and butterfly grid gives good results however butterfly grid agrees the most. V. Hernandez-Perez et al. recommend the usage of the butterfly grid for the simulation of two-phase flow in a pipe.

1.2.3. Conclusion

The reviewed literature suggests the use of butterfly grid for circular geometries and an overset grid for complex geometries. In complex geometries, translation of CAD data to STEP format can cause misalignments, so it is needed to eliminate gaps and establish connectivity between the curves. Stretched grid point distribution is useful in long curves and uniform grid distribution can be used for short curves. Identifying groups of 4 curve domains and concatenating them saves a lot of time and effort. In circular geometries such as an intake manifold, rectangular H-grid gives acceptable results at low mesh densities, while pave grid gives accurate results at low and high mesh densities. O-grid is not recommended due to its inaccurate results. High mesh density with butterfly grid results in a smaller time step but it might take a longer time to build the mesh. Best results are obtained with the use of butterfly grid, so it is recommended.

2. MESH TYPES AND QUALITY

2.1.Two-Dimensional Mesh Types

- Triangular
- Quadrilateral
- Tri & Quad Combined

2.2.Three-Dimensional Mesh Types

- Tetrahedral
- Hexahedral
- Polyhedral
- Pyramid
- Wedge Cells (combination of these)

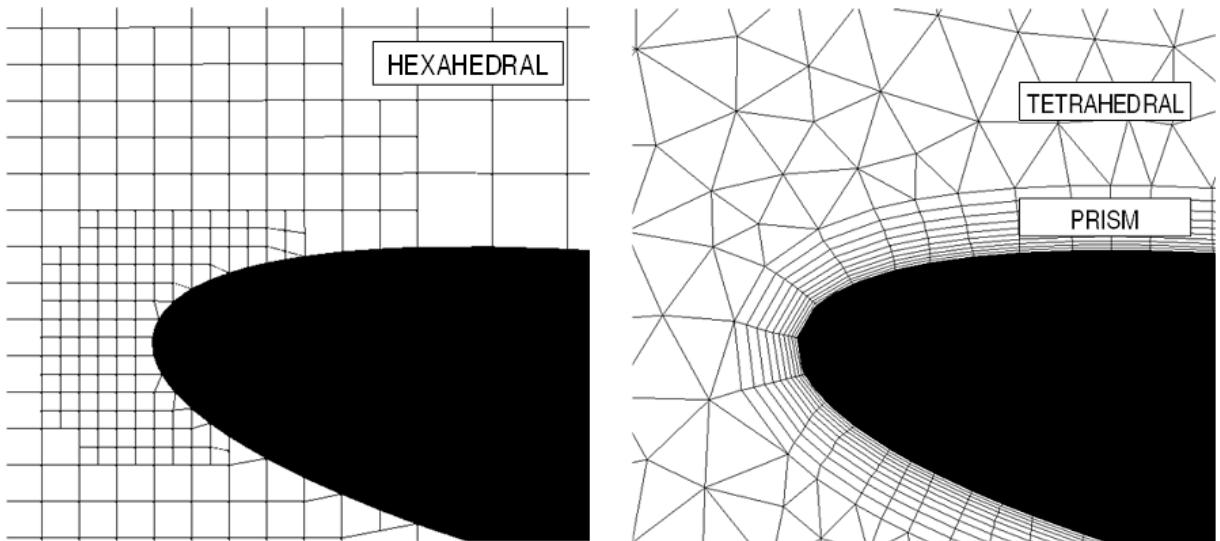


Figure 2.1: Visualisation of Hexahedral Meshing (left) and Tetrahedral Meshing with Prism Layers (right)

2.3. Classification of Grids

2.3.1. Structured grids

Quadrilateral elements in 2D and hexahedral elements in 3D can be selected to get structured grids. Structured grids have higher resolution and better convergence, and they are space efficient.

2.3.2. Unstructured grids

Unstructured grids typically employ triangles in 2D and tetrahedral in 3D. Unstructured grids may have any shape of element.

2.3.3. Hybrid grids

Hybrid Grids are a combination of structured and unstructured grids. This type of grid generation might be useful in the areas of complex geometry or areas that are close to the boundaries.

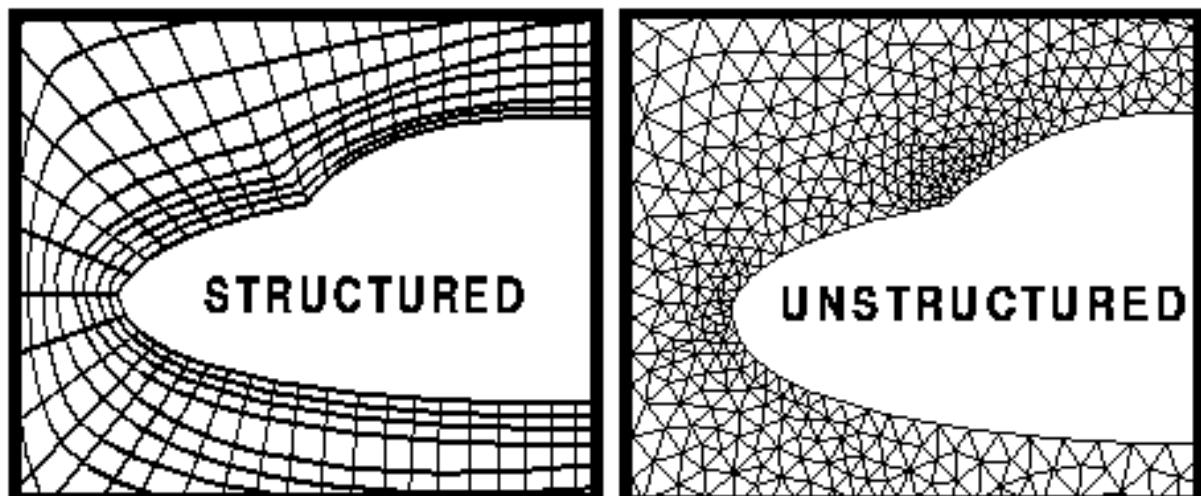


Figure 2.2: Structured Grid and Unstructured Grid Around the Nose of an Airplane

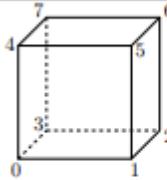
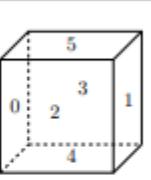
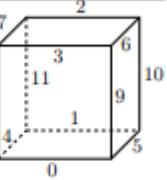
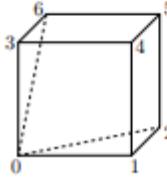
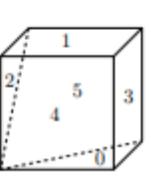
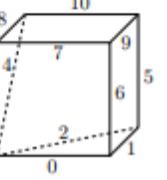
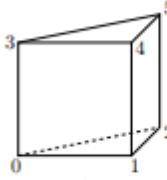
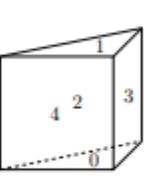
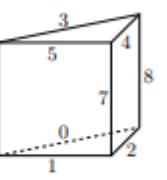
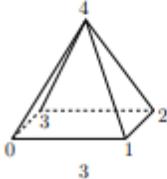
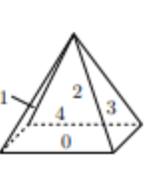
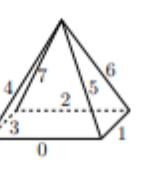
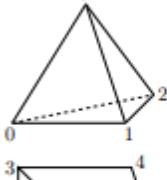
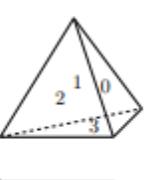
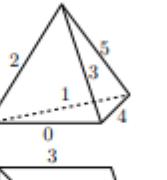
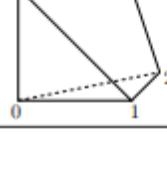
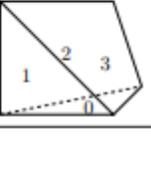
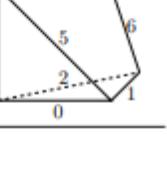
Cell type	Keyword	Vertex numbering	Face numbering	Edge numbering
Hexahedron	hex			
Wedge	wedge			
Prism	prism			
Pyramid	pyr			
Tetrahedron	tet			
Tet-wedge	tetWedge			

Figure 2.3: Cell types and corresponding OpenFoam keywords

2.4. Choosing the correct mesh

Choosing the correct mesh is depended on 3 parameters according to ANSYS:

- Setup time
- Computational expense
- Numerical Diffusion

2.4.1. Setup time

In our flow analysis, we have complex geometries in some regions. The creation of structured meshes such as quadrilateral or hexahedral can be extremely time consuming. So setup time is the critical concern while dealing with complex geometries. Using unstructured meshes with triangular or tetrahedral cells would save a lot of time for complex geometries.

2.4.2. Computational expense

In relatively simple geometries with the flow conforms well to the shape of geometry, using high-aspect ratio hexahedral cells might lower the element number than using tetrahedral cells. For simple geometries, it is recommended to use quadrilateral/hexahedral meshes. For relatively complex geometries, using triangular/tetrahedral cells with prism layers is recommended. For extremely complex geometries, using pure triangular/tetrahedral cells is recommended.

2.4.3. Numerical diffusion

Numerical diffusion is a dominant source of error in multidimensional situations. Numerical diffusion is minimal when the flow is aligned with the mesh. While using triangular/tetrahedral mesh, it is impossible for the flow to be aligned with the mesh. It is more likely to happen when using quadrilateral/hexahedral meshes, but *not for complex flows*. Creating prism layers near the walls might be useful if the flow is parallel in those areas.

2.5. Decision Process

From the information given above, we can understand that the different cell types and different meshing methods can give very different results and the setup time can vary greatly. To optimize the accuracy and calculation time we have decided to try tetrahedral, hexahedral and hybrid meshing. Our goal is to obtain the shortest calculation and setup time while having accurate results.

2.6.Mesh Quality

Mesh quality can be considered one of the most important parameters in a CFD analysis. A high quality mesh is recommended to be obtained for accurate results, time savings, and less computational expense. There is no such thing as best mesh, but a general quality check can be done by inspecting mesh metrics. In this chapter, a few metrics will be inspected

Orthogonality:

The mesh orthogonality concept is about how close the angles between adjacent element faces or edges are to an optimal angle. This angle is dependent on topology. The orthogonality measure ranges from 0 to 1, where 0 is bad and 1 is good.

Skewness:

Skewness in tetrahedral elements is described as the deviation from an optimal (equilateral) volume.

$$\text{Skewness} = \frac{\text{optimal cell size} - \text{actual cell size}}{\text{optimal cell size}}$$

For all kinds of cells, it can be defined by the Normalized Angle Deviation Concept by finding the minimum angle between two lines joining opposite-mid sides of the element.

$$\text{Skewness} = \max \left[\frac{\theta_{\max} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{\min}}{\theta_e} \right]$$

Skewness metric ranges from 0 to 1, where 0 is good and 1 is bad.

Aspect Ratio:

The aspect ratio is defined as length to height ratio in 2D or as the radius ratio of circumscribed to the inscribed circles in 3D.

3. CAD EXCHANGE DATA FORMATS

Computer Aided design (CAD) files are generated with different types. Each file types have advantages and disadvantages. Neutral (Non-proprietary) and Native (proprietary) are the two main classifications of CAD file formats. Product should pass certain phases of CAD before manufacturing. Procedures of CAD file creation process could be generated in various software so compatibility of files have strong relation with efficiency of time. CAD Industry is needed to be extremely precise. Modern CAD tools can represent holes a micron in diameter on extremely huge parts . These models are commonly called “precise solids,” or more technically, “boundary representation” (B-rep) solid models. the models need to be converted to an approximation of the essential design to visualize the precise solid on the graphics processors. These graphics “tessellated,” or “mesh” models are generated every time for modifying the design. Most CAD systems store the graphics in their files along with the precise data. Some issues occur when files are exchanged between software. Some issues are listed in the below.

- Missing, collapsed or inverted faces.
- Lines that cross at corners.
- Lines that do not meet at corners
- Liners or surface coincident with other surface of lines.
- Models that do not become closed solids.
- Lines that building by various short line group.
- Curves that not coincident to lines or surfaces

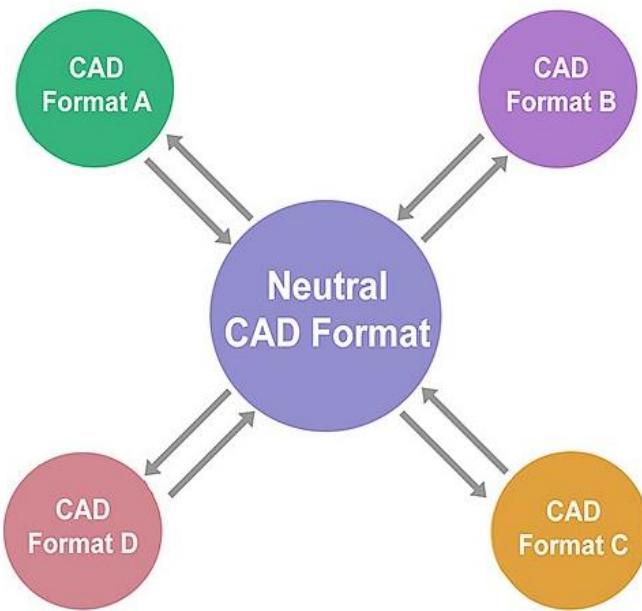


Figure 2.4: Schematics of CAD data exchange

3.1. Neutral Cad File Format

Neutral CAD file formats are effective for sharing data between different CAD software. Neutral CAD file formats have significant role to transfer CAD data but neutral formats cannot store all information about CAD model because their main aim is transferring geometry data with minimum capacity. High storage capacity causes increase of storage cost. Popular neutral CAD file formats are explained below.

3.1.1. STL - Stereolithography or “Standard Tessellation Language”

STL is a native file format of the stereolithography CAD software created by 3D Systems. STL file format is supported by various software. This file format is one of the common native exchange file format. STL has several backronyms like Standard Triangle Language and Standard Tessellation Language. STL files contain no scale information and units are arbitrary. STL files commonly used for 3D printing, Computer-Aided Manufacturing (CAM). STL files focus on the model's geometry so any model colour, texture or any attributes are not included.

3.1.2. STP - ISO 10303 or STEP

ISO 10303 (STEP) is ISO standard for exchange of product manufacturing information. This standard called as STEP. STEP is one of the most widely used CAD file format. Most modern CAD software supports STEP file format. STEP files stores 3D images in an American Standard Code for Information Interchange (ASCII) format. STEP files contain three-dimensional data. STEP file format had improved.

3.1.3. IGES - Initial graphics exchange specification

IGES is another popular non-proprietary format. This format is first developed in 1980 and widely used in the CAD industry since then. It is developed exclusively by the US Air Force, but once they have started to use other technology, the format is taken over by ANSI. This file format is used to display wireframes and circuit diagrams, freeform surface or solid modelling representation of product models. Because of its lack of colour rendering and textures, it should generally not be used to share designs with third-party organizations, unless they know the design process or the product. IGES file format is editable but it is not simple because IGES files are always included surfaces. This situation occurs editing problems and decreasing time efficiency. IGES file format cannot contain some information about model like volume, surface is centroid or moment of inertia.

Feature	IGES	STEP
Import/Export 2D CAD	Yes	Yes
Import/Export 3D CAD	Limited	Good
Editable?	Yes	No
Continuing support	No	Yes
Capture 3D solid models?	No	Yes
Scope: From design to life cycle?	No	Yes

Figure 3.1: Comparison of IGES and STEP file formats.

3.2. Native CAD File Format

Native file formats are created by commercial companies for use in their software. Native CAD formats are proprietary to specific CAD software. Format reading capacity is increased because more information and more accurate geometric and model attributes are required for high quality modelling and production. Quality of model is crucial to effective communication of different departments of companies or between members of a project. Some popular native CAD file formats are DXF, DWG, SLDPR, SLDASM, MODEL, PAR, ASM, PSM, IPT, IAM, X_T.

3.2.1. DXF and DWG (AutoCAD)

Drawing Exchange Format, DXF, is a vector file format and was created by Autodesk as an exchange medium between different types of CAD software. It is an open standard, so it's supported by practically every CAD program in the market.

DWG is used as an abbreviation for 'Drawing,' which is a proprietary vector file format created by Autodesk in 1982. Developers need a license to access this format in their software. DWG files are smaller than DXF files due to their binary makeup. Users can view and edit DWG files with other programs too, including Scan2CAD, though they are designed to be used in AutoCAD.

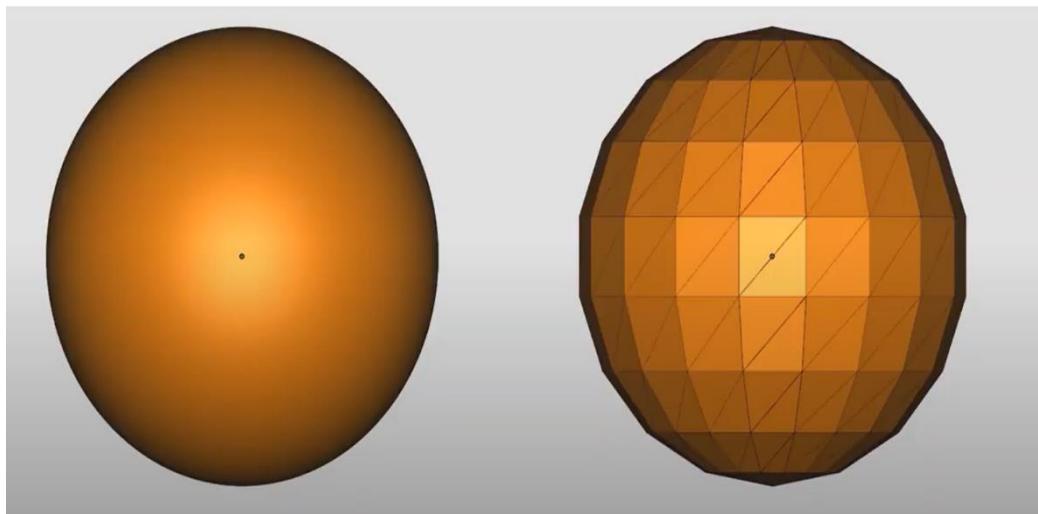


Figure 3.2: Precise (left) and Tessellated (right) models



Figure 3.3: Comparison of CAD, STL CAD file formats.

4. REPAIRING PROCESS

Design and Simulation have three phases. There are, pre-processing, processing and post processing. Pre-processing includes sketching or modelling, repairing and meshing. Processing contains solver stage and post-processing covers graphs, reports, visual contents.

Pre-processing stage is significant for reducing solver operating time and increasing accuracy of results. This stage contains creating 3D model and simplification of geometry due to efficiency. 3D models should have been optimized for time and accuracy of simulation. In this study ANSYS SpaceClaim was used for fixing geometry issues.

4.1.ANSYS SpaceClaim Repair Section Tools

ANSYS SpaceClaim various repairing tools. SpaceClaim can import a variety of native and neutral CAD formats, but it may be needed to clean up and repair this data for using in SpaceClaim and for Computer Aided Engineering (CAE). The Repair tab includes tools that can be used to repair imported models and prepare designs for exporting and analysis.

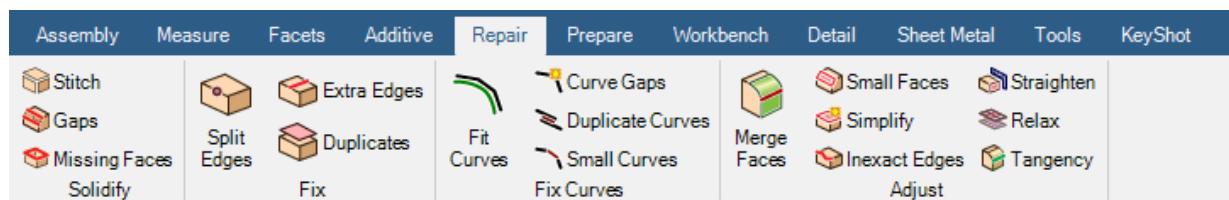


Figure 4.1: Repair panel of ANSYS SpaceClaim

4.1.1. Solidify section

Stitch: Stitch used for combining surfaces to convert a body.

Gaps: Detecting and fixing gaps in body surface.

Missing Face: Detect and fixing missing face on the surface.

4.1.2. Fix section

Split Edges: Split edges used for fixing coincident of edges. Coincident edges occurred when reading different file format from used software file format.

Extra Edges: Extra edges tool solves multiple line of edges. Deleting useless lines occur more reliable simulation.

Duplicates: Duplicate tool is used for deleting extra faces. Solid body should be formed surfaces which are containing only one layer of surface. Extra surfaces cause more complex geometry.

4.1.3. Fix curves section

Fit Curves: Fix curves tool is used for increasing arc, spline or curve quality. Some file formats cannot create high quality curve so in this situation increasing quality of curve needed.

Curve Gaps: This is utilized for merging curve gaps. If geometry has got any gaps, solid model cannot be generated.

Duplicate Curves: Detecting and deleting redundant curves are needed for simplification of geometry.

Small Curves: Extremely small curves were obstructed while creating smooth curve. Curve divisions, which are undesired situations, results in complexity.

4.1.4. Adjust section

Merge Faces: Merge face used for decreasing number of a face.

Small Faces: Small faces tool eliminates small or sliver faces. More faces increase calculation time.

Simplify: Simplifying curves and faces decrease the complexity of body. Cones, cylinders, arcs and planes are simplified with this tool.

Inexact Edges: Inexact edges tool solves edges that do no precisely lie at the intersection of two faces.

Straighten: Straighten tool is used to straighten faces that are at unwanted angles.

Relax: Relax tool is utilized to reduce the number of control points in imported surfaces to make them more stable.

Tangency: Tangency tool's aim is to change nearly tangent faces so they are tangent.

5. IMPORTING CAD FILE AND REPAIRING SECTION

In this study, CAD files is given three different file formats. There are stp, .scdoc and x_t file formats. Step file format used in SolidWorks for detaching the assembly. CAD files includes all parts of intake manifold, so unnecessary parts are removed from 3D model. Main geometric dimensional data is conserved in step file format. Outlets of the intake manifold have complex geometry. Curves and fillets causes difficulties at the meshing and solver stages. CAD models should be simple for flow analysis because of flow domain is a significant point. The mesh on the inner surface of the manifold wall directly affects the mesh of the flow domain. Curves and fillets occurred diverging of simulation. Parts have some CAD problem. Repairing was started after detaching and combining the essential parts. Fluid domain volume was created in SpaceClaim. Repairing is executed in SpaceClaim software.

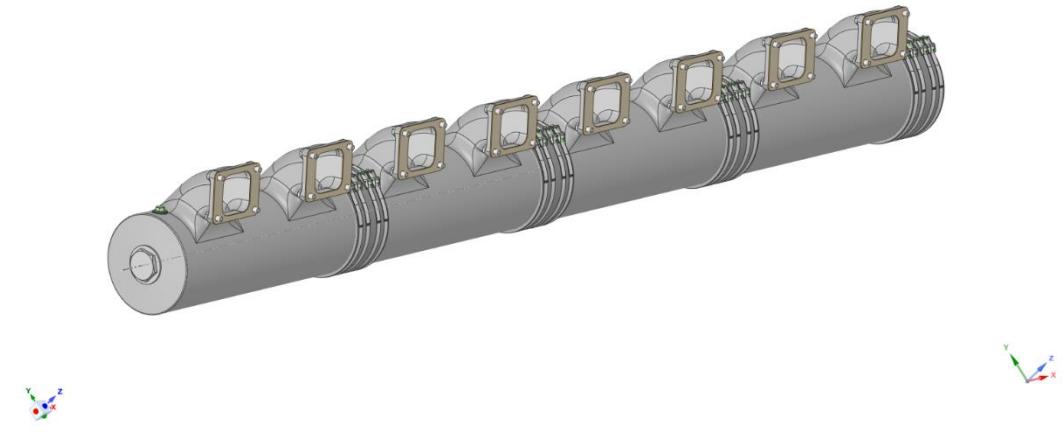


Figure 5.1: Before removing unnecessary parts.

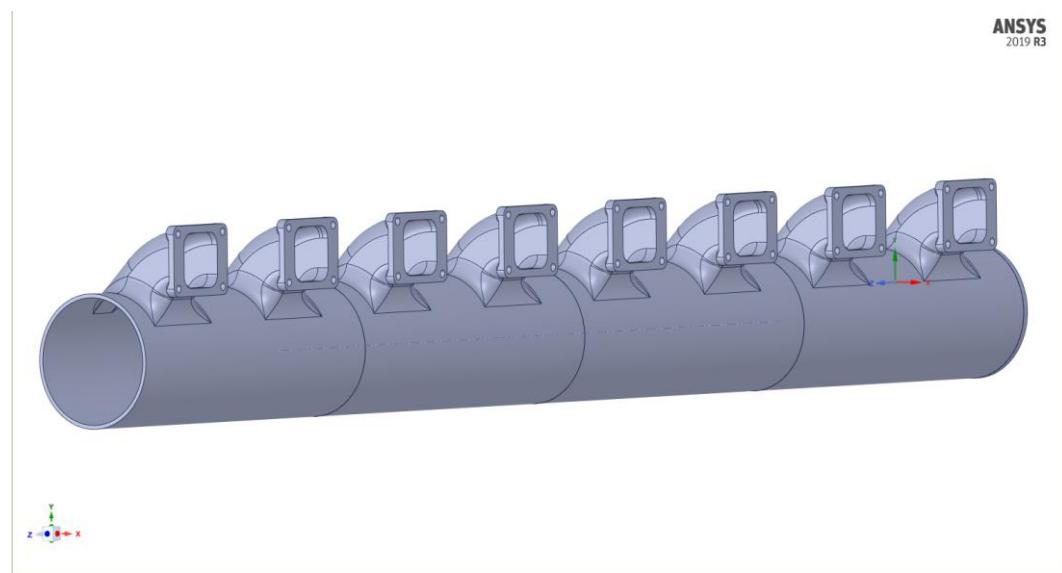


Figure 5.2: After removing unnecessary parts.

In SpaceClaim some geometric issues are fixed with repair tools. There were no solidification problems in the geometry. Split edges and duplication tools were used for initial repairing. Most of problems come from the fillets or curves. Small faces are eliminated or merged with certain faces on the flow domain. Simplify tool doesn't work properly because of the curves of the outlet section. It is not solvable for repairing tools and exact CAD data cannot be accessible in the step file format.

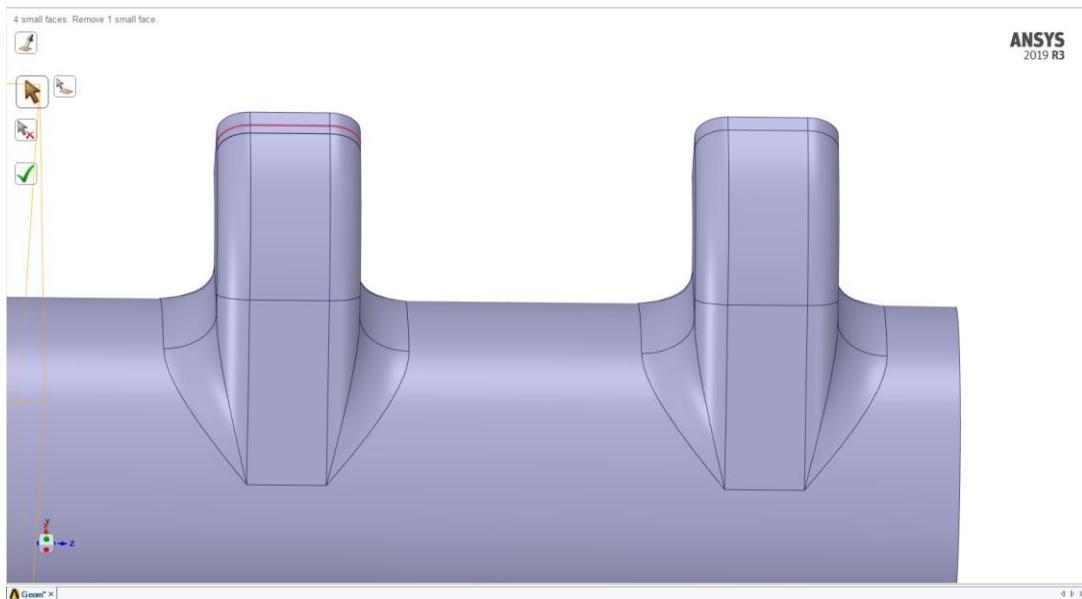


Figure 5.3: Small Faces issue in intake manifold

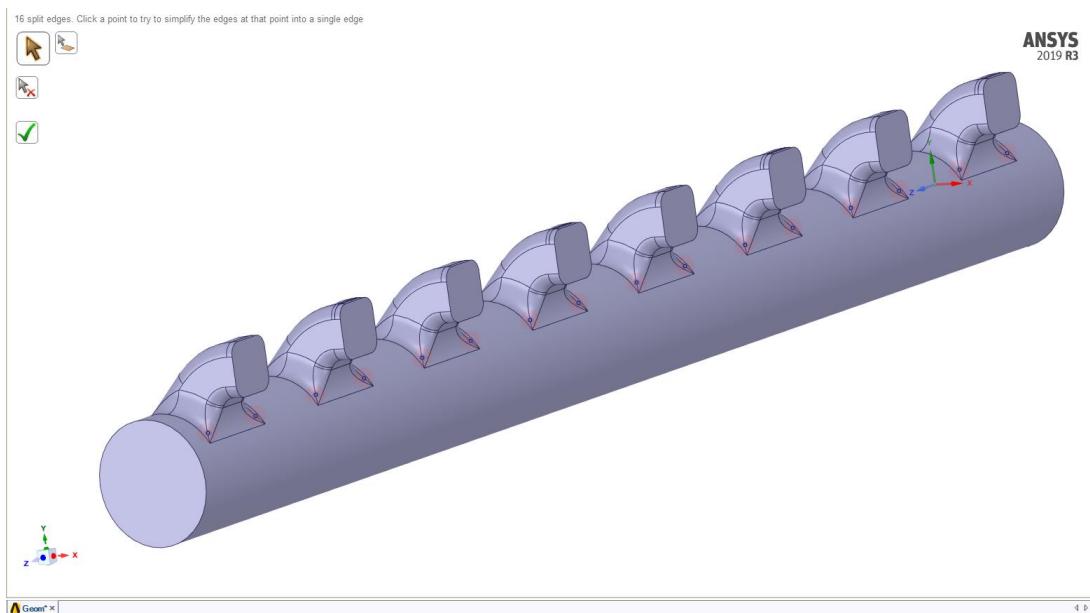


Figure 5.4: Split Edges issue in intake manifold

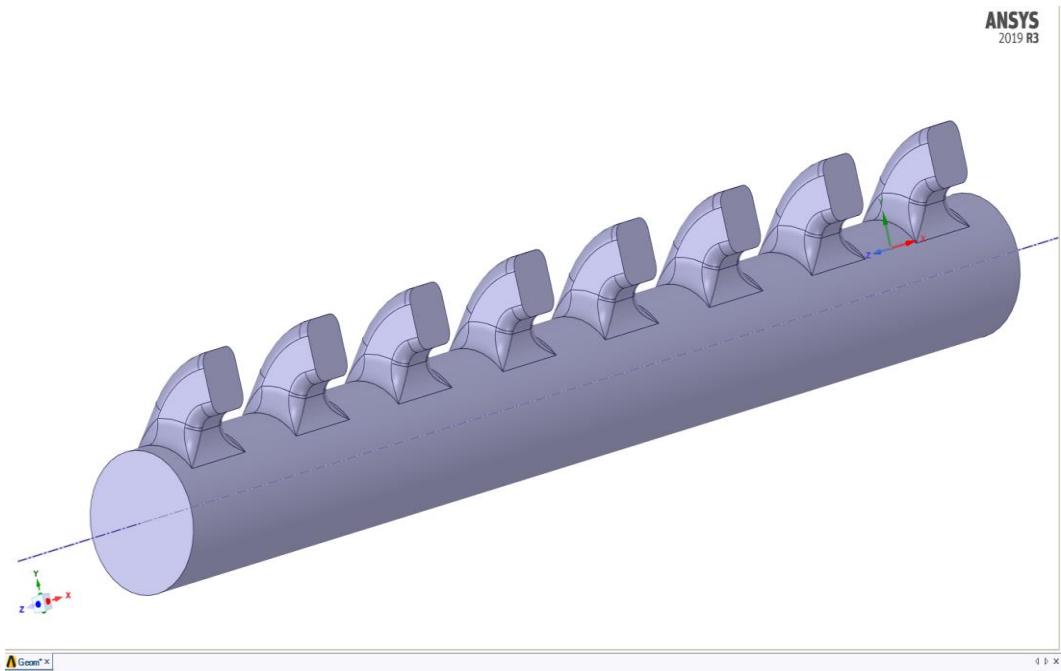


Figure 5.5: Fluid Domain after repairing.

6. CFD SOFTWARE

6.1. Open-Source CFD Software:

Open-source software solutions have some advantages and disadvantages when executing the process of simulation. Open-source software allowed the widely change option in the simulation especially during meshing and solver stages. Open Source Field Operation and Manipulation (OpenFoam) is the most common platform of open-source CFD programs. OpenFoam is a framework for developing application executable that use packaged functionality contained within a collection of approximately 100 C+ Libraries. OpenFoam is shipped with approximately 250 pre-built applications that fall into two categories: solvers that are each designed to solve a specific problem in fluid mechanics and utilities, that are designed to perform task that involve data manipulation. The solvers in OpenFoam contain a wide range of issues, utilities and libraries in OpenFoam, using some pre-requisite knowledge of underlying method, physic and programming techniques involved. OpenFoam cooperated various open-source software like SALOME, Gmsh and CAESES.

6.1.1. Pre-Processing

Cases are setup in OpenFoam by editing case files. Users should select an editor of choice with which to do this. Editing files is possible in OpenFoam so this condition allows the changing each pre-processing tools.

6.1.2. Mesh generation

OpenFoam always operates in a 3 dimensional Cartesian coordinate system and all geometries are generated in 3 Dimensions but it can be modified to solve in 2 dimensions by specifying a special empty boundary condition.
BlockMesh and Snappyhexm

6.1.2.1. BlockMesh

The BlockMesh utility creates parametric meshes with grading and curved edges. The principle of BlockMesh is the decompose the domain geometry into a set of 1 or more three dimensional, Hexahedral blocks.

6.1.2.2. SnappyHexMesh

The snappyHexMesh utility generates 3-Dimensional meshing containing hexahedra (hex) and split-hexahedra automatically from triangulated surfaces geometries or tri surfaces in STL or Wavefront object (OBJ) format.

6.1.3. Mesh conversion:

OpenFoam allowed the meshes which creates other programs like ANSYS Fluent, Gambit, CFX or Star-CD / Prostar.

6.1.4. Solver:

OpenFoam has different type of solver which simulation is needed. Incompressible, Compressible, Multiphase flow, Particle Tracking Flows, are some of the solver of OpenFoam.

6.1.5. Post-Processing

OpenFoam used ParaView post-processing utility. ParaView uses the Visualisation Toolkit (VTK) as its data processing and rendering engine and can therefore read any data in VTK format.

6.2.Gmsh

Gmsh is an open source 3D finite element mesh generator with build in CAD engine and post processor. Its design aim is to provide fast, light and user friendly meshing tools with parametric input and advanced visualization capabilities but in complex geometries Gmsh does not work properly and proprietary 3D CAD software are easier than Gmsh because of selection each point, curve or volume data and Gmsh needs the capability of editing geometry on textbook. For simple geometries, meshing and post- processing in Gmsh can be suitable.

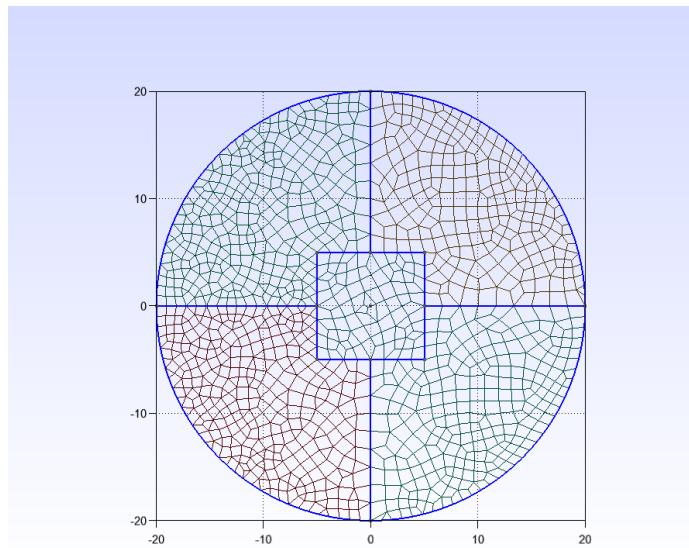


Figure 6.1: Recombine meshes on Gmsh

6.3.SALOME

Salome is an open-source software that provides a generic Pre and Post Processing platform for numerical simulation. It is based on an open and flexible architecture. Like Gmsh, Salome is not best on the complex geometry. Geometry must be repaired and curves should be smooth otherwise creating mesh last longer.

6.4.ANSYS Fluent

ANSYS Fluent is a computer program for modelling fluid flow, chemical reactions and heat transfer for complex geometries. ANSYS Fluent can create unstructured meshes for complex geometries. The program supports 2D quadrilateral/triangular, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and hybrid meshes. Meshes can be refined or coarsened by the program.

Meshes can be read by ANSYS Fluent or can be created by using the mesh mode of Fluent. Remaining operations such as setting boundary conditions, defining fluid properties, executing the solution, refining the mesh, viewing the results and post processing can be performed by the solution mode of Fluent.

6.4.1. Modelling turbulence in ANSYS Fluent

Turbulence is described by Navier-Stokes equations and solving these equations by Direct Numerical Simulation is not convenient in most of the cases, since they exceed the capabilities of a regular CPU by far. Therefore, averaging procedures are applied and the most widely used procedure is Reynolds-Averaged Navier-Stokes (RANS) equations. The averaging process introduces new unknown terms into the transport equations that need to be provided by suitable turbulence models. The selected turbulence model has a crucial impact on the quality of the simulation, so it is important to make choose the proper model.

Some models provided by ANSYS Fluent:

- Spalart-Allmaras model
- k- ε models
 - Standard k- ε model
 - Renormalization-group (RNG) k- ε model
 - Realizable k- ε model
- k- ω models
 - Standard k- ω model
 - Baseline (BSL) model
 - Shear-Stress transport (SST) model
- Transition k-kl- ω model
- Transition SST model
- Reynolds stress models (RSM)

And many more.

k- ε models:

Two equation models are the most widely used models in industrial CFD history. The standard k- ε model in ANSYS Fluent falls in this category. The model is robust, economical and accurate for wide range of turbulent flows. This model is not recommended for external aerodynamics. In ANSYS Fluent, the use of the Realizable k- ε model is recommended relative to other k- ε variants.

k- ω models:

The k- ω equation has several advantages relative to the k- ε model. A significant one of them is that the equation can be integrated without additional terms through the viscous sublayer. The k- ω models are typically better at predicting adverse pressure gradient boundary layer flows and separation. The downside of the standard k- ω equation is a relatively strong sensitivity of the solution depending on the freestream values of k and ω outside the shear layer. For this reason, the use of k- ω model is not generally recommended in ANSYS fluent.

6.4.2. Solvers in ANSYS Fluent

Two solver technologies are available in ANSYS Fluent:

- Pressure based
- Density based

For incompressible and mildly compressible flows, pressure-based solver is used. The density-based approach was designed for high speed compressible flows.

Two algorithms exist under the pressure-based solver in ANSYS Fluent: a segregated algorithm and a coupled algorithm. In the segregated algorithm the governing equations are solved sequentially, while in the coupled algorithm the momentum equations and the pressure-based continuity equation are solved in a coupled manner. The convergence speed is generally significantly faster when coupled algorithm is used, but it requires more memory.

6.4.3. Segregated algorithms in ANSYS Fluent

ANSYS Fluent provides 4 types of segregated algorithms: SIMPLE, SIMPLEC, PISO and Fractional Step (FSM). These are referred as pressure-based segregated algorithms. SIMPLE and SIMPLEC are generally used for steady-state calculations, while PISO is recommended for transient calculations. PISO may be useful for steady-state and transient calculations on highly skewed meshes. For relatively uncomplicated problems, SIMPLEC algorithm may realize faster convergence because the under-relaxation factor is set to 1.0, but it can lead to instability if there is high mesh skewness.

7. GRID GENERATION IN ANSYS FLUENT

The mesh has a great impact on the results and convergence of a CFD analysis. Different meshes result in different time-steps, the time it takes to build the mesh also varies according to the mesh and the accuracy of the results are directly impacted by the mesh. For this reason, it is recommended for the designer to find the optimal mesh for every specific problem. However, at least for similar geometries, like circular pipes and intake manifolds, a general meshing guide can be composed.

An intake manifold designed for a V16 Locomotive engine with a diameter of 198 mm is used for analyses. The intake manifold has 8 outlets and its length is 2380mm. The design of the manifold is granted by BMC. The k- ϵ model is used for the solution model since flow inside the manifold is considered to be turbulent. Solution method SIMPLE is selected. The number of cells affects the solution greatly, so a varying number of cells is achieved by manipulating the global element size and maximum element size parameters. The first generated mesh is fully automated unstructured mesh with tetrahedrons by ANSYS Fluent Meshing. Generation of this mesh is very fast and it converges quickly. The results are close to estimated results but not sufficient. The second mesh is a modified unstructured tetra mesh. It is generated automatically by the program, but it is improved manually. Global element size and maximum element size are kept in default. This results in a very fast mesh generation and a fast convergence. Boundary layers are added and fine smoothing is selected. More accurate and relatively fast results are obtained with this mesh. The third mesh is hexahedral dominant mesh. This mesh is also generated by the

program automatically but improved manually. Element size is reduced and edge sizing is applied. In comparison with tetrahedral meshing, this mesh takes a lot of time to build and a lot of time to solve. The fourth mesh is rectangular mesh. This mesh results in high skewness in denser areas and the solution does not converge.

Complex geometries can be split into bodies and they can be meshed separately to achieve better results. Structured meshing can be generated effectively in separate bodies and the meshes can be combined. This gives the most accurate results, however in this CFD analysis, the part has many complex curves so contact areas give bad results when it's split into bodies.

7.1.Fully Automated Unstructured Tetra Mesh

ANSYS Fluent can generate full automated meshes for complex geometries. By default options, a very quick mesh is generated by the program. The mesh is constructed

by tetrahedrons and relatively large elements. This is not ideally the best mesh but it may be useful to obtain very quick results.

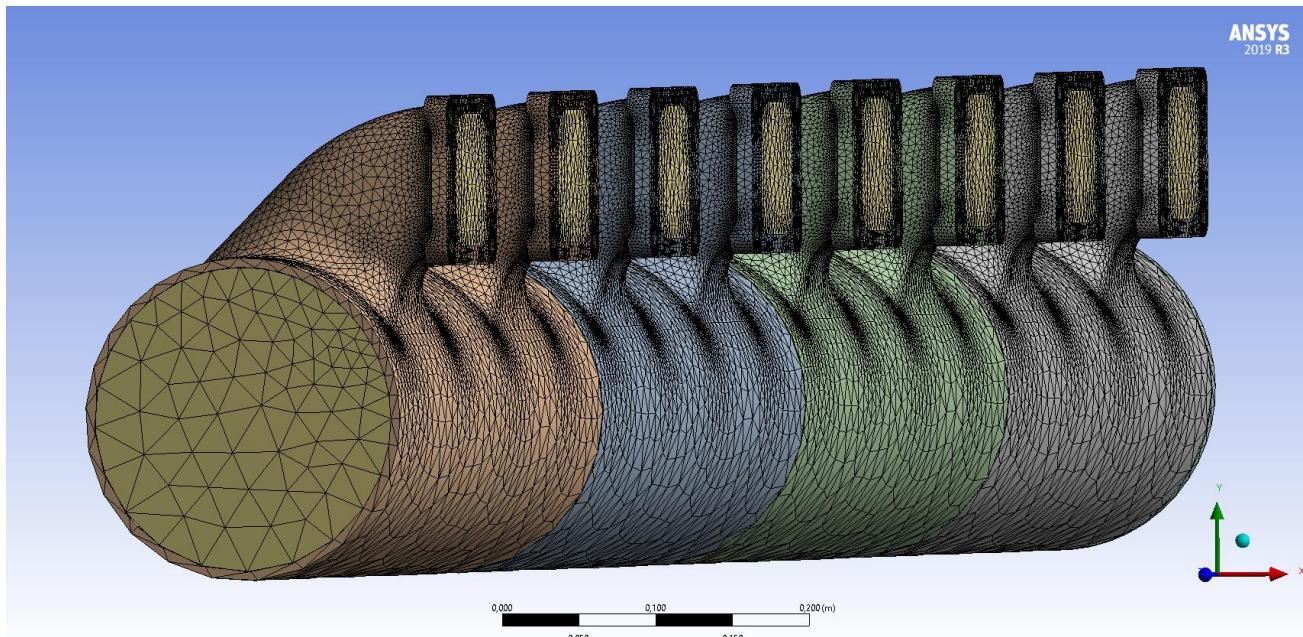


Figure 7.1: Fully Automated Mesh with Tetrahedrons

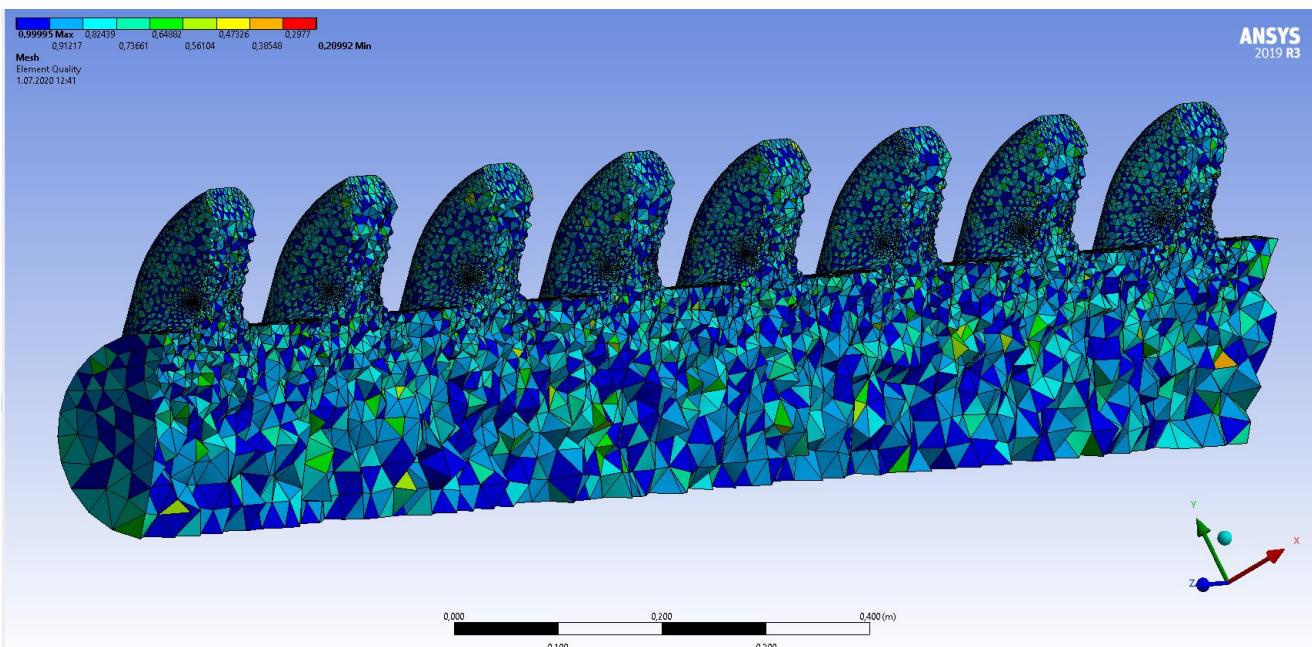


Figure 7.2: Whole Elements of the Flow Domain Coloured According To Element Quality

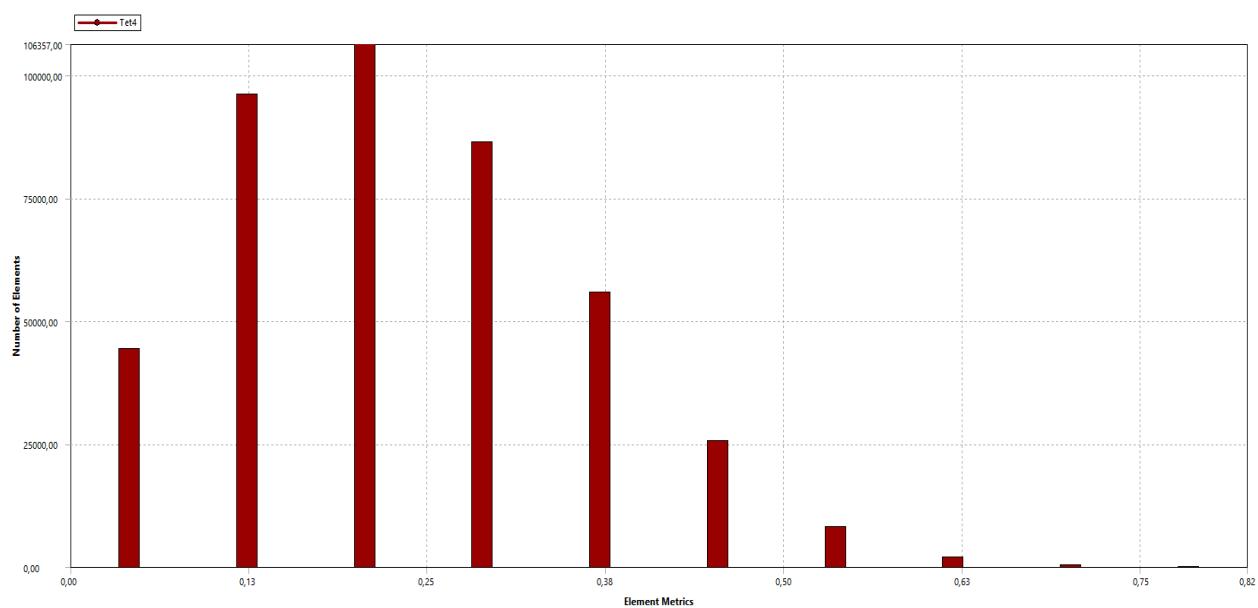


Figure 7.3: Number of Elements versus Skewness Graph in the Flow Domain

Since this is a CFD analysis, only the elements in the flow domain are analysed. Finer mesh is needed in the flow domain and coarser mesh can be used in the walls of the intake manifold

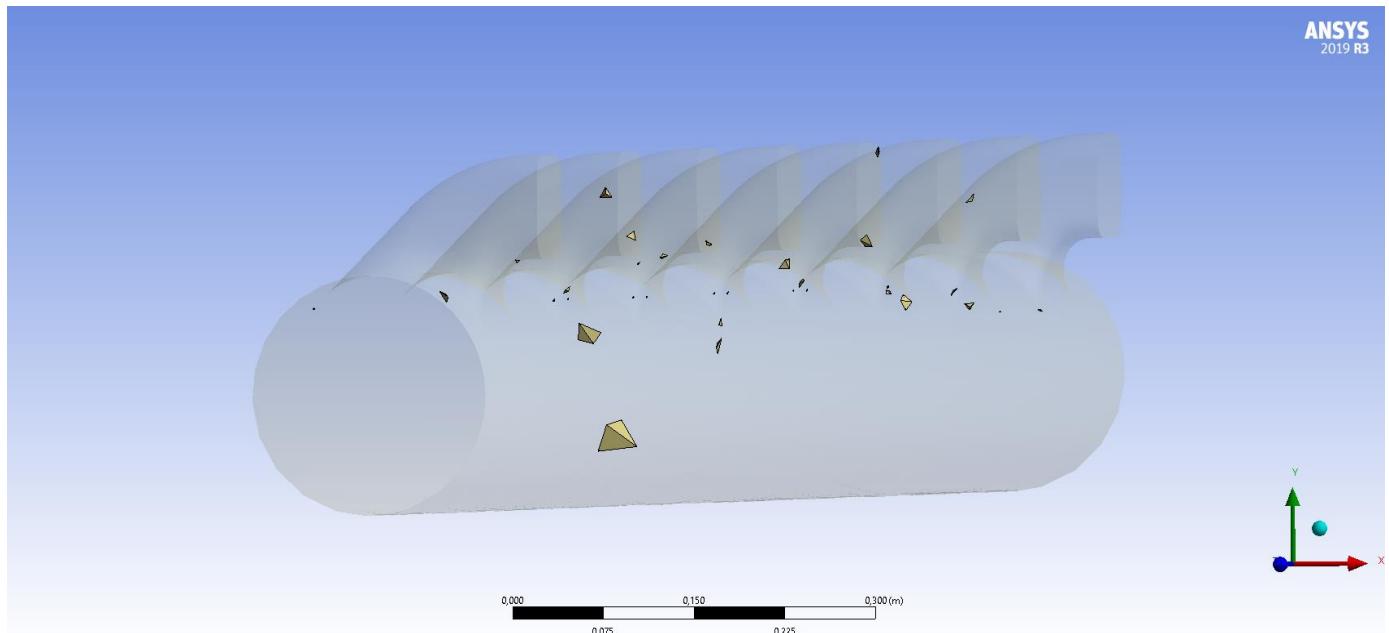


Figure 7.4: Elements with the highest Skewness value

7.2. Manually Improved Unstructured Tetra Mesh

An unstructured mesh with tetrahedrons was obtained in the first meshing. This mesh can easily be improved by the designer for better results. In this mesh, boundary layers were added for better results near the pipe walls. Edge sizing is added for more uniform element distribution.

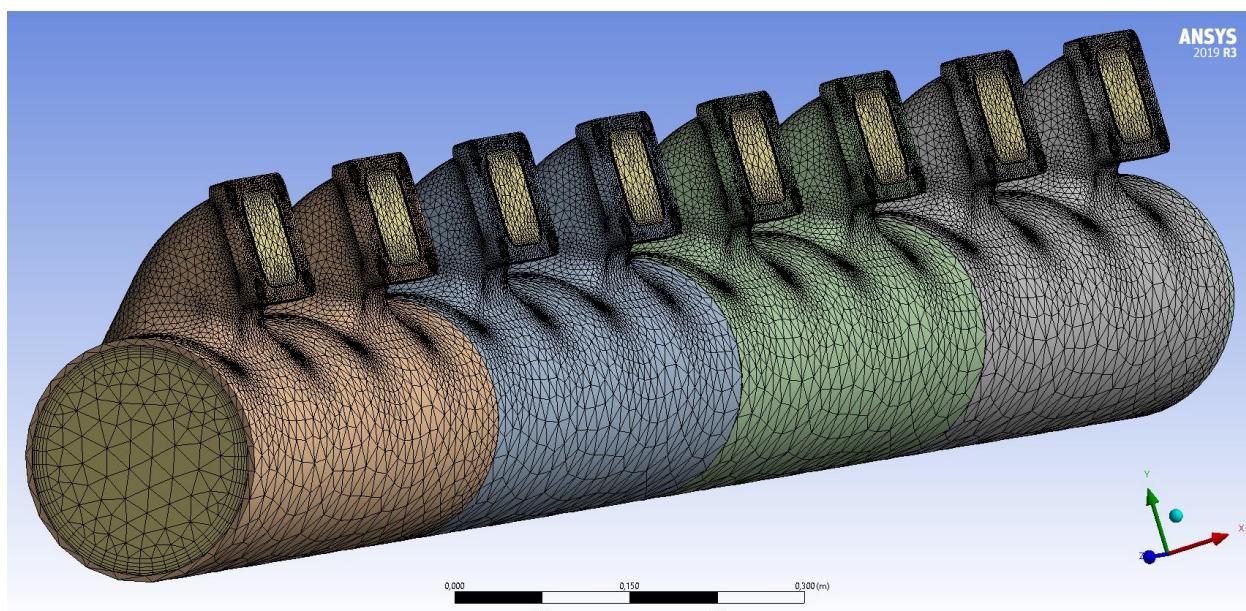


Figure 7.5: Improved Tetra Mesh of the Manifold

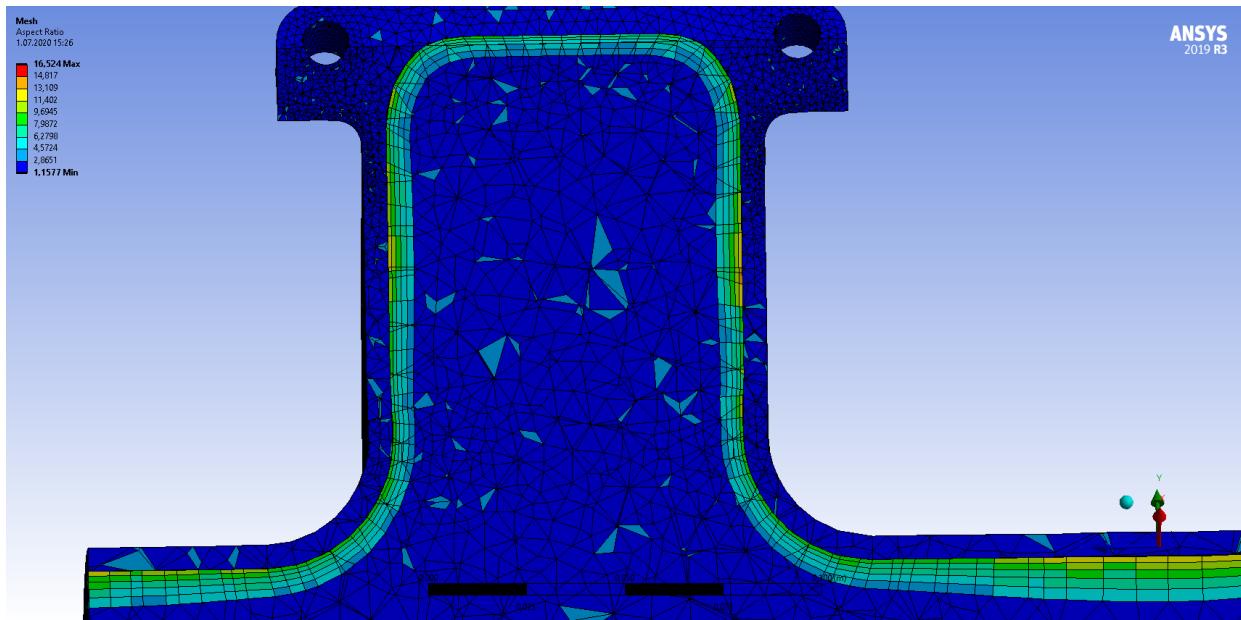


Figure 7.6: Inflation (Boundary Layers) in the mesh

It is observed that the boundary layers have high aspect ratios, thus a lower element quality. Nevertheless, having elements parallel to the flow direction proves useful near the pipe surface, where no-slip condition occurs.

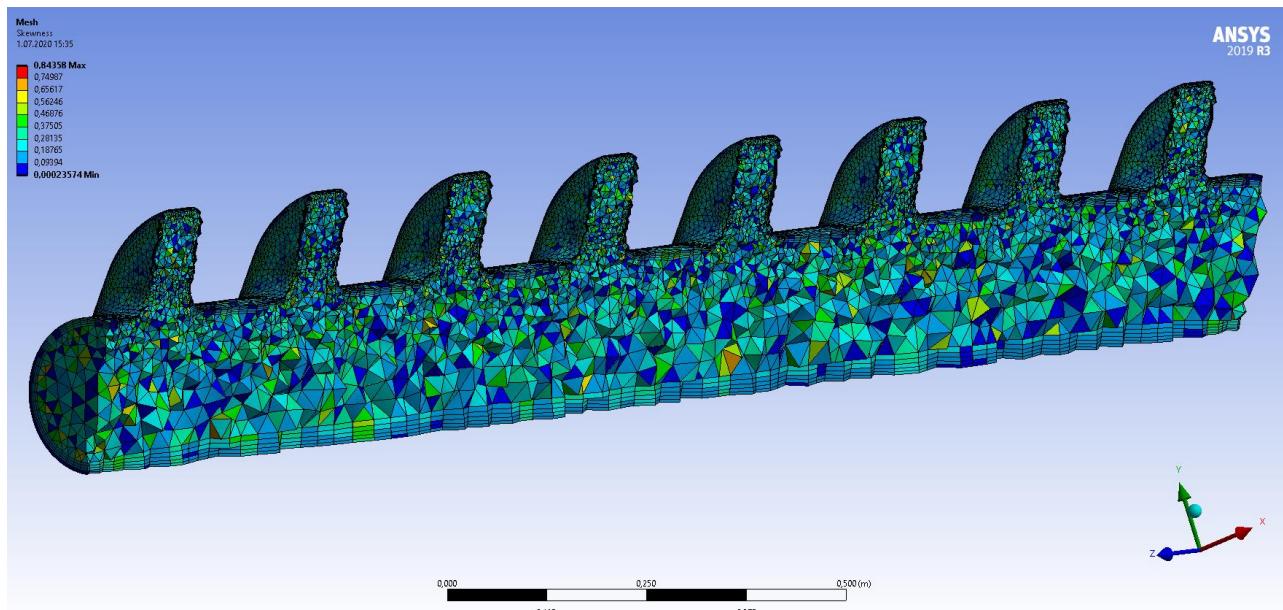


Figure 7.7: Whole Elements Coloured According to Their Skewness.

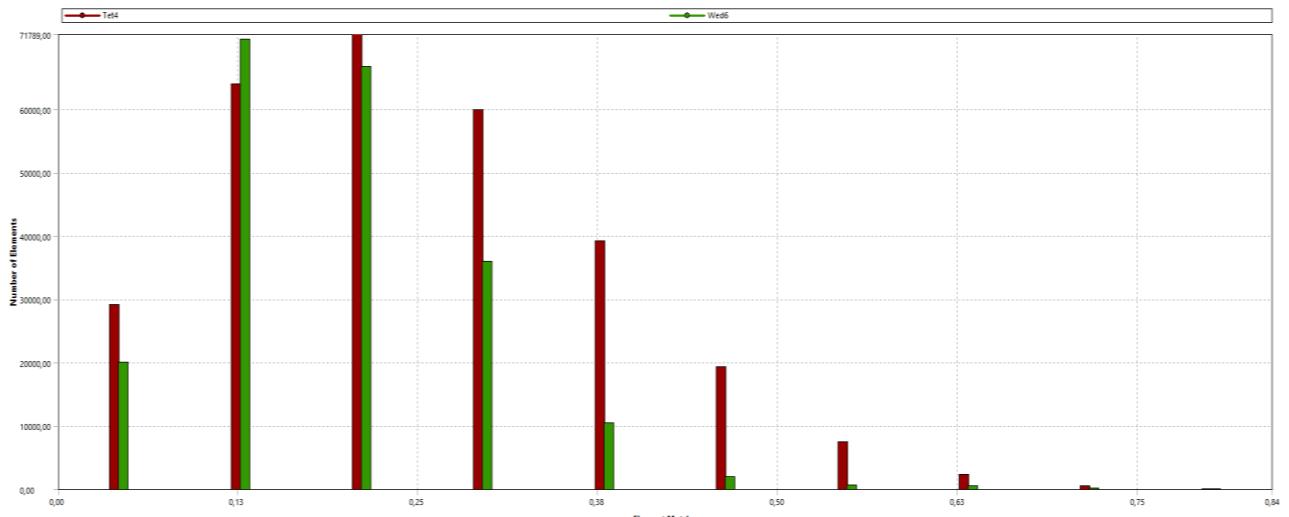


Figure 7.8: Number of Elements versus Skewness Graph in the Flow Domain

7.3.Unstructured Pave Grid

Pave grid with hybrid elements is recommended in the literature for CFD analyses for circular pipes. In this geometry though, this grid resulted in very highly skewed elements. An unstructured pave grid is generated automatically by ANSYS Fluent. This grid has both hexahedral and tetrahedral elements. Default element size was selected and high smoothing is used. There are some highly skewed tetra elements in this mesh, which are shown in figure 11. Figure 12 shows the amount of highly skewed tetra elements. Due to these high number of skewed elements, a better grid is necessary.

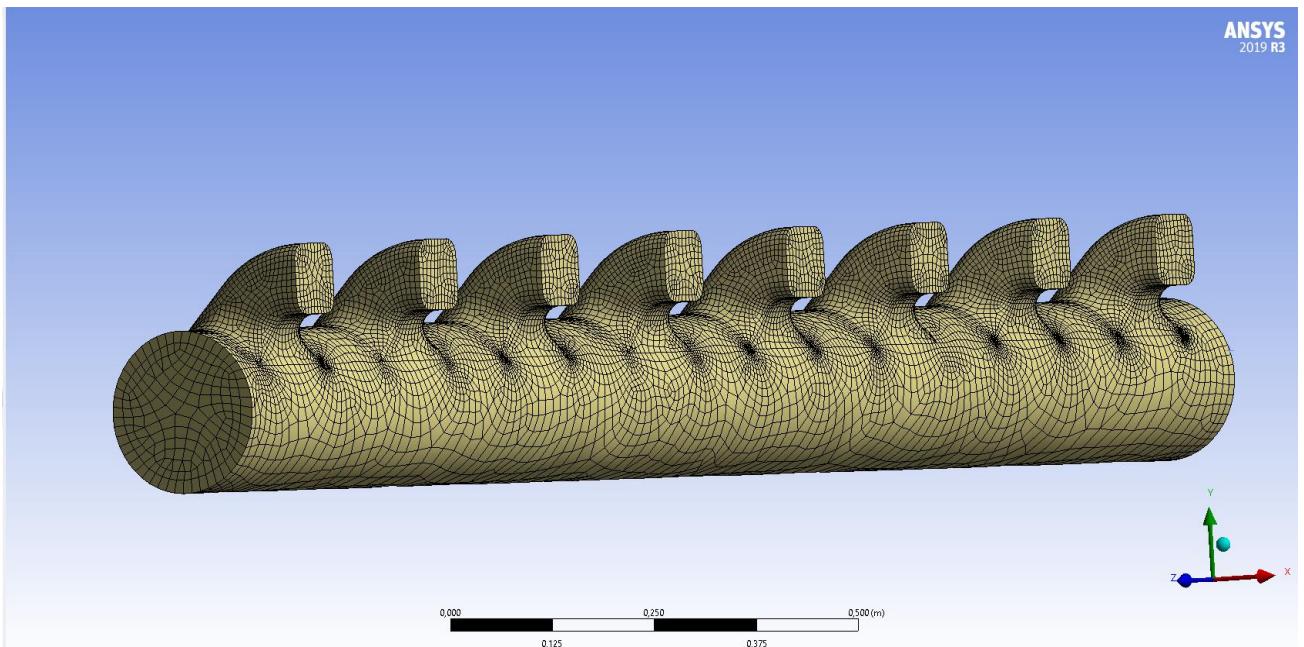


Figure 7.9: Unstructured Pave Grid

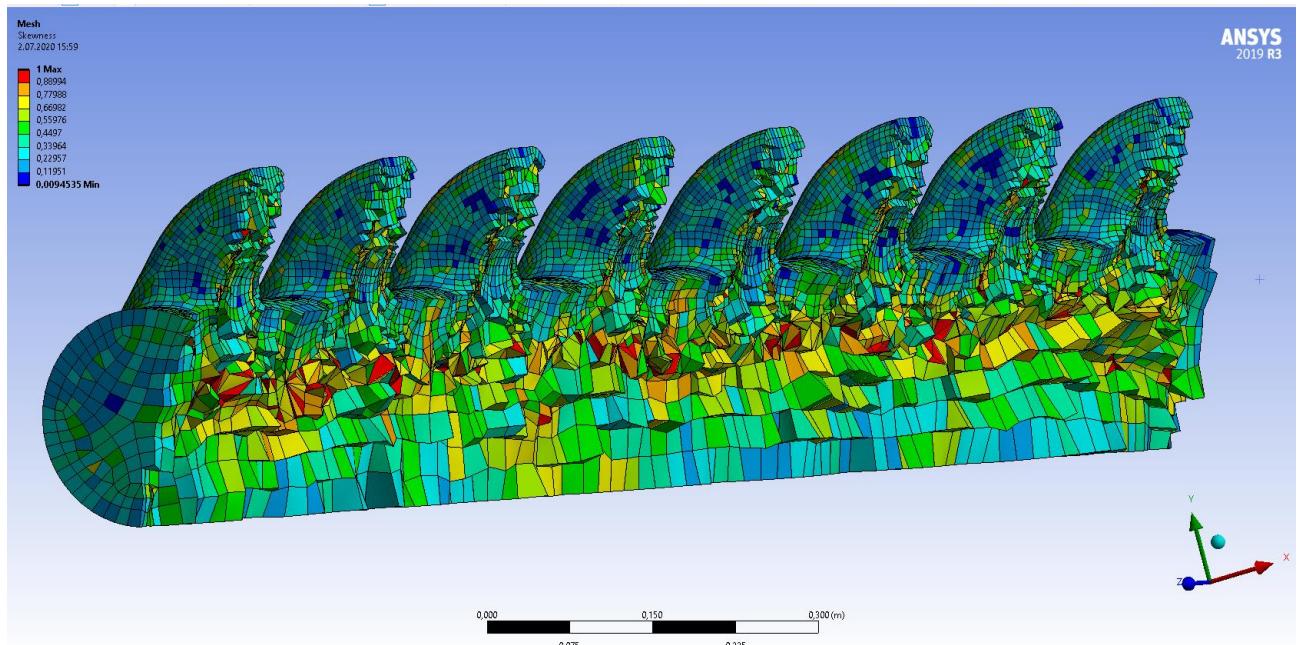


Figure 7.10: Elements coloured according to their skewness in the cross-section of the manifold

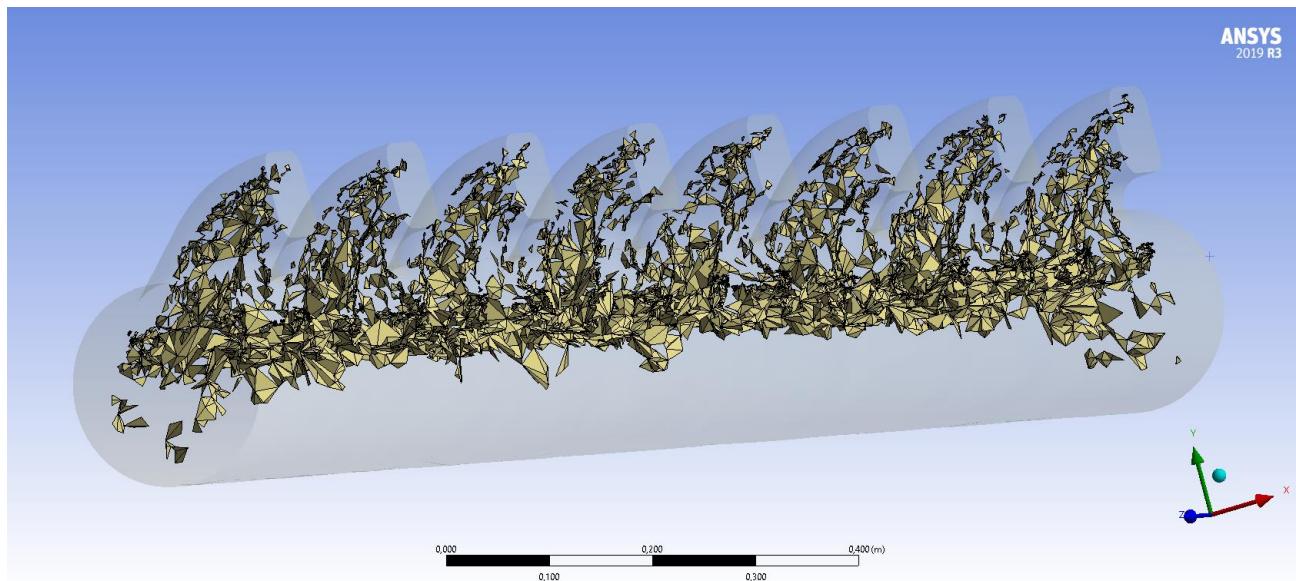


Figure 7.11: Highly skewed elements

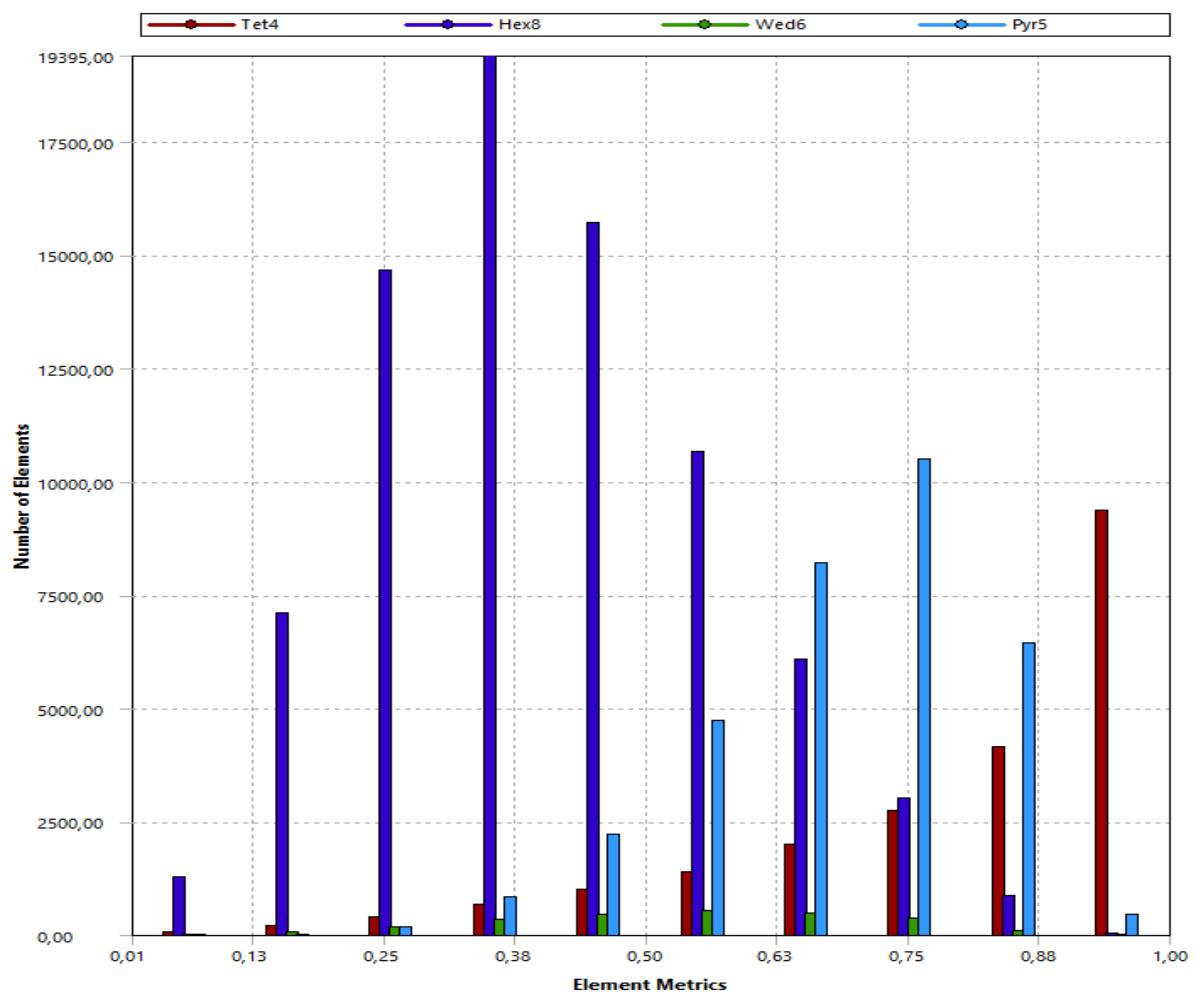


Figure 7.12: Number of elements versus their Skewness value

7.4.Improved Unstructured Pave Grid

The previous pave grid is refined and the maximum element size is decreased. Global element size is reduced to 0.008 mm and the maximum element size reduced to 0.016mm. This mesh takes a lot of time to build but it has much better overall element quality.

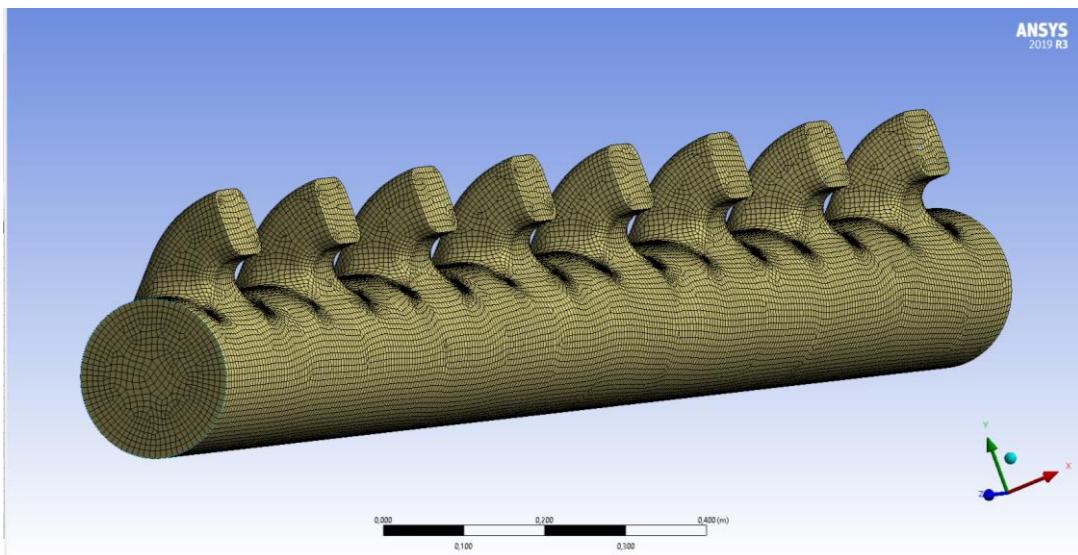


Figure 7.13. Flow Volume of the Air with Hexahedral Dominant Meshing Inside the Intake Manifold

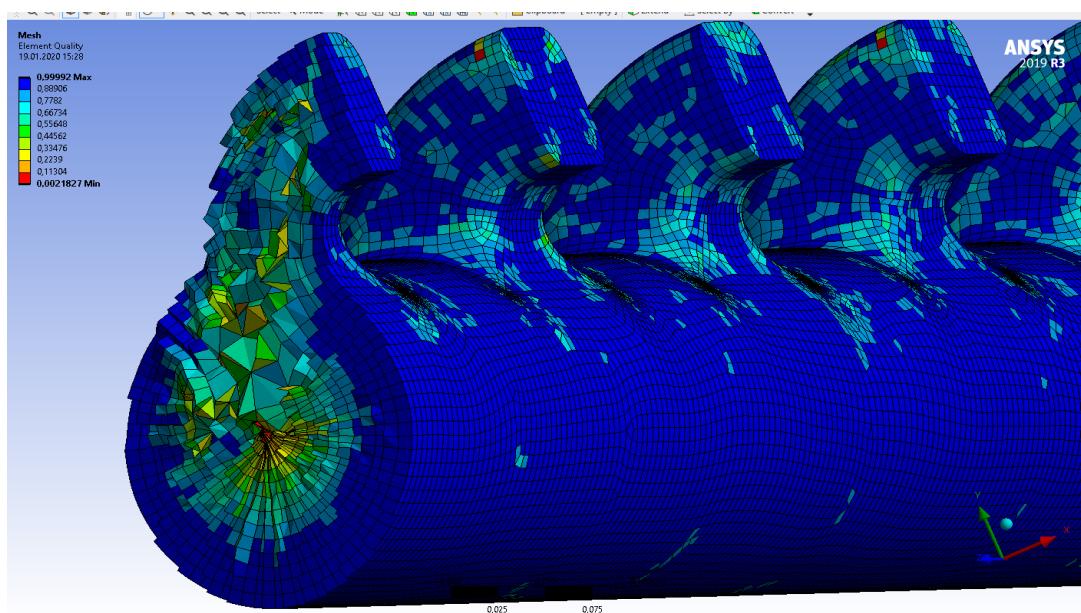


Figure 7.14. Whole Elements Coloured According To Their Quality

7.5.Rectangular H-grid

Rectangular H grid also analysed for the geometry. Rectangular grid is not recommended in the literature due to bad element qualities in the denser areas. This analysis shows similar results.

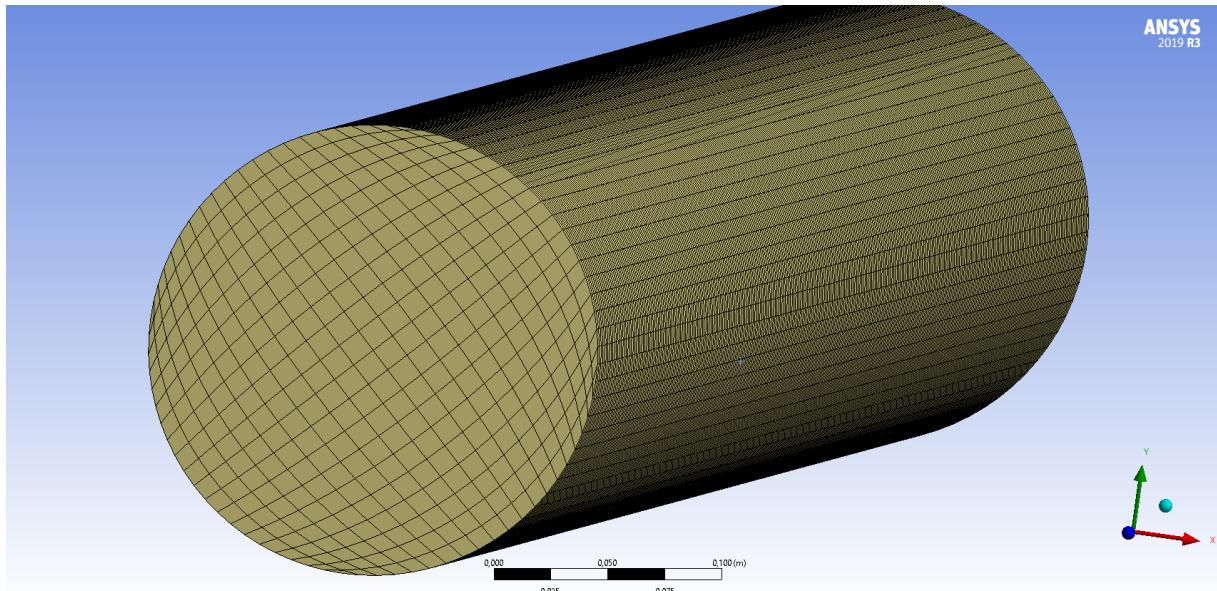


Figure 7.15: Rectangular Grid at the Circular Area of the Flow Volume

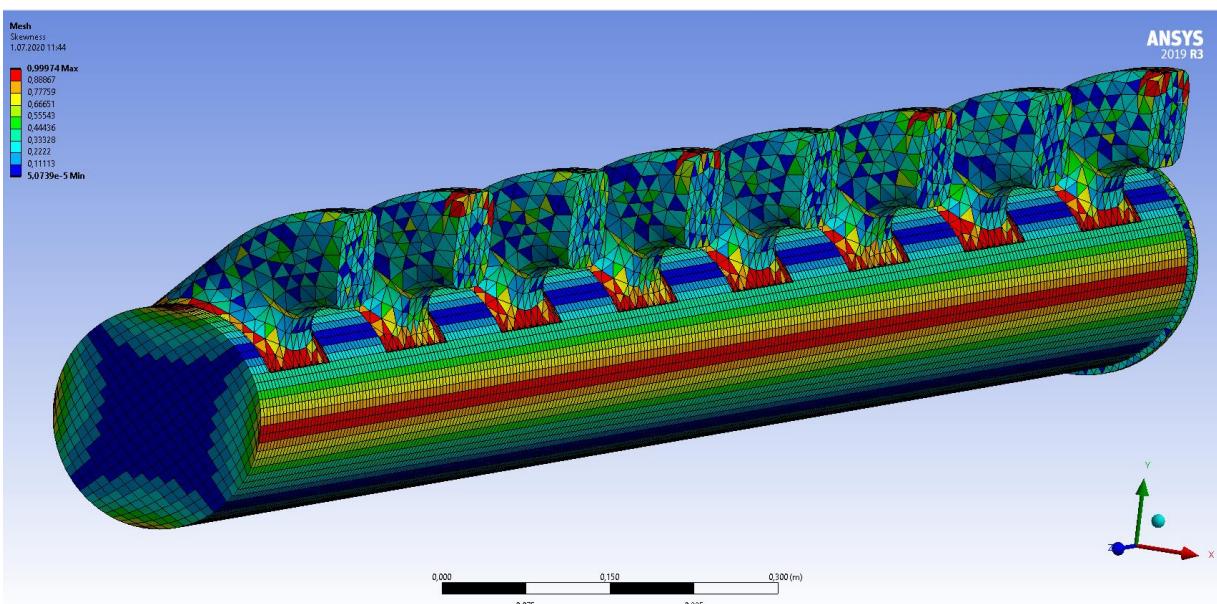


Figure 7.16: Elements Coloured According To Their Skewness

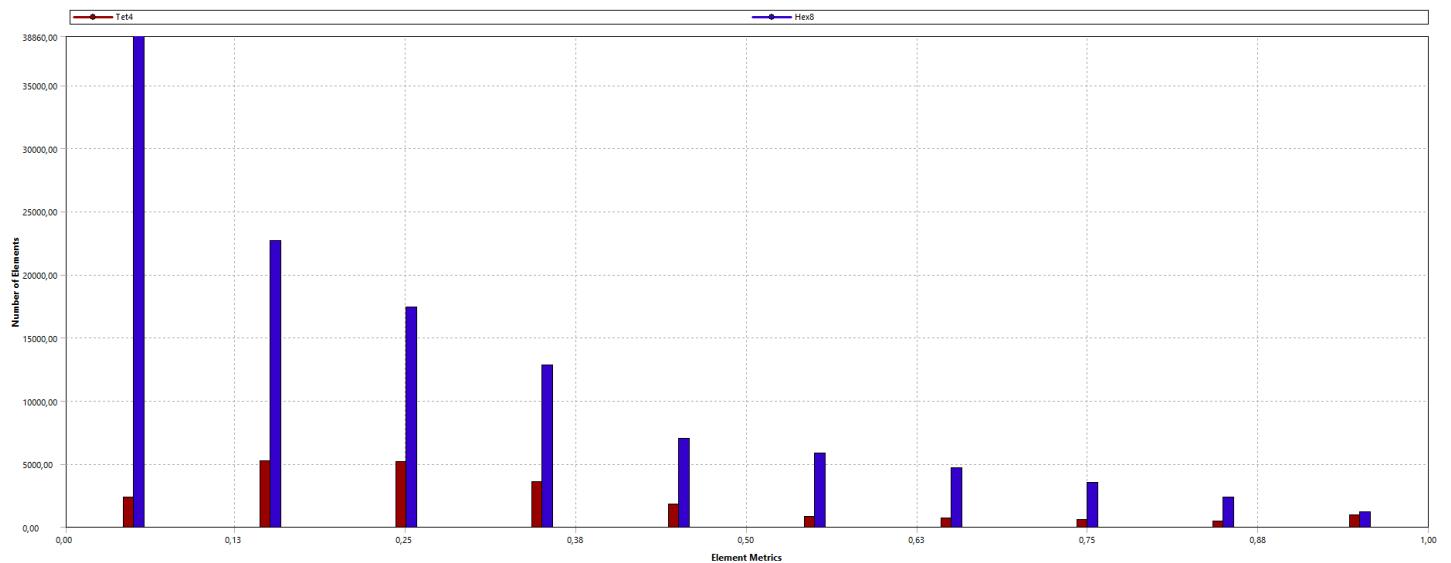


Figure 7.17: Number of Elements versus Skewness

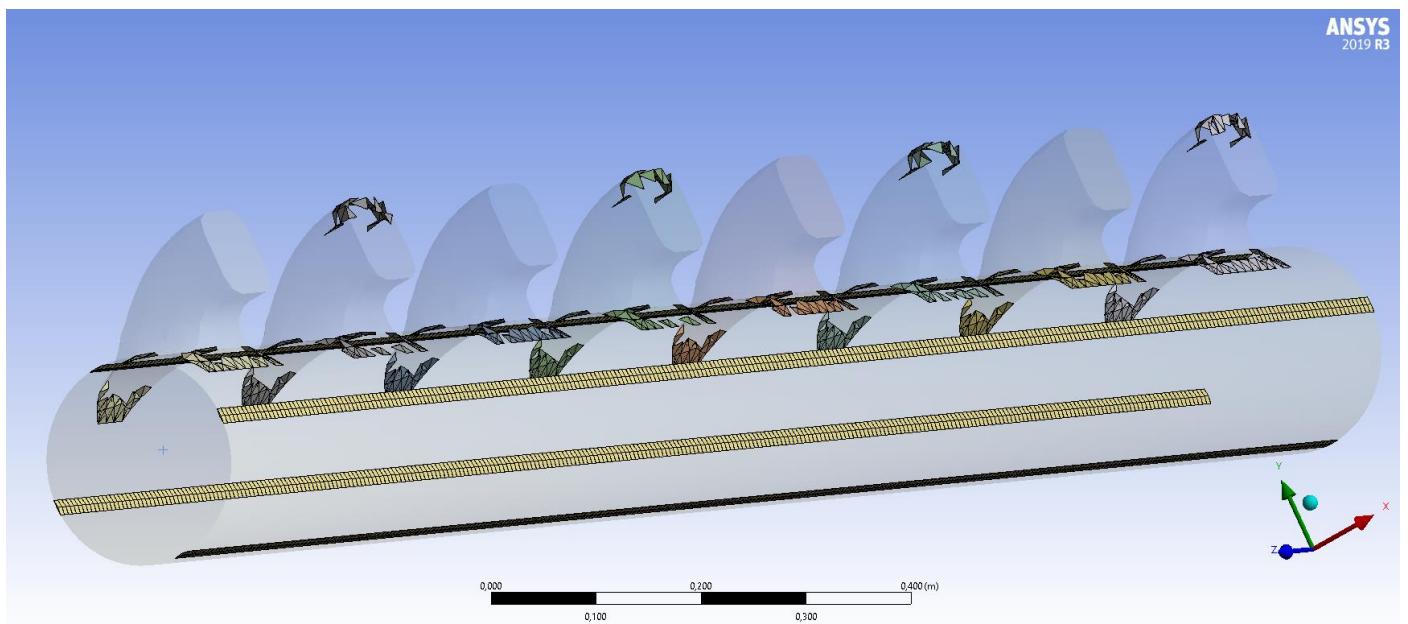


Figure 7.18: Tetrahedral and Hexahedral elements with Skewness value close to 1

Number of elements in the Flow Volume: 137296

Number of Nodes in the flow volume: 133827

7.6. Butterfly Grid

In ANSYS Meshing, structured mesh type was generated in simple geometry before testing on original geometry. Intake manifold is a complex geometry due to curves and disorganized faces. Simpler geometries are needed good quality meshes. Arranging the body is a significant point for decreasing solver calculation time.

Structured Mesh type was used for more precise solution at critical edges or faces. Creating structured mesh takes more time for meshing compared to unstructured meshes. Geometries which have various fillets and complex curves are needed to be simplified. However structured meshes are more accurate when applied correctly. Butterfly mesh is one of the most common structured mesh types. This model contains rectangular or square geometries in the cylinder back and front faces. This situation allows a different mesh density in the centre and a different mesh density at the outer surface of the cylinder. Two main criteria are researched to compare element quality. These criteria are skewness and aspect ratio, orthogonality. There are 33156 nodes and 30800 elements in the Butterfly mesh model.

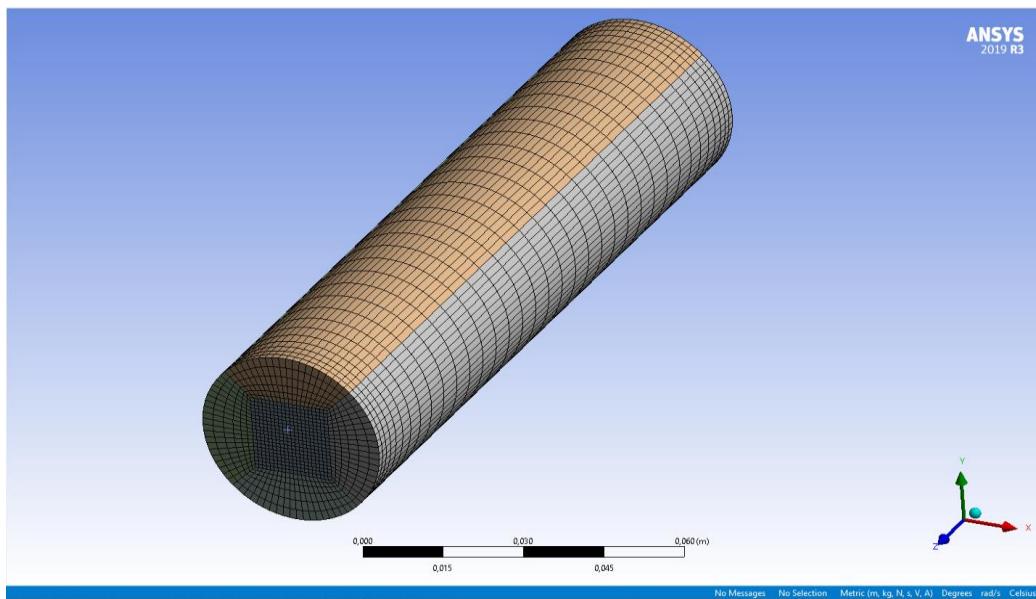


Figure 7.19: General view of model Butterfly mesh.

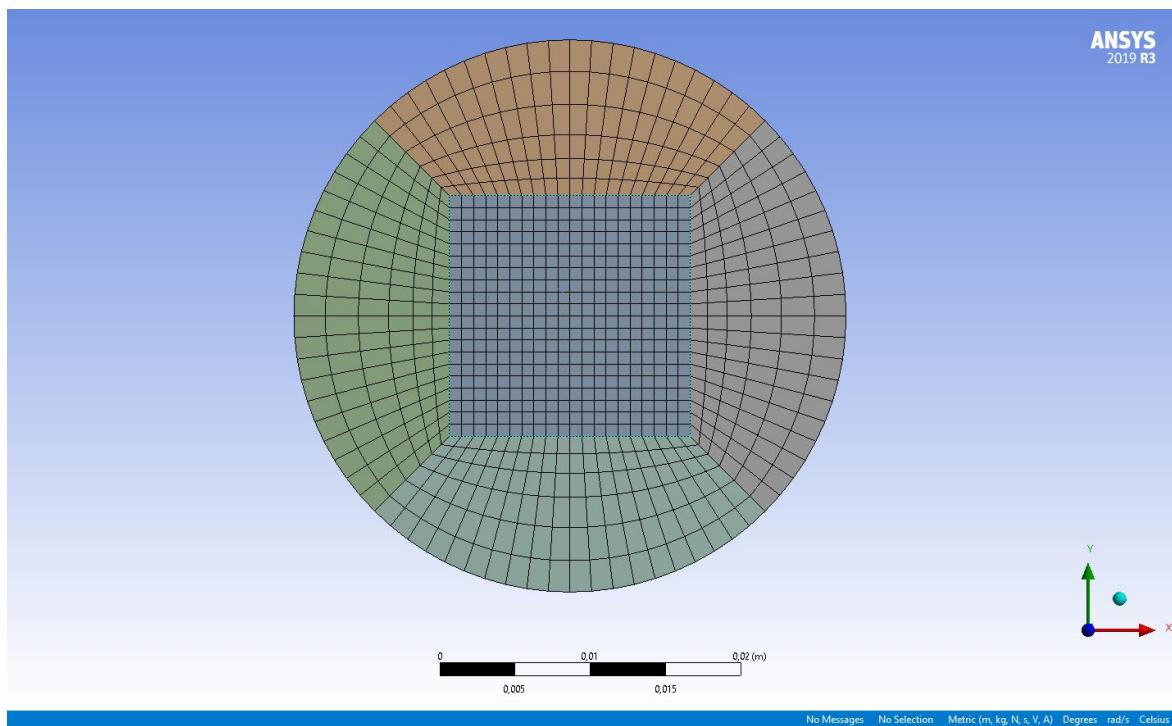


Figure 7.20: Front Plane of Butterfly Mesh model of Cylinder.

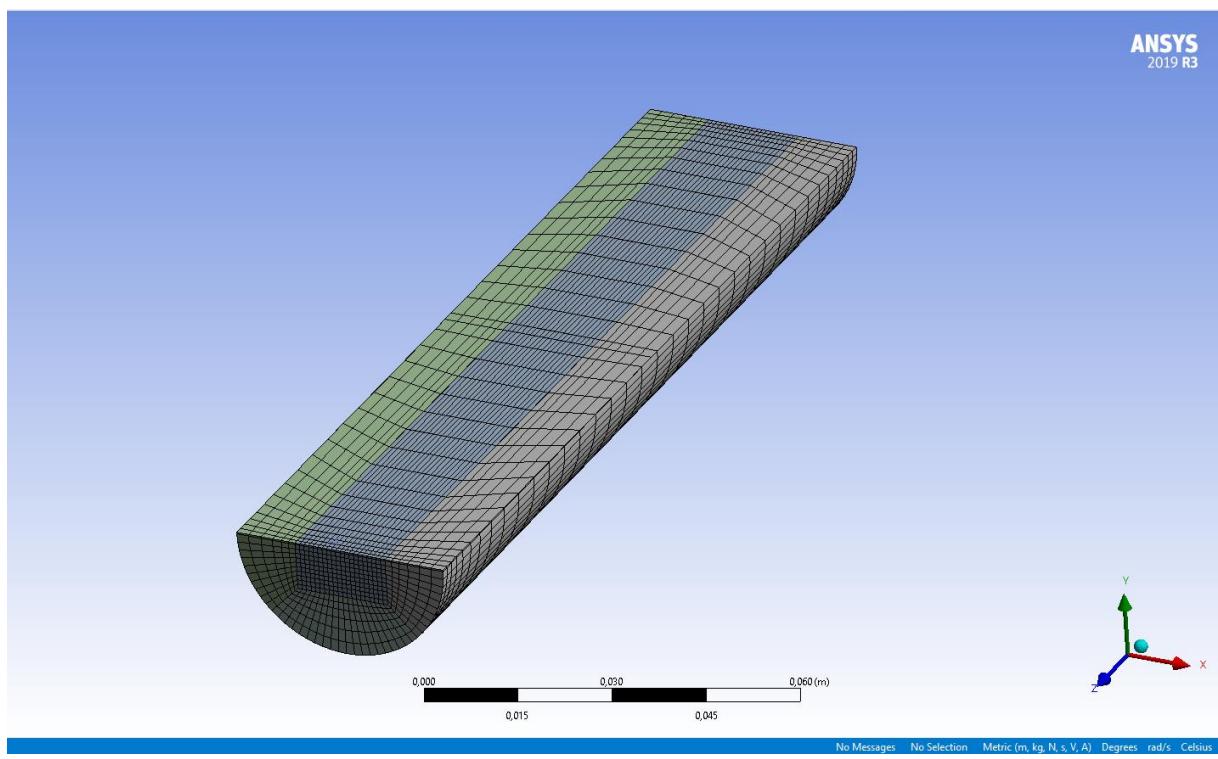


Figure 7.21: Lateral section view of Butterfly mesh

Element quality of Butterfly mesh is different at the outer and inner side of the geometry. At the outer faces, element quality is good enough for an accurate solution but the element quality decreases at the inner side of the cylinder due to the increased aspect ratio. However, this problem can be solved by decreasing the length of the element in the z axis.

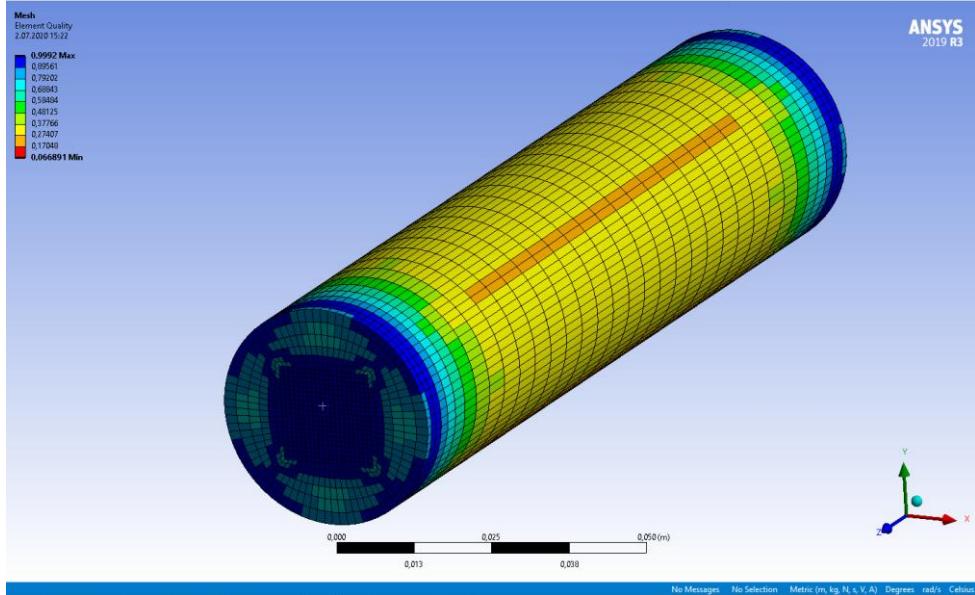


Figure 7.22: Element quality of Butterfly mesh.

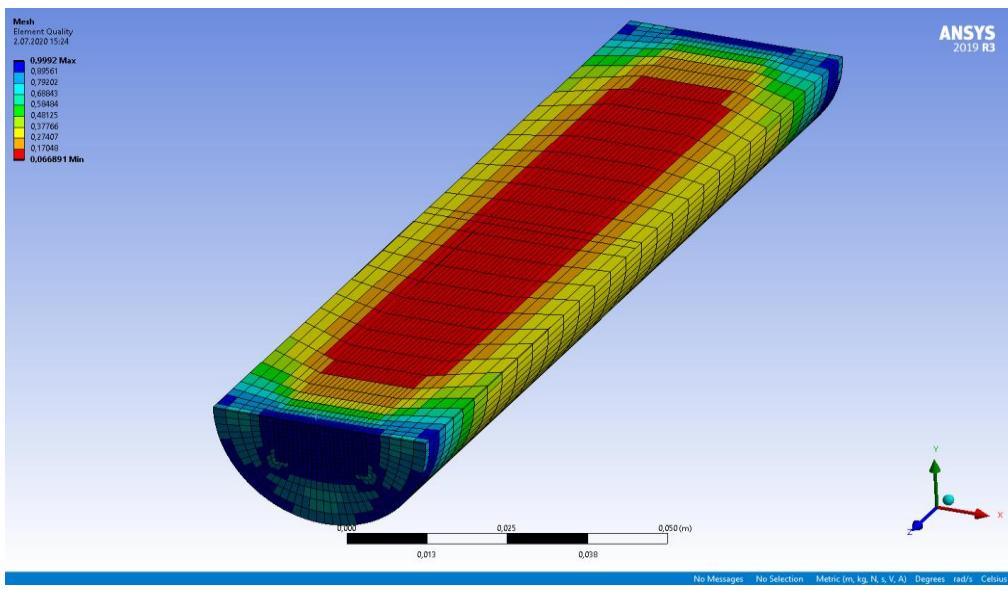


Figure 7.23: Element quality of inside of Butterfly mesh.

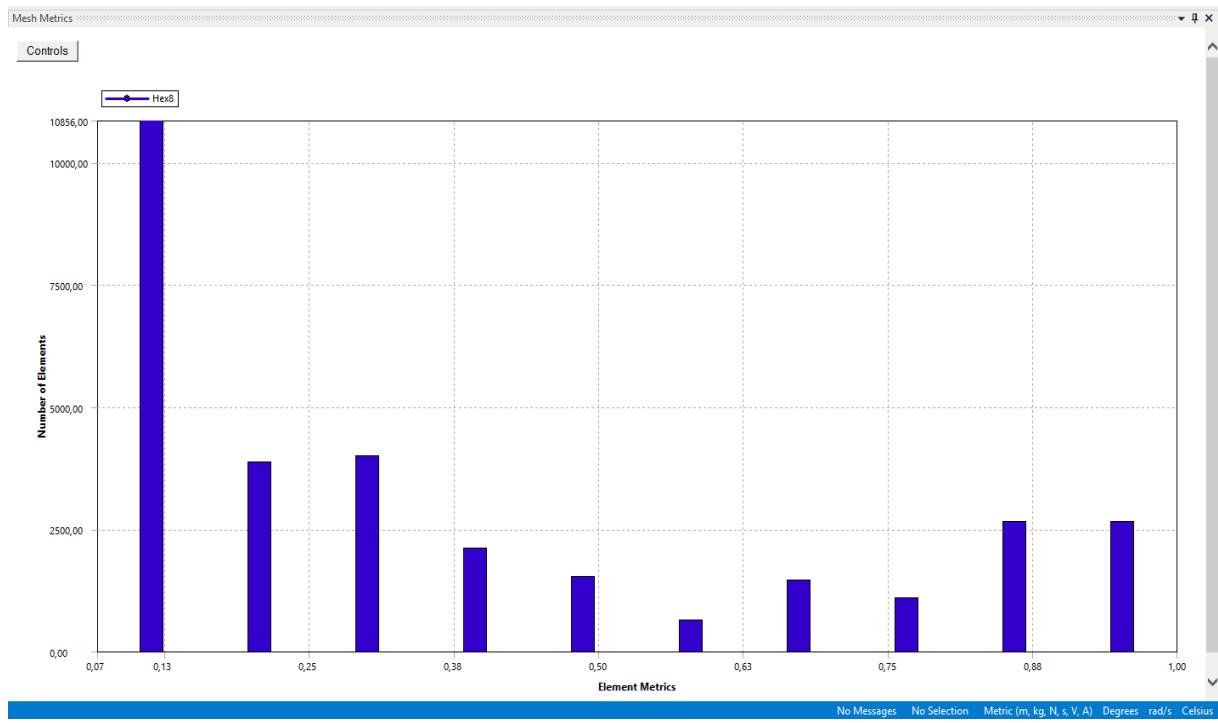


Figure 7.24: Distribution of Butterfly mesh element quality

Most of the elements seem to have bad quality because of high aspect ratio. The quality of the elements must be improved before any simulation attempt.

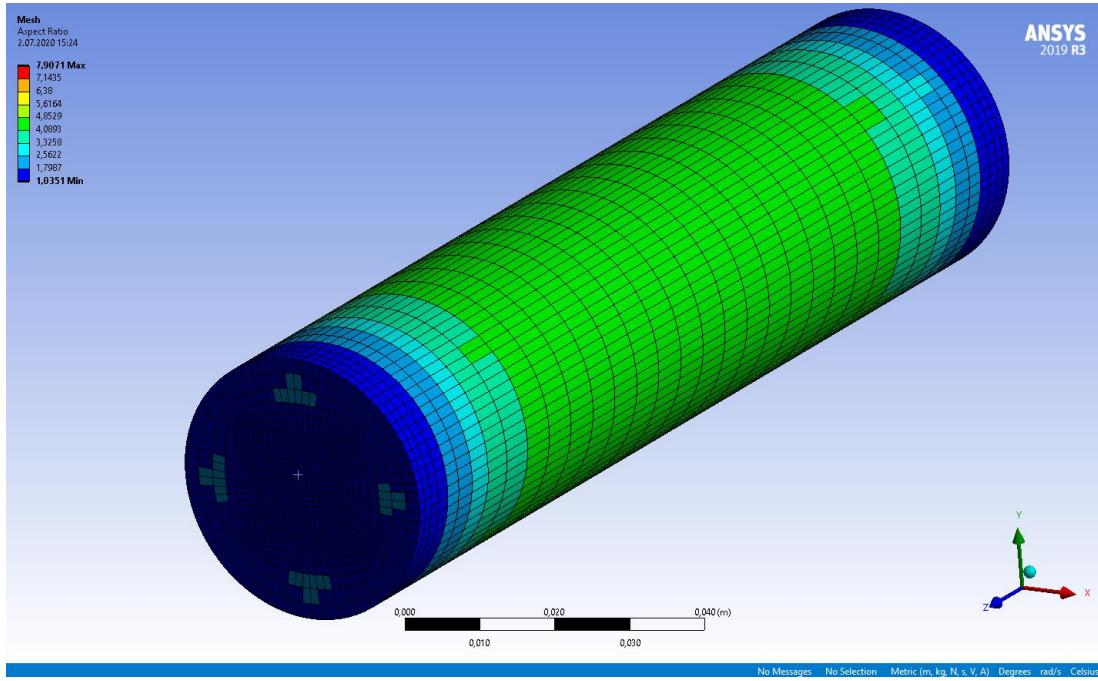


Figure 7.25: Aspect ratio of Butterfly mesh.

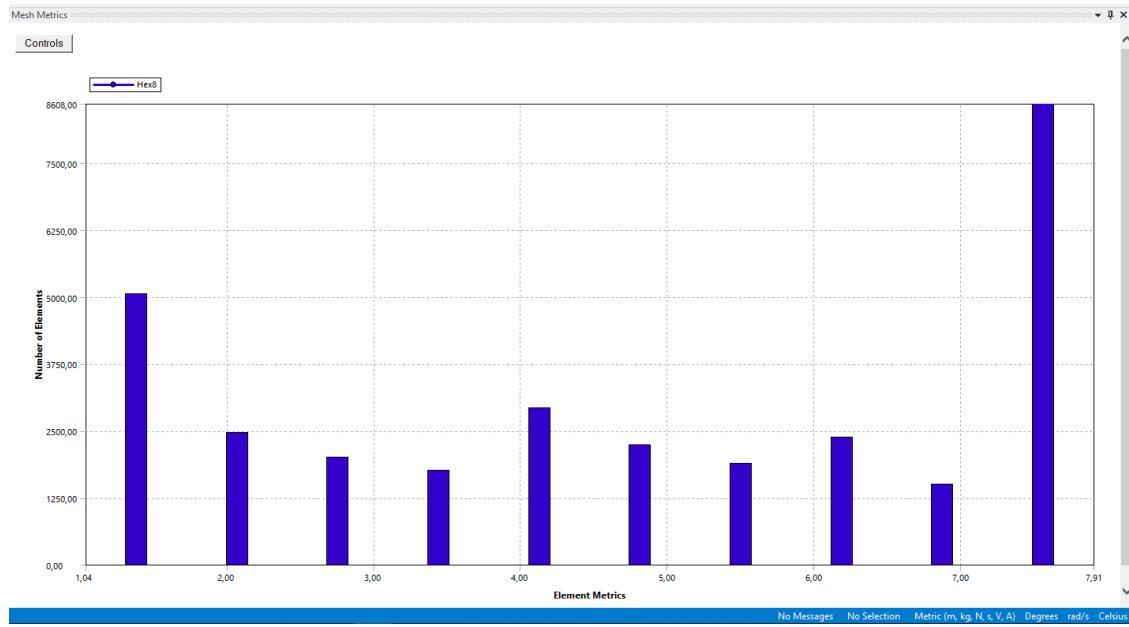


Figure 7.26: Distribution of Aspect Ratio in the Butterfly Mesh

Skewness have good distribution at the Butterfly mesh. Nearly half of elements' skewness so close to zero. Structured mesh occurs more reliable skewness for the cylinder shape. Centre elements of the cylinder have low skewness but high aspect ratio, so the element quality is decreased. Convergence is highly affected by skewness. In the case of high skewness, convergence difficulties are occurred and it may require the change of the solver controls, such as reducing under-relaxation factors and/or switching to the pressure-based coupled solver.

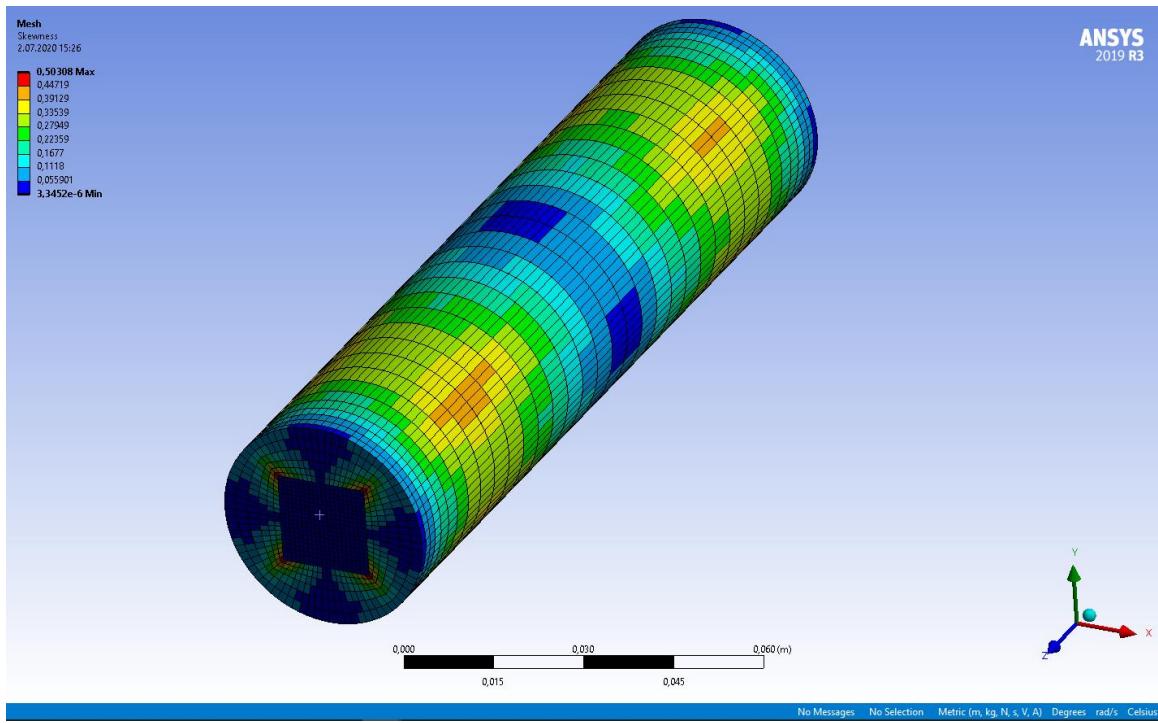


Figure 7.27: Skewness of Butterfly mesh.

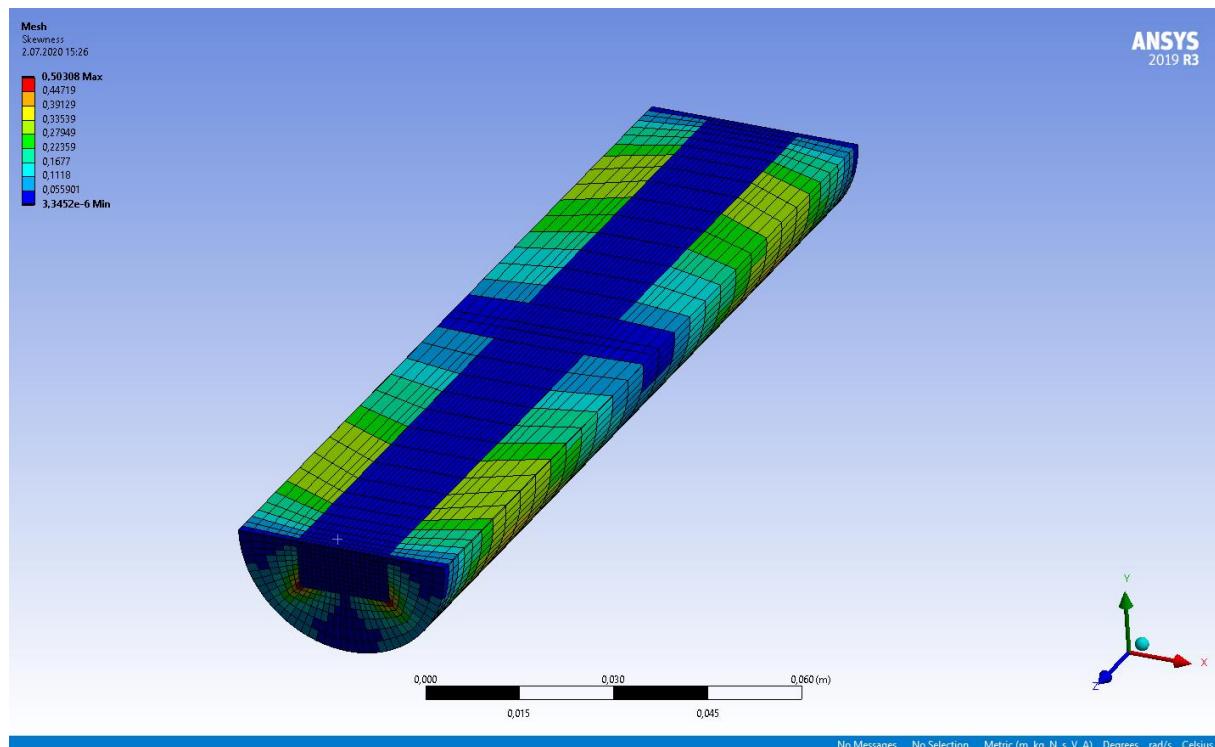


Figure 7.28: Lateral view Skewness for Butterfly mesh.

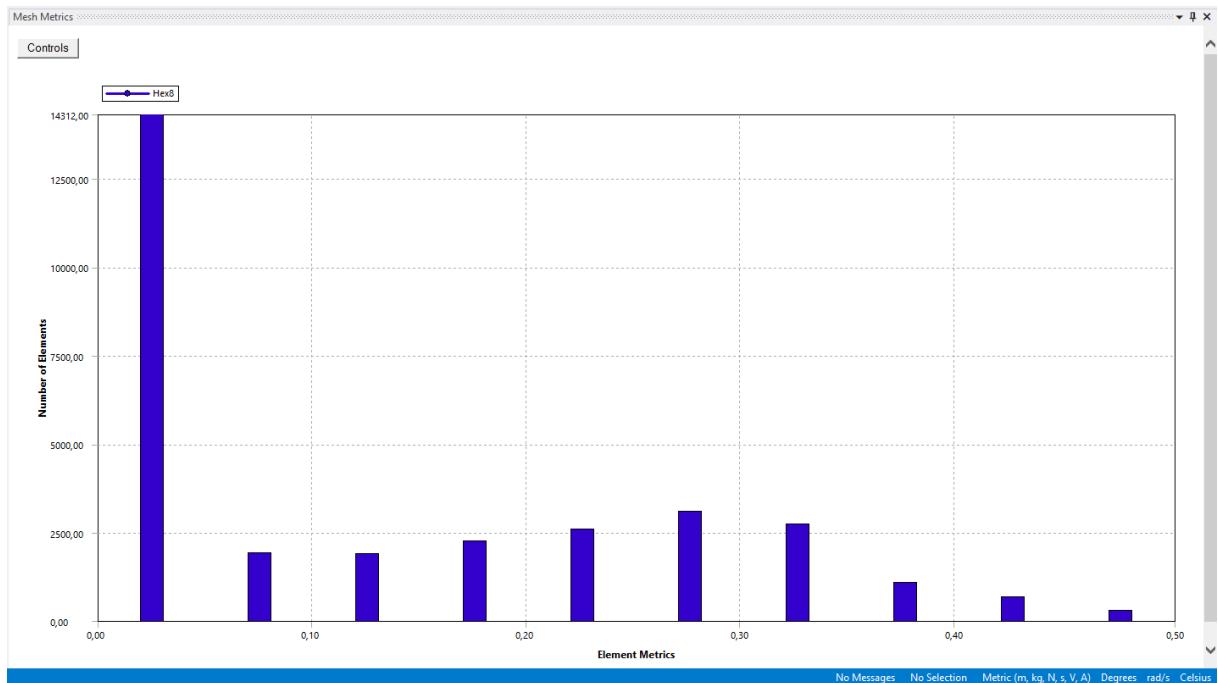


Figure 7.29: Distribution of Skewness ratio

7.7. Butterfly Grid in Salome

Salome can also generate the butterfly grid. The grid is generated for a pipe with 100mm length and 10mm radius. This mesh has only hexahedrons, exactly 40,000 of them. The mesh has 42861 total nodes.

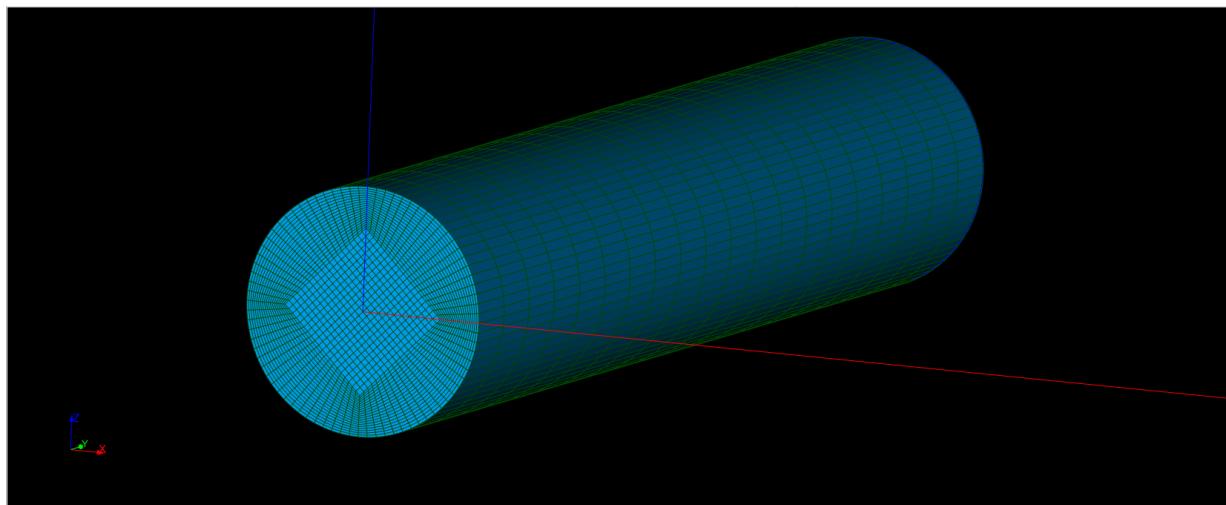


Figure 7.30: Butterfly mesh in Salom

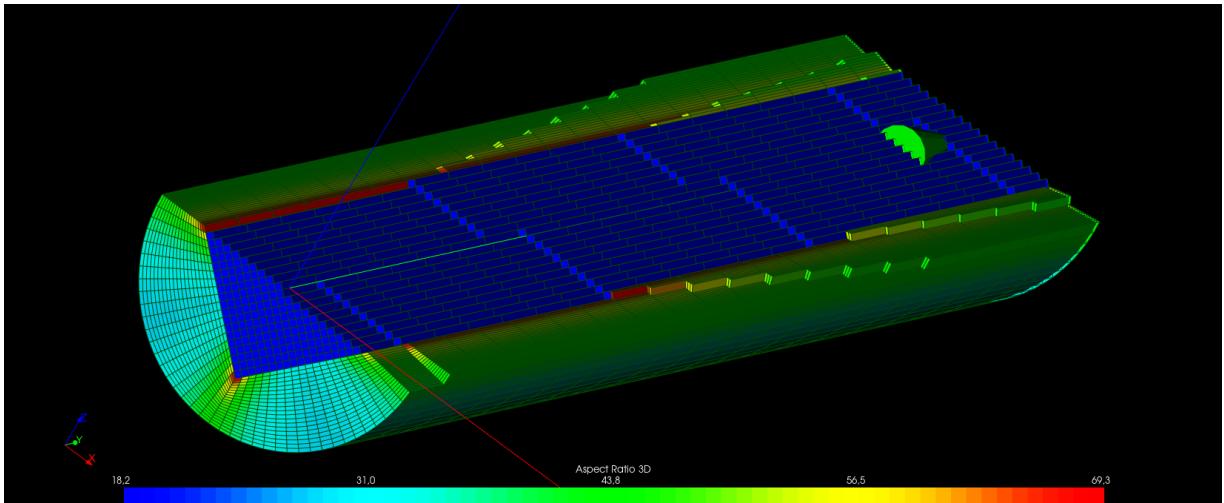


Figure 7.31: Elements coloured according to their aspect ratio

From the Figure 7.31, it can be seen that overall aspect ratio of the elements are low. There are some elements with higher aspect ratios in denser areas, but they make a small percentage of all the elements. This mesh is ideal for flow simulations in pipes.

8. RESULTS AND DISCUSSION

The results are obtained for various meshes mentioned in the section 7. k- ϵ model is used for solution and Fluent's SIMPLE algorithm is used. Geometries were taken in SCDOC format. The conditions at the inlet of the manifold is 30 m/s for air and 300K. Outlets are specified as pressure outlets. The intake manifold is made of aluminium.

8.1. Results for Fully Automated Unstructured Tetra Mesh:

Figure 1 and Figure 2 shows the velocity contour and velocity streamlines in the flow domain, respectively. Here the maximum air velocity is 40 m/s and the minimum velocity is 0.136 m/s. It can be seen that especially at the end of the manifold, there is some incorrect behaviour of air. This is due to general element quality throughout the mesh, which is not very high in this case. In figure 3, pressure contour of the air is shown. There are pressure drops on the side of the elbows where air passes by, and pressure increase on the sides where air strikes. Also there is a pressure increase at the end of the manifold which is logical. However, these results does not represent real and accurate flow due to mediocre/coarse element quality. For this solution, the average wall-clock

time per iteration is 1.499 seconds. For 250 iterations, it takes a total wall-clock time of 374.631 seconds.

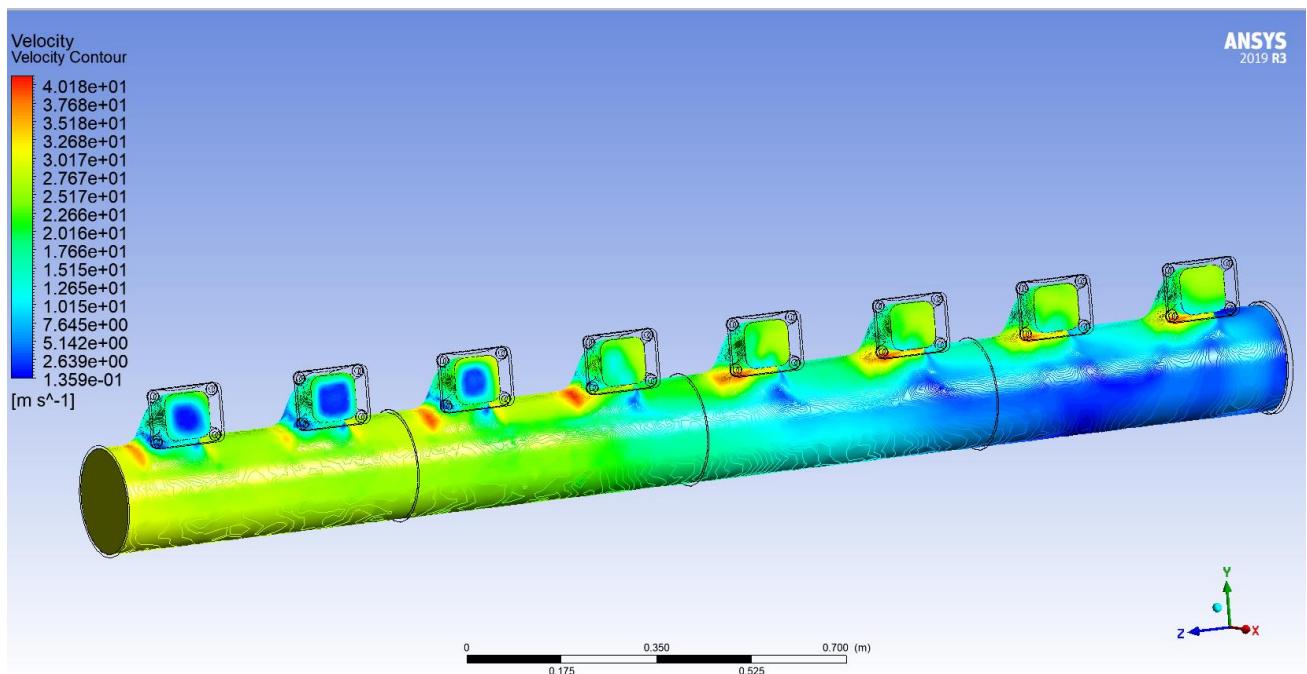


Figure 8.1: Velocity Contour of the air

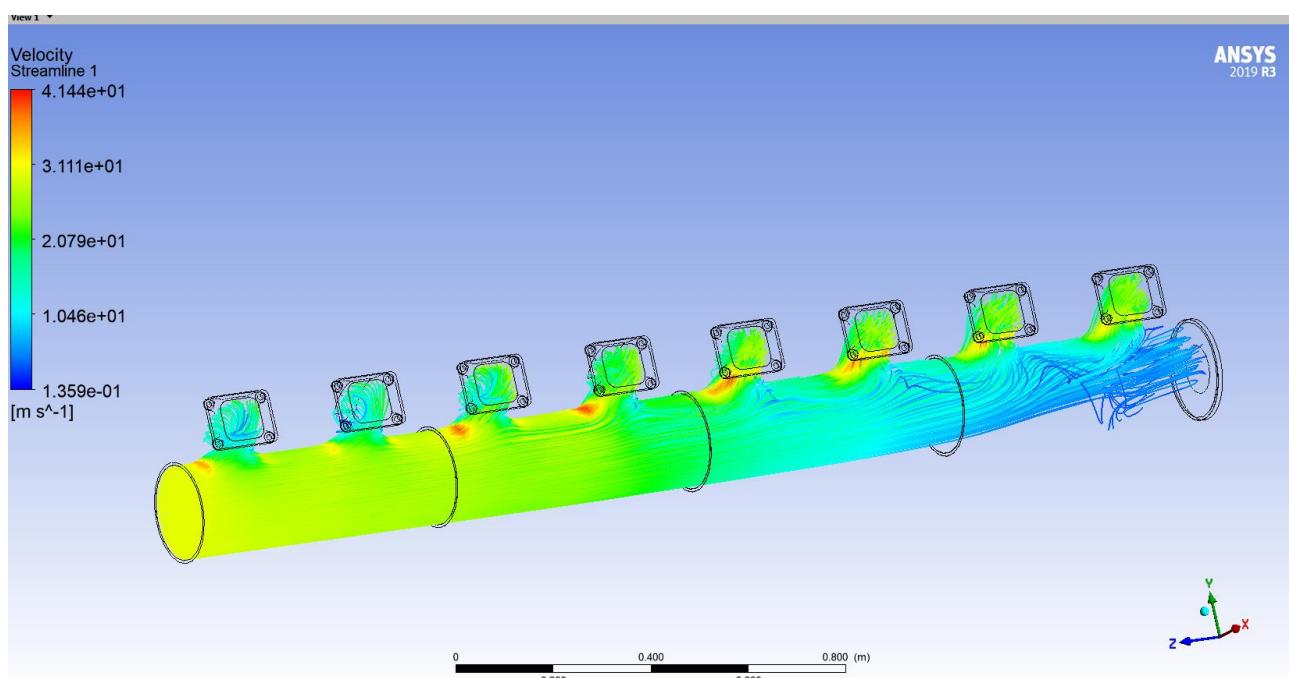


Figure 8.2: Velocity Streamlines of the air

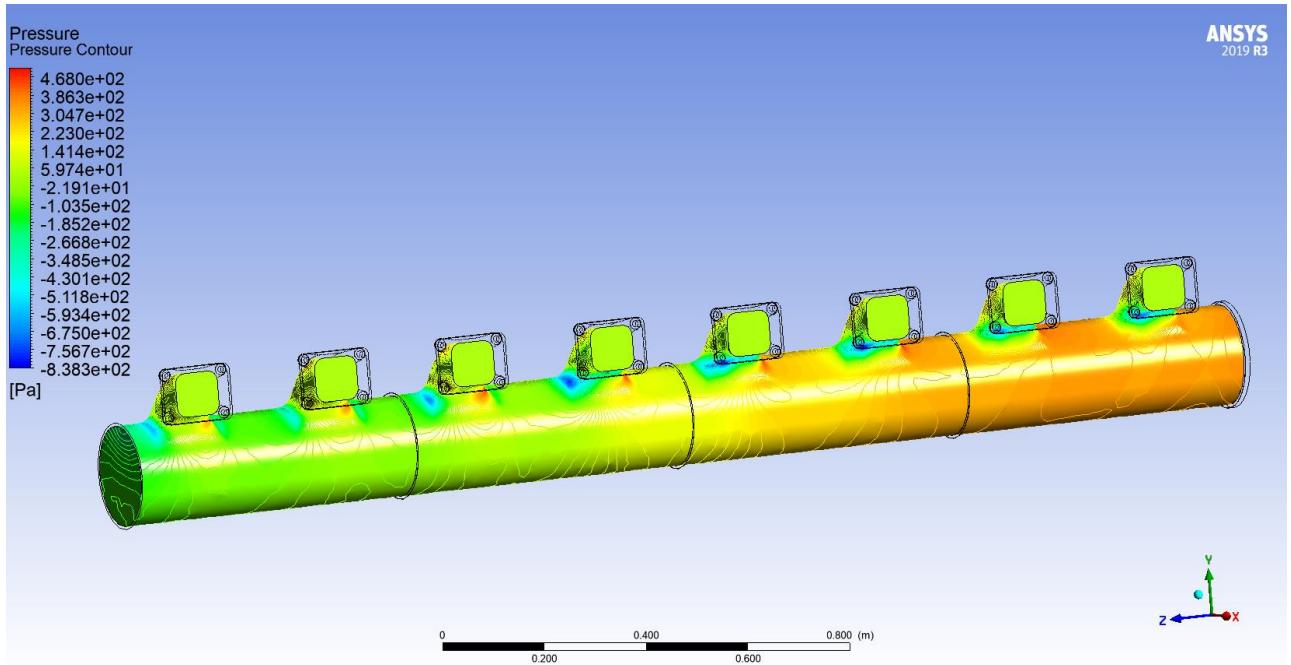


Figure 8.3: Pressure Contour of the air

8.2. Results for Improved Tetra Mesh

Manually improved tetra mesh results in much more satisfying results than the fully automated tetra grid. Although only simple improvements are made, better results are obtained instantly. Higher smoothing option in the program and inflation operation increases the mesh building time and computational expense but better results can be obtained with the sacrifice of small amount of time. In figure 4 and figure 5, velocity contours and velocity streamlines of the air can be seen respectively. The streamlines are much better in this mesh compared to fully automated mesh, but there are still some fluctuations of speed in certain areas, which realizes a non-smooth flow along the z axis of the manifold. The maximum air velocity is calculated to be 36.8 m/s, and the minimum velocity is 0.08 m/s. Figure 6 shows the pressure inside the manifold. The pressure increases along the flow direction in the z axis, as expected. Outlets are taken to be at 0 Pascal. A pressure drop on left side the elbows can be seen, while the pressure increases on the right side, where air strikes. The average wall-clock time per iteration is 1.802 seconds and total wall-clock time for 250 is found to be 450.493 seconds. This is just

around 75 seconds more than the previous solution, but the results are significantly improved.

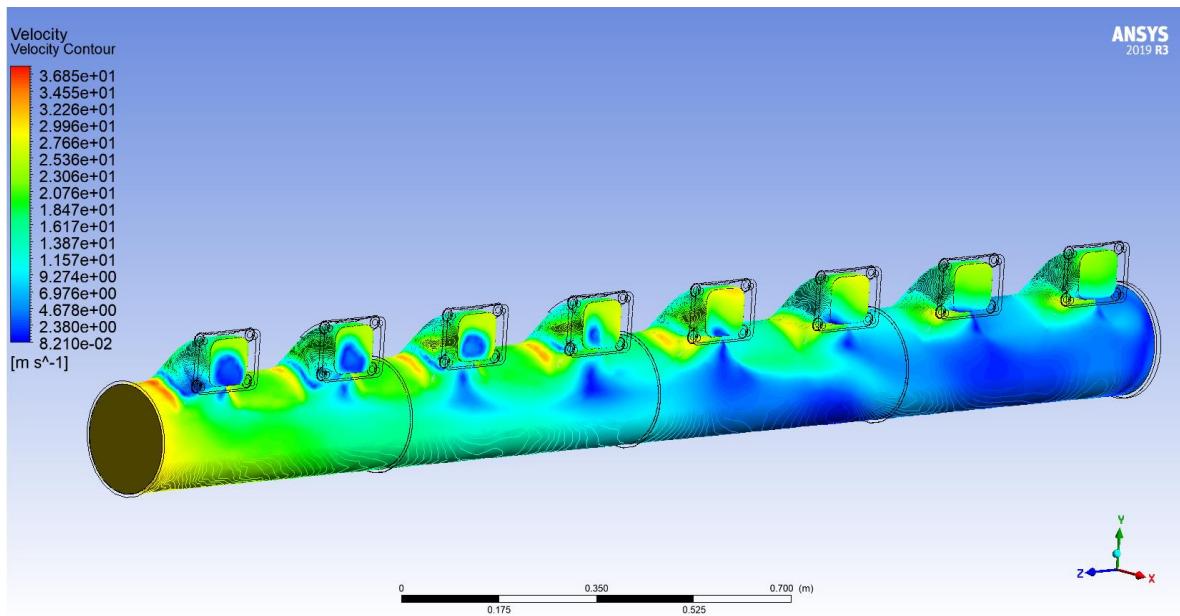


Figure 8.4: Velocity Contour of the air

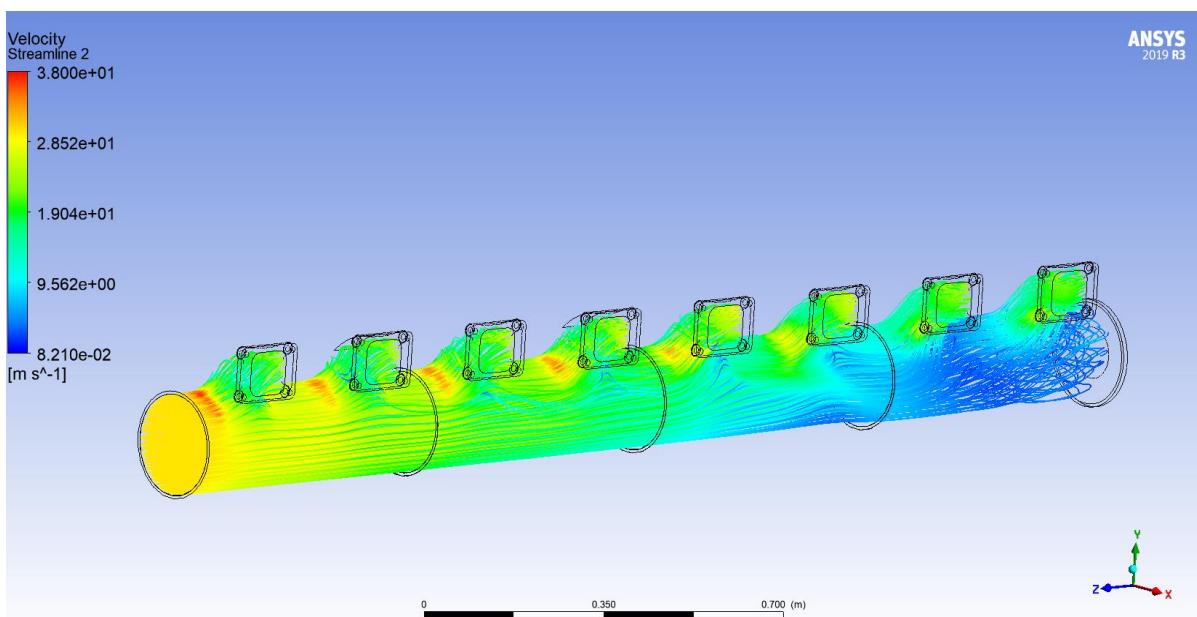


Figure 8.5: Velocity Streamlines of the air

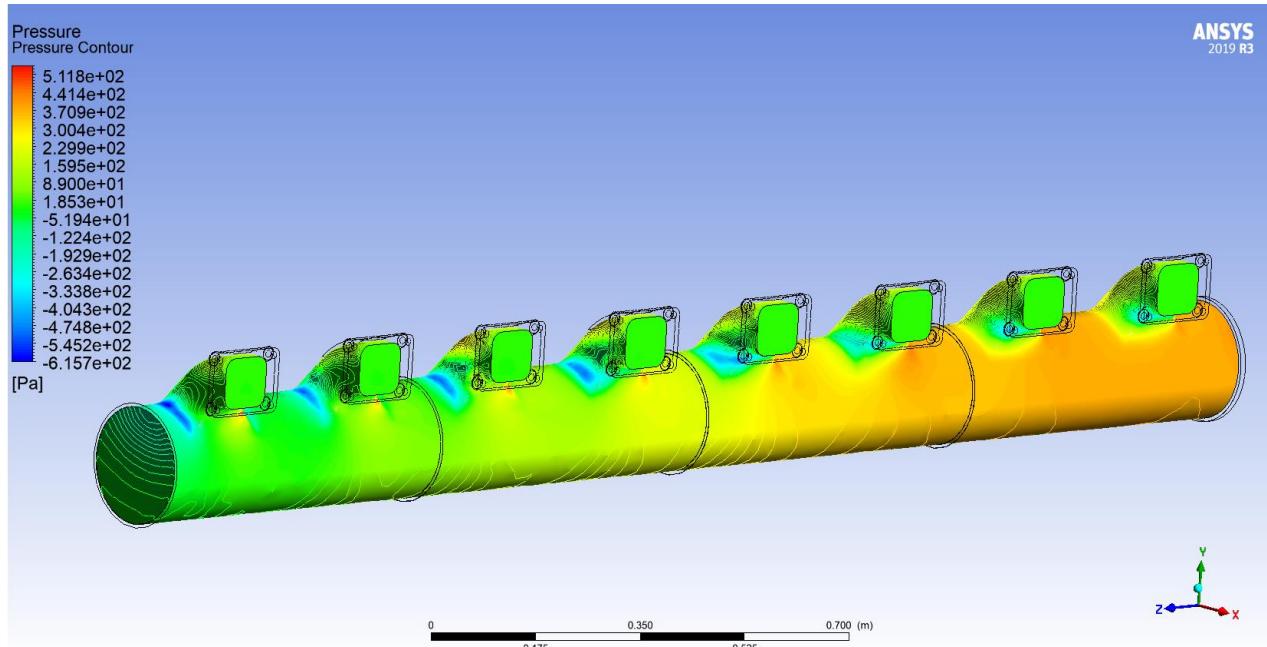


Figure 8.6: Pressure Contour of the air

8.3. Results for Unstructured Pave Grid

The unstructured pave grid was generated with very low element density, thus the solution converged extremely quickly, two times faster than the tetra mesh (see Table1). Despite the fast solution and low element density, building time of the mesh is still greater than tetra meshes due to the complexity of the elements. Figure 7 shows similar results with tetra meshes but from figure 8 it can be seen that the rightmost elements of the mesh are not taken into account. The maximum air velocity is found to be 33.2 m/s and the minimum air velocity is found 0.074 m/s.

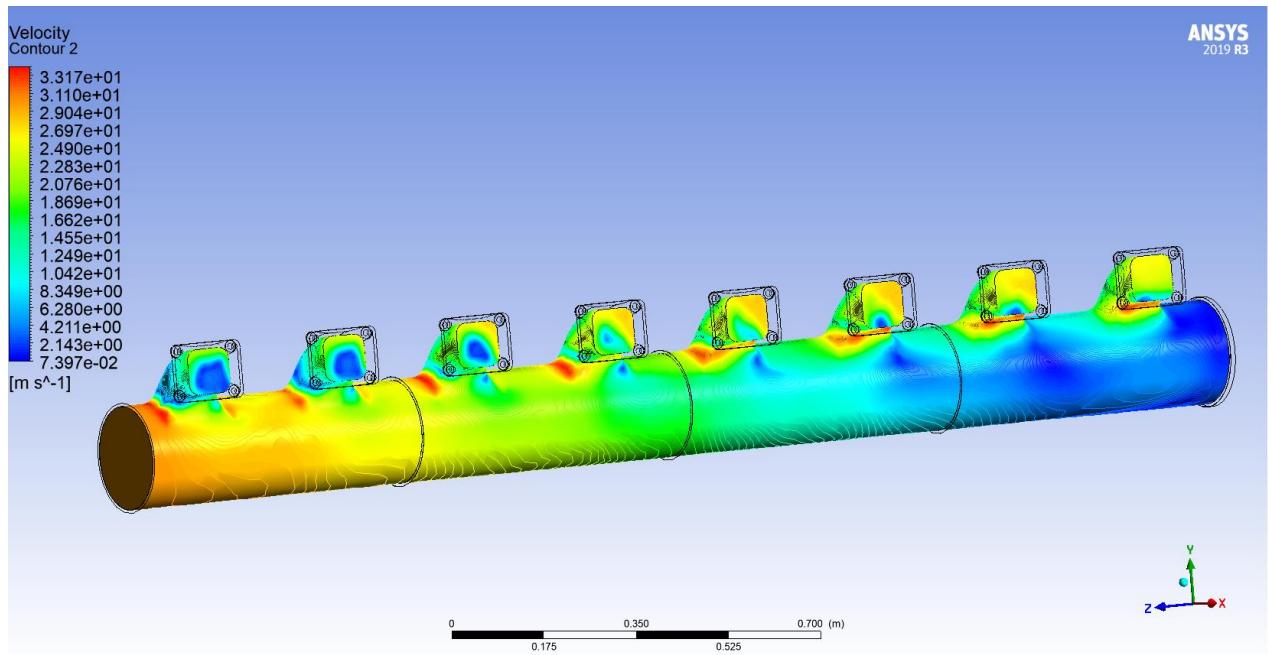


Figure 8.7: Velocity Contour of the air

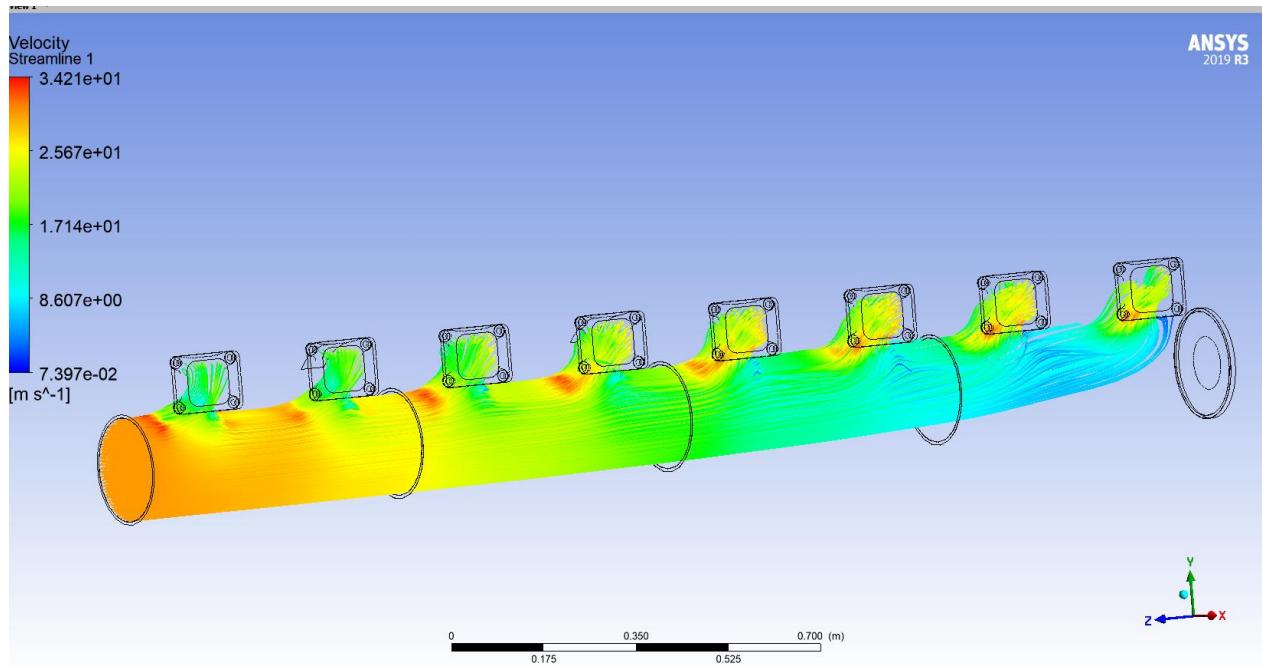


Figure 8.8: Velocity Streamlines of the air

8.4.Results for Improved Unstructured Pave Grid

The unstructured pave grid was expected to give the best results, according to the findings in the literature. This mesh is a hybrid mesh consisting of both hexahedral and tetrahedral elements. Unstructured pave grids are very efficient compared to structured grids and they can be applied in geometries where structured cannot be applied. Correctly aligned hexahedral elements give the best results for CFD analyses in general, and also in this case. In this improved unstructured pave grid, global element size is reduced to 0.008 mm in this grid and high smoothing is selected. High element density is achieved by reducing the sizes of the elements. Figure 9 shows the velocity streamlines in flow domain. This figure shows a much smoother flow than the previous solutions. The maximum air velocity is observed to be 35 m/s and the minimum air velocity is 0 m/s in this solution. The average wall-clock time per iteration is 2.601 seconds and the total wall-clock time for 250 iterations is 650.25 seconds. The average wall-clock time is increased 1.44 times compared to improved tetra mesh. Also total amount of time to build the mesh increases a lot compared to the previous meshes. This increase in time is caused by higher density of elements and the complexity of hexahedrons. The most realistic results are obtained with this mesh.

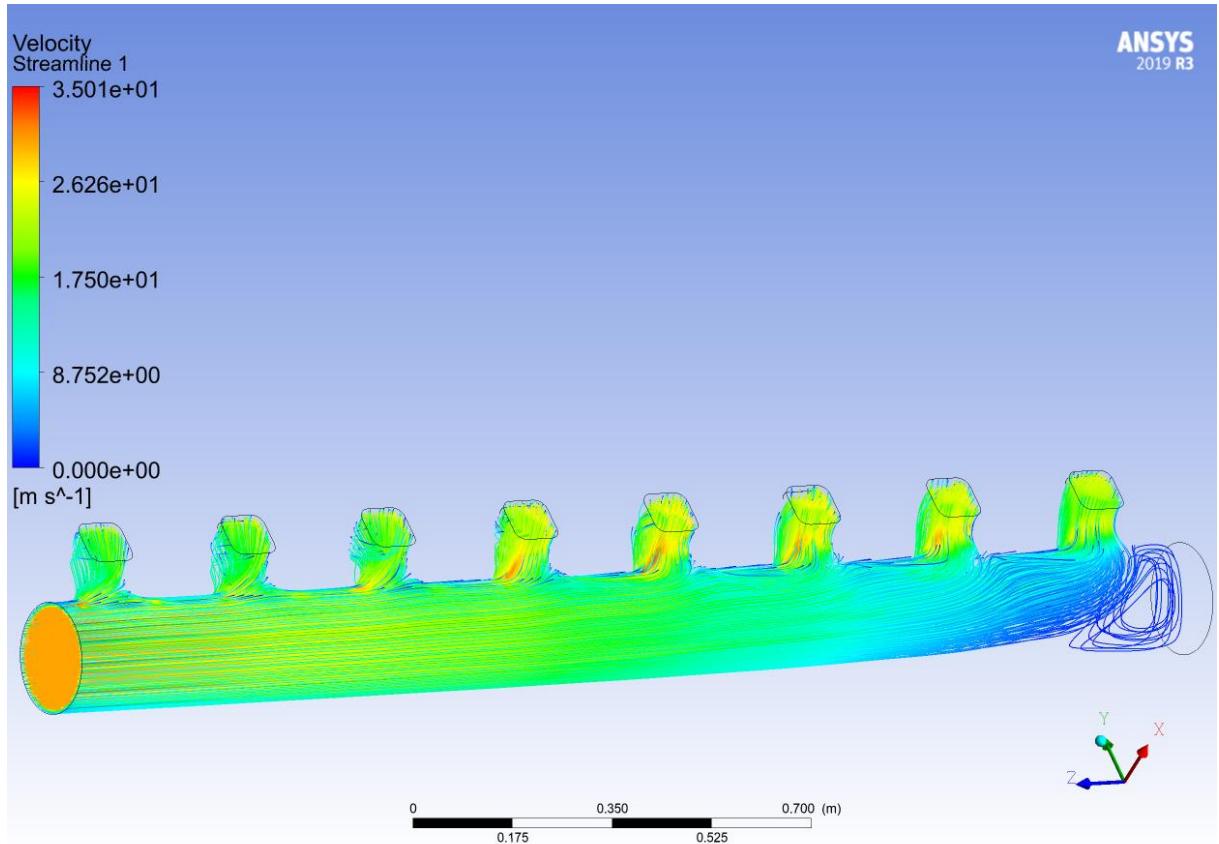


Figure 8.9: Velocity Streamlines of the air

Some important parameters for computing expense for each grid are compared in Table 1. The number of elements and nodes are only for the low boundary. Even though the manifold is meshed in the analyses, most of the computing takes place in the flow boundary since this is a CFD analysis. Finer mesh is applied to the flow boundary and coarser mesh is used for the manifold. Looking at table 1, it can be seen that hexahedral dominant grid has a lower number of elements, but more number of nodes due the geometry of hexahedrons. This results in a longer total time for solution but it should be kept in mind that the time for building the mesh also increases significantly.

Table 1. Comparison between the grids

Parameters	Tetra Grid	Improved Tetra Grid	Unstructured Pave Grid	Improved Unstructured Pave Grid
Iteration Number	250	250	250	250
Element Number	425024	500178	136523	482502
Node Number	82573	165079	104982	384863
Calculating Time Each Node (seconds)	1.499	1.802	0.705	2.601
Total Time (seconds)	374.631	450.493	176.363	650.25

8.5.Comparing Open-Source and Proprietary Software

CAD process and CFD analysis process on the software have multiple phases. Each phase is affected by each other so suitable CAD files causes less meshing time and more reliable meshes. Mesh quality affects solver time. Whole process of the simulation is dependent on effective time management. Only a few software includes each simulation phase like ANSYS. Accurate data transferring is crucial for reliable results. Compatibility between software is significant for the reliability and accuracy of the simulation. Proprietary software allows quick and more visual content of simulation but for open source software, the CFD modeller needs good knowledge of Linux Operation System (OS) and certain programming experience. In open source software, modification of simulation is allowed for wide simulation limits and techniques. Libraries of OpenFoam contains nearly every type of simulation and it is developing. Other benefit of open-source software is that it is more economical.

9. CONCLUSION

A comparison between four different meshes have been made and different grid types are investigated. Different programs with mesh generation abilities have been inspected and various Computer Aided Design (CAD) file formats have been used for meshing. The influence of CAD problems in mesh quality have been tested. The following conclusions can be made:

The CAD geometry has a big influence on the grid generation process. Complex curves should be minimized and extra surfaces such as fillets are recommended to be removed before mesh generation. Complex geometries should be designed in a way such that the part can be split into different bodies for efficient meshing. This also makes it possible to obtain structured meshes. Connecting surfaces and edges should be as smooth as possible. These parameters have great influence on mesh metrics (e.g. orthogonality, skewness, aspect ratio etc.) therefore the quality of the mesh. It is recommended to perform repairs on the CAD geometry before mesh generation.

Unstructured tetrahedral meshes are much easier to obtain on complex geometries compared to structured hexahedral meshes. These meshes can also be generated much quicker. One downside of these meshes in CFD analyses is that they don't give the most accurate results and their element quality is relatively low. Quick tests can be performed and fast results can be obtained using unstructured tetrahedral grids. Unstructured tetrahedral meshes can be generated automatically by many software in the market. Unstructured tetrahedral meshes can be improved to obtain better results. Improvements can be made by mesh refinement, adding boundary layers and altering the element size. It is recommended to use boundary layers near the wall areas, if no slip condition occurs. Better results were obtained with the use of improved tetrahedral mesh. Rectangular mesh has good overall element quality but element skewness increases at denser areas. A hexahedral dominant unstructured mesh can also be obtained automatically by computer programs. Correctly placed hexahedral elements give better results in most of the cases in CFD applications, however they take much longer time to build and may need longer time to solve due high amount of nodes. Fully automatically generated hexahedral mesh took relatively longer to build compared to tetrahedral meshes. It converged very quickly due to very small number of elements, however, the results are inaccurate and it is not

recommended to use this type of mesh without improvement/refinement. The hexahedral dominant hybrid mesh can be improved by increasing the element density and proper edge sizing. These operations result in a significant increase in building time of the mesh and solution time, but the most accurate results are obtained with improved hybrid mesh. Butterfly mesh has the best element quality and therefore it is expected to give the best results as it is reported by [1]. However, this type of structured mesh cannot be applied on every geometry. So the use of butterfly type of grid is recommended in circular pipes. If the geometry is too complex for its application, good results can still be obtained with the use of unstructured pave mesh (hybrid mesh). If the geometry is highly complex and time is crucial, then it is advised that the unstructured tetrahedral mesh should be used with small improvements and prism layers depending on the flow.

10. REFERENCES:

[1] *Strategies Toward Automation of Overset Structured Surface Grid Generation*,
William M. Chan, NASA Ames Research Center, 2017

[2] *Grid Generation Issues in the CFD Modelling of Two-Phase Flow in a Pipe*,
V.Hernandez-Perez*, M. Abdulkadir, B.J. Azzopardi, 2010

<https://all3dp.com/2/overview-of-3d-cad-file-formats/>

<https://www.designpresentation.com/blog/cad-file-formats/>

<https://fileinfo.com/filetypes/cad#:~:text=Common%20CAD%20file%20extensions%20include,DGN%2C%20and%20.>

https://www.researchgate.net/publication/321709563_A_comparative_study_of_CAD_data_exchange_based_on_the_STEP_standard

https://www.researchgate.net/publication/228976876_An_overview_of_3D_data_content_file_formats_and_viewers

<https://transmagic.com/cad-formats/>

ntrs.nasa.gov

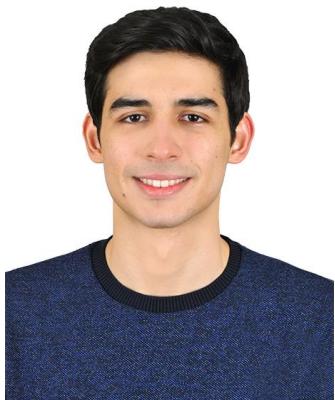
onlinelibrary.wiley.com

[sciencedirect.com](https://www.sciencedirect.com)

<https://www.researchgate.net/>

CURRICULUM VITAE

Alaattin Kerem Soğancı



Personal Details

Place of Birth: Mersin

Date of Birth : 01.05.1996

Military Status: Incomplete

Driving License: B

Contact

E-mail: kerem.soganci@hotmail.com

GSM: +90 531 840 48 45

Feneryolu Mahallesi Göktepe Sokak
Kadıköy/İstanbul

Linkedin: Alaattin Kerem Soğancı

Education

Marmara University (2015-2020)

Bachelor of Science in Mechanical
Engineering

GPA: 2.9

Languages

English - C1

German – A1

Work Experience

- Summer Intern in Production Department of Orhan Automotive
(June 2018 -July 2018)
- Summer Intern in After Sales Department of Mitsubishi Electric Turkey
(June 2019- August 2019)

Software Skills

AutoCAD, MATLAB, SolidWorks, Inventor, Fusion360, ANSYS Fluent, ANSYS CFX, Autodesk CFD, Microsoft Office, Photoshop,

Organizations

Marmara University Mechanical Engineering Society (MarmaraMES)

- 2016-2017 Member
- 2017-2018 Member of Board
- 2018-2019 Chairman of Board
- 2019-2020 Chairman of the Supervisory Board

Marmara University Faculty of Engineering Robotics Team (MUFE Robotic)

- 2017-2018 Stair Climbing Category Team Member
- 2018-2019 Stair Climbing Category Team Leader
- 2019-2020 Member of Design Team

Marmara University Underwater Sports Society (MÜSAS)

2017-Present Member

Certificates

- Autodesk Fusion 360 Certificate (2016)
- “Tıp İçin Tasarla” Competition and Symposium (2017)
- Effective Communication Certificate (2020)

Field of Interest

Design, Scuba Diving, Robotics, Gas Turbine, Internal Combustion Engines, ,3D Printer, Basketball, Camping and Trekking.

CURRICULUM VITAE

Arda Çağlar Pulat



Personal Information

Place of Birth: İstanbul

Date of Birth : 22.06.1998

Driving License: B

Contact Information

Gürsel Mahallesi, Seğmen Sokak

No:9 Daire: 12 Kağıthane/İSTANBUL

e-mail: ardacaglarpulat216@gmail.com

GSM: +90 542 653 52 94

Education

2016-2020 Marmara University

Bachelor of Science in Mechanical Engineering

2007-2016 Darüşşafaka Schools

Student Exchange Program

28.01.2019 – 24.06.2019, Erasmus+ Programme, Budapest University of Technology and Economics, Hungary

Work Experience

- 07.2019 – 08.2019 Yıldız Teknopark
Internship
Worked on optimizing workflow and B2B buying reports
- 07.2018-08.2018 Ustun Plastik (Plastmore)
Internship
Gained experience in injection moulding, CNC machining and CAD design.

Skills

Language:

English – C1

German – A1

Software:

MATLAB SolidWorks

ANSYS Fluent AutoCAD

Fusion 360 Microsoft Office

Social Clubs

Marmara University Mechanical Engineering Society (MES) – Member

Marmara University Underwater Sports Society (MÜSAS) – Member

Erasmus Student Network (ESN) Budapest - Member

Erasmus Student Network (ESN) Marmara - Member

Projects

Coffee Cup Design in Fusion 360

Jet Engine Design in SolidWorks

Licenses

- Driving License
B
- Parachute Certificate
THK-FAI
- CMAS* Diver
CMAS

Interests

Aviation, Martial Arts, CAD Design, Defence Technologies