



MARMARA UNIVERSITY  
FACULTY OF ENGINEERING



INVESTIGATION OF GRID GENERATION STRATEGIES  
FOR THE ANALYSIS OF EXHAUST MANIFOLDS

---

Abdulhalim Aslanaçier, Göksu Hazar

**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 EXHAUST MANIFOLDS

by

**Abdulhalim Aslanaçier**

**(150415034)**

**Göksu Hazar**

**(150415058)**

**July 05, 2020, Istanbul**

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

**OF**

**BACHELOR OF SCIENCE**

**AT**

**MARMARA UNIVERSITY**

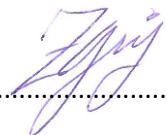
The author(s) hereby grant(s) to Marmara University permission to reproduce and to distribute publicly paper and electronic copies of this document in whole or in part and declare that the prepared document does not in anyway include copying of previous work on the subject or the use of ideas, concepts,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

Signature of Author(s) .....Abdulhalim Aslanaçier, Göksu Hazar.....

Department of Mechanical Engineering

Certified By .....Prof. Dr. Mehmet Zafer Gül.....



Project Supervisor, Department of Mechanical Engineering

Accepted By.....

Head of the Department of Mechanical Engineering

## **ACKNOWLEDGMENT**

We would like to thank Prof. Dr. Mehmet Zafer Gül for his valuable supervision and support throughout the development and improvement of this thesis.

Göksu Hazar

Abdulhalim Aslanaçier

## TABLE OF CONTENTS

ACKNOWLEDGMENT .....	I
ABSTRACT .....	IV
ABBREVIATIONS.....	VI
LIST OF FIGURES.....	VIII
LIST OF TABLES.....	X
1. INTRODUCTION.....	1
1.1. Literature Survey .....	2
1.1.1. Introduction.....	2
1.1.2. Systematic Approaches for Mesh Generation.....	2
1.1.3. Computational Fluid Dynamics (CFD) Researches.....	3
1.2. Exhaust Manifold .....	4
1.3. Mesh Generation .....	5
1.3.1. Types of meshes.....	5
1.3.1.1. Quad / Hexahedral meshes .....	6
1.3.1.2. Tri / Tetrahedral meshes .....	6
1.3.2. Classification of grids .....	6
1.3.2.1. Structured and unstructured mesh .....	6
1.3.2.2. Hybrid mesh.....	9
1.3.4. Mesh quality .....	11
1.3.4.1. Aspect ratio .....	11
1.3.4.2. Skewness.....	12
1.3.4.3. Orthogonality.....	13
1.3.4.4. Smoothness.....	13
2. GEOMETRY .....	14

<b>2.1.</b>	<b>Exhaust Manifold Parts .....</b>	<b>14</b>
<b>2.2.</b>	<b>Geometry Cleanup.....</b>	<b>15</b>
<b>3.</b>	<b>MESH GENERATION.....</b>	<b>18</b>
<b>3.1.</b>	<b>ICEM CFD .....</b>	<b>18</b>
<b>3.1.1.</b>	<b>Exporting mesh .....</b>	<b>19</b>
<b>3.1.2.</b>	<b>Mesh Generation.....</b>	<b>20</b>
<b>3.1.2.1.</b>	<b>Mesh smoothing criterias.....</b>	<b>30</b>
<b>3.1.3.</b>	<b>Mesh Generation On Part 2.....</b>	<b>36</b>
<b>3.2.</b>	<b>GMSH.....</b>	<b>40</b>
<b>3.2.1.</b>	<b>Starting Gmsh .....</b>	<b>41</b>
<b>3.3.</b>	<b>Salome Software .....</b>	<b>44</b>
<b>3.3.1.</b>	<b>Features.....</b>	<b>44</b>
<b>3.3.2.</b>	<b>Mesh generation on a simple cubical geometry .....</b>	<b>45</b>
<b>3.3.2.1.</b>	<b>Qaudraliteral mesh .....</b>	<b>45</b>
<b>3.3.3.</b>	<b>Mesh Generation Approaches on Exhaust Manifold .....</b>	<b>47</b>
<b>4.</b>	<b>CONCLUSION.....</b>	<b>51</b>
<b>5.</b>	<b>REFERENCES .....</b>	<b>52</b>

## **ABSTRACT**

### **Investigation Of Grid Generation Strategies For The Analysis Of Exhaust Manifolds**

HAZAR, Göksu

ASLANAÇIER, Abdulhalim

Department of Mechanical Engineering

Supervisor: Prof. Dr. Mehmet Zafer Gü'l

In this thesis, different types of meshes applied for various geometries to develop a systematic way for grid generation. In phase one, geometry defects has been fixed according to interface of the software. The next phase, after the geometry defects prevented mesh generation practices started to understand the mesh types and their importance on different places on the geometry. After identification of the mesh types, developing a systematic way for mesh generation approaches started. 3 main software used for this project as free and commercial software. In the first one, structured mesh.

ANSYS ICEM CFD software with ANSYS Mesher have been used for structured mesh generation process. The shell mesh was generated on Mesher automatically and then the volume mesh was completed on ICEM CFD to increase the overall quality of elements. The manual interference feature of Ansys. Different types of shell and volume mesh have been generated after the geometry clean up. Triangular, Tetrahedral, Hexahedral,

Quadrilateral mesh types with patch dependency and independency options are generated.

The quality of mesh is checked observing the parameters such as skewness, element quality, orthogonal quality, aspect ratio features. Afterwards, the mesh is run in the optimization process. The pressure, temperature and velocity streamlines are observed using ANSYS Fluent to see if there is convergence in the post processing step.

The second software which is Gmsh was used to generate tetrahedral mesh elements for three dimensional hence hexahedral mesh could not obtained. The third software, Salome was used to generate hexahedral mesh elements beside tetrahedral mesh elements. Structured hexahedral mesh obtained for volume with simple geometries.

## **ABBREVIATIONS**

AFT	:	Advancing Front Technique
AR	:	Aspect Ratio
CAD	:	Computer Aided Design
CAE	:	Computer Aided Engineering
CFD	:	Computational Fluid Dynamics
FEA	:	Finite Element Analysis
1D	:	1 Dimensional
2D	:	2 Dimensional
3D	:	3 Dimensional



## LIST OF FIGURES

Figure 1.1: Three dimensional demonstration of hex vs tetrahedral mesh.....	6
Figure 1.2 : Structured mesh example .....	8
Figure 1.3 : Unstructured mesh example.....	9
Figure 1.5 : Ideal and skewed triangles and quadrilaterals.....	12
Figure 1.6 : Visualization of skewness .....	13
Figure 1.7 : Demonstration of smoothness.....	14
Figure 2.1 : Fluid domain of exhaust manifold (1).....	14
Figure 2.2. : Fluid domain of exhaust manifold (2).....	15
Figure 2.3 : SpacemClaim repair operations section.....	15
Figure 2.4 : Repaired exhaust manifold part .....	16
Figure 2.5 : Interior geometry with unremoved parts.....	17
Figure 2.6 : Repairing process of unnecessary parts .....	17
Figure 2.7 : Interior geometry with removed unnecessary parts .....	17
Figure 3.1 : Surface mesh.....	20
Figure 3.2 : Mesh quality information.....	21
Figure 3.3 : Overall quality of the mesh.....	22
Figure 3.4 : Mesh quality information for smaller element size .....	23
Figure 3.5 : Critical region of the exhaust manifold.....	23
Figure 3.6 : Hexahedral volume mesh cross-section (1) .....	24
Figure 3.7 : Hexahedral volume mesh cross-section (2) .....	24
Figure 3.8 : Hexahedral volume mesh cross-section (3) .....	25
Figure 3.9 : Hexahedral volume mesh cross-section (4) .....	25
Figure 3.10 : Hexahedral volume mesh cross-section (5) .....	26
Figure 3.11 : Shell mesh information .....	27
Figure 3.12 : Information of shell mesh .....	28
Figure 3.13 : Quad shell mesh aspect ratio.....	29
Figure 3.14 : Quad shell mesh quality .....	29
Figure 3.15 : Quad mesh quality after smoothing .....	31
Figure 3.16 : Tetrahedral volume mesh quality.....	32
Figure 3.17 : Section view taken in the X plane.....	33
Figure 3.18 : Section view of volume mesh after smoothing.....	34
Figure 3.19 : Zoomed section view .....	34
Figure 3.20 : Tetrahedral mesh section view along the manifold tube.....	35
Figure 3.21 : Aspect ratio section view .....	35
Figure 3.22 : Laplace smoothing applied on tetrahedral mesh.....	36
Figure 3.23 : Second part of exhaust manifold extracted interior volume .....	37
Figure 3.24 : Mesh generation on second part.....	38

Table 3.3 : Mesh information of second part.....	38
Figure 3.25 : Section plane to see the volume mesh .....	39
Figure 3.26 : Tetrahedral mesh section view on mid -x plane.....	39
Figure 3.27 : Quad mesh on surface and hexaheral mesh on volume .....	40
Figure 3.28 : Section view of hexahedral volume mesh.....	40
Figure 3.29 : CAD data of Exhaust Manifold via Gmsh .....	41
Figure 3.30 : Tetrahedral mesh on exhaust manifold .....	42
Figure 3.31 : Mesh information for tetrahedral .....	43
Figure 3.32 : Closer look to the tetrahedral mesh generated .....	43
Figure 3.33 : Mesh information panel .....	44
Figure 3.34 : Mesh generated on geometry .....	45
Figure 3.35 : Mesh quality information panel .....	46
Figure 3.36 : Hexahedral mesh generated on cubical geometry.....	47
Figure 3.37 : Mesh base information panel .....	47
Figure 3.39 : Mesh generation demonstration on exhaust manifold .....	48
Figure 3.40 : Mesh generation parameter selection for 3D .....	48
Figure 3.41 : Mesh generation parameter selection for 2D .....	49
Figure 3.42 : Mesh generation parameter selection for 1D .....	49
Figure 3.43 : Wire discretization parameter selection .....	49
Figure 3.38 : Mesh computation error panel .....	50

## **LIST OF TABLES**

Table 1.1 : Skewness ranges and cell quality.....	12
Table 3.1 : Shell mesh information.....	30
Table 3.2 : Mesh information.....	32
Table 3.3 : Mesh information of second part.....	38
Table 3.4 : Chosen parameters for the mesh generation.....	42

# CHAPTER 1

## 1. INTRODUCTION

A preprocessing step for the computational field simulation is the discretization of the domain of interest which is called mesh generation. A mesh is a discretization of a geometric domain into small simple shapes, such as triangles or quadrilaterals in two dimensions and tetrahedra or hexahedra in three dimensions. Computational modelling and simulation is underpinning a diverse range of applications, from computer graphics and animation to complex numerical simulations of physical phenomena, such as fluid dynamics and structural mechanics.

Modern mesh generation is concerned with the development of efficient and automatic algorithms to construct and maintain high-quality meshes for complex objects and domains. Meshes are expected to conform to a number of often divergent requirements, adherence to complex geometrical features, facilitation of adaptive and incremental refinement and significantly, preservation of optimal element geometry.

This dissertation concerns about to execute a systematic way for grid generation for complex geometries. In this project, a specific exhaust manifold used for proceeding to determine the systematic way of mesh generation. Grid generation process is applied on different softwares and the optimization of grid has been searched. Initially, the geometry has been cleaned and simplified to get rid of complexities, small edges and areas on the surfaces. For each grid, the quality parameters such as aspect ratio, skewness, orthogonal quality are being observed and the best grid with maximum element number and qualities has been illustrated. Also, set of analysis are carried out to see the pressure, velocity and temperature changes on the exhaust manifold. The quality and number of elements have increased and the mesh has run with the same boundary conditions to see if there is a convergence at the end.

Different softwares such as ICEM CFD, Mesher, Salome, Gmsh have been used for mesh generation. Various techniques have developed to optimize the mesh process such as utilizing the surface quality of Mesher and transferring the files to continue the process

on ICEM CFD. Initially surface mesh has generated and after the quality approved the volume mesh generation has done. Quadrilateral, triangular mesh types for the surface; hexahedral, tetrahedral and hybrid mesh types have generated for the volume. The details explained step by step on the following pages.

The importance of this issue is that engineers may come across with different kinds of complex geometries and to produce high quality mesh generation with expected parameters takes time for each different geometry. This dissertation aims to reduce the amount of time spent to generate mesh by determining a systematic way for the analysis. Consequently, we concentrate on two key areas: (i) the development of meshing techniques and steps, and (ii) the research for grid generation programs.

## **1.1.Literature Survey**

### **1.1.1. Introduction**

Modern mesh generation is concerned with the development of efficient and automatic algorithms to construct and maintain high-quality meshes for complex objects and domains. The importance of this issue is that engineers may come across with different kinds of complex geometries and to produce high quality mesh generation takes time for each different geometry. A systematic approach needed to reduce the time spent for mesh generation. Aim of this survey is to demonstrate the researches done on this issue.

### **1.1.2. Systematic Approaches for Mesh Generation**

The literature search on this topic shows that the researches on this issue are restricted. A research done by Hengjin Liu aims to establish a number of criteria for evaluating 3-D mesh-generation software, and to select the 3-D mesh generator that is most suitable for use in our software pipeline for modelling and simulation of complex natural structures [1]. Different sorts of software as free and commercial investigated on this research. A software chosen as the best 3D mesh generator for the purposes of software pipeline. Some important criteria prioritized for this purpose. “The evaluation criteria for our project include the following characteristics: ability to preserve the boundary-surface mesh, volume mesh quality, robustness, time efficiency and cost.”[1]. The software selected to search for this project are COG, DistMesh, Gmsh, GRUMMP, Netgen, Tetgen, SolidMesh as free softwares and ADINA, ANSYS, HyperMesh, GiD

as commercial software. The Gmsh program is finally selected as the best 3-D mesh generator for the purposes of software pipeline.

Another research done for mesh generation approaches by Monan Wang, Jian Gao and Xinyu Wang [2]. The research investigates “High-quality mesh generation for human hip based on ideal element size: methods and evaluation”. High quality surface and volume meshes were generated using the advancing-front technique (AFT) and Delaunay algorithms in this paper. The generated mesh was uniform and contained smooth transitions. In another research done by Kirk S. Walton describes the AFT method as following. “The benefits of advancing front algorithms include: being general enough to mesh any model, providing high quality elements near the boundary where quality is critical, and not being restricted by mesh interaction with multiple adjoining models.”

### **1.1.3. Computational Fluid Dynamics (CFD) Researches**

One of the research that was carried out for an optimal geometry and reducing the back pressure [3]. The exhaust manifold system considered in the case has 4 inlets connected to the exhaust port of the engines and a single outlet from where the flow is passed on to the exhaust system before ejection into the ambient. Due to lack of experimental data, an industrially available manifold geometry was considered for the present analysis. It consists of the pipe diameter of 42 mm and total span of the manifold to be 0.6 m. The base model has the outlet placed besides the first port having smaller length of the curved pipe from the individual engine exhausts.

A steady state single phase single-species simulation with exhaust gas as the working fluid was carried out for the two geometries at 4 different mass flow rates from each inlet to determine the pressure drop. The geometry was created using ANSYS ICEM CFD and a multi-block structured mesh was created. ANSYS CFX is used for doing the flow simulation under isothermal conditions. The overall analysis is performed on ANSYS Workbench. A steady state single-species simulation is carried out under isothermal conditions for exhaust gas. Turbulence will be modeled by turbulence model appropriate to account for high velocities and strong streamline curvature in the flow domain. The reference pressure is set at 1 atm and all pressure inputs and outputs were obtained as gauge values with respect to this.

The flow analysis of exhaust manifold was performed. The existing manifold is modified by changing its geometry to get optimal geometry. Both old and new models are analyzed under same boundary conditions. The results of new model were compared with the existing model. Pressure and velocity graphs were drawn for the new model and are compared with existing model. The decrease in back pressure is shown by using contour and vector diagrams. The flow is made efficient by decreasing the exhaust gas back pressure in the newly modified model thus increasing the volumetric efficiency of the engine.

## **1.2.Exhaust Manifold**

An exhaust manifold collects the exhaust gases from multiple cylinders into one pipe. It is attached downstream of the engine and is major relevance in multi-cylinder engines where there are multiple exhaust streams that have to be collected into a single pipe. When an engine starts its exhaust stroke, the piston moves up the cylinder bore, decreasing the total chamber volume. When the exhaust valve opens, the high pressure exhaust gas escapes into the exhaust manifold or header, creating an exhaust pulse comprising three main parts: The high pressure head is created by the large pressure difference between the exhaust in the combustion chamber and the atmospheric pressure outside of the exhaust system. As the exhaust gases equalize between the combustion chamber and the atmosphere, the difference in pressure decreases and the exhaust velocity decreases. This forms the medium-pressure body component of exhaust pulse. The remaining exhaust gas forms the low pressure tail component. This tail component may initially match ambient atmospheric pressure, but the momentum of the high and medium pressure components reduces the pressure in the combustion chamber to a lower than atmospheric level. This relatively low pressure helps to extract all the combustion products from the cylinder and induct the intake charge during the overlap period when both intake and exhaust valves are partially open. The effect is known as scavenging. Length, cross-sectional area, and shaping of the exhaust ports and pipe works influences the degree of scavenging effect.

There have been many CFD flow analysis implemented on exhaust manifolds. One of the most important part of the analysis is the mesh generation process. Most of the time and the effort is spent on this step as it is quite related to the other processes of analysis

such as setting up parameters, solving, post processing and result. Creating a high qualified mesh would result in more convenient and accurate result at the end.

### **1.3.Mesh Generation**

Meshering can be used for a wide variety of applications, the principal application of interest is the finite element method. Surface domains may be subdivided into triangular or quadrilateral shapes, while volumes may be subdivided primarily into tetrahedral or hexahedral shapes. Meshering algorithms ideally define the shape and distribution of the elements.

Mesh generation is usually considered as the pre-processing step of numerical computational techniques. Meshes used in numerical solution algorithms must satisfy several conditions depending on the problem.

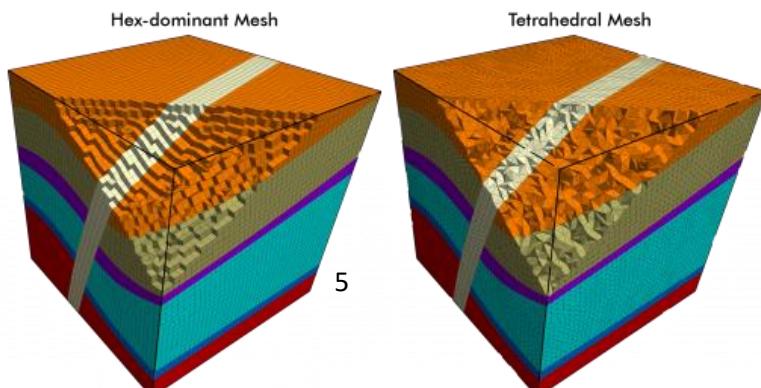
#### **1.3.1. Types of meshes**

Mesh elements can be 1D, 2D or 3D elements in function of the model to simulate. For this project we have searched for 3D mesh elements which are our research field.

The most common types of elements utilized in numerical approximations are triangles or quadrilaterals in two-dimensions and tetrahedral or hexahedral elements in three-dimensions.

In the case of tetrahedral meshing, algorithms are available that can generate greater than 400,000 tetrahedra per minute. However, automated hexahedral mesh generation algorithms are available for a more limited class of geometries.

Because of the limited class of geometries for which hexahedral meshes can be built, a significant amount of time in generating a hexahedral mesh is devoted to decomposing (cutting up) a model into pieces for which a known hexahedral mesh generation algorithm will succeed.



**Figure 1.1:** Three dimensional demonstration of hex vs tetrahedral mesh

### **1.3.1.1.Quad / Hexahedral meshes**

Surface domains can be subdivided into quadrilateral elements, whereas volumes can be subdivided into hexahedral elements by structured as well as unstructured meshing methods. Isoparametric coordinates can be used to generate both quad / hexahedral meshes, which are considered as structured. As for unstructured quad / hexahedral meshes, they generated using direct and indirect approaches. Meanwhile, some methods are also available that combine hexahedral and tetrahedral elements in a single three-dimensional domain.

Generally, unstructured mesh generation algorithms use triangle and tetrahedral mesh elements. As a result of this, most of the literature and software are triangle and tetrahedral, although there is a significant group of literature that focuses on unstructured quad and hexahedral methods.

### **1.3.1.2.Tri / Tetrahedral meshes**

Triangle and tetrahedral meshes are the most common forms of unstructured mesh generation. Most techniques currently in use can be considered in three main categories: Octree, Delaunay and Advancing Front techniques. Tri / tetrahedral meshes can also be constructed from quad / hexahedral mesh elements.

## **1.3.2. Classification of grids**

Meshes can be categorized as structured, unstructured and hybrid meshes by configuration. The choice of the mesh type is clearly related to the application.

### **1.3.2.1.Structured and unstructured mesh**

There are often some misunderstandings regarding structured/unstructured mesh, meshing algorithm, and solver. A mesh may look like a structured mesh but may or may not have been created using a structured algorithm based tool. For example, GAMBIT is an unstructured meshing tool. Therefore, even if it creates a mesh that looks like a structured (single or multi-block) mesh through pain-staking efforts in geometry decomposition, the algorithm employed was still an unstructured one. On top of it, most

of the popular CFD tools like: ANSYS FLUENT, ANSYS CFX, Star CCM+, OpenFOAM, AxSTREAM CFD, etc. are unstructured solvers. Unstructured solvers can only work on an unstructured mesh even if provided with a structured-looking mesh created using structured/unstructured algorithm based meshing tools.

ANSYS ICEM CFD can generate both structured and unstructured meshes using structured or unstructured algorithms which can be given as inputs to structured as well as unstructured solvers, respectively.

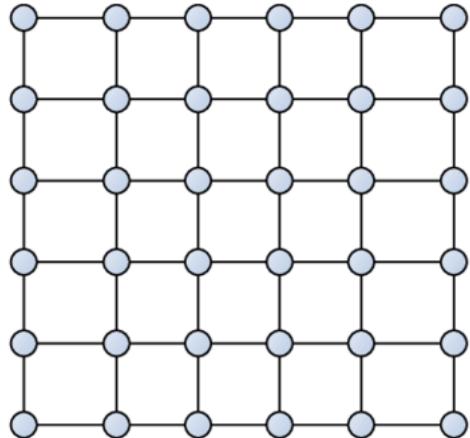
### **Structured mesh**

Structured meshes are widely used in many real CFD applications. In structure mesh all interior vertices are topologically alike. This can be generated algebraically by interpolation from boundaries for example transfinite interpolation. In structured meshes points are laid in the region of interest in a regular pattern; these points form quadrilateral cells in 2D and hexahedral in 3D.

There are some significant advantages associated with structured meshes due to which they are preferred over other grid methods in many applications.

The advantages of structured grid methods are given below:

- From the solver point of view, they have simplicity and easy data access.
- From users point of view, they possess high degree of control. For example user can easily control the refinement level in a particular region by placing more dense points on the boundaries of that region.
- Memory requirement to simulate flow in a structured domain is less as compared to other methods.
- Element could easily be flow aligned in structured meshes which offer more accuracy of the results.
- Examination of flow field in post processing is easier with structured mesh.



**Figure 1.2 :** Structured mesh example

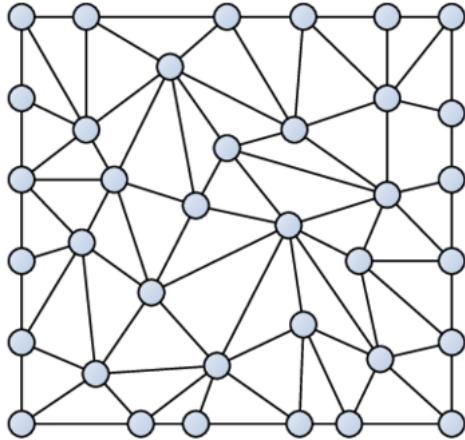
### Unstructured mesh

Unstructured meshes are used in many FEA and CFD applications which have irregular shape of object or region of interest. Unlike structured mesh, they can have elements of arbitrary topology in a confined region. In unstructured meshes points are laid in the region of interest in a random order forming triangular subdivision of the region. Unstructured mesh in its most basic form contains triangles in 2D and tetrahedral cells in 3D, but may be made of hexahedral or elements of any shape in general. Unstructured meshes are more easily generated and adapted but require a much more complex solution data structure. In generation of unstructured mesh user control is limited as interior nodes are generated automatically without any interaction of user.

The advantages of unstructured grid methods are given below:

- They have the ability to adopt better around the complex geometries.
- They can be generated with less efforts and interactions.

- They can be easily adopted by the applications which require automation of the mesh generation process .



**Figure 1.3 : Unstructured mesh example**

### 1.3.2.2.Hybrid mesh

General configurations can conceivably be treated with either type of mesh, and hybrid combinations are also possible, using individual structured meshes near boundaries, with the sub-regions being connected by an unstructured mesh. Thus, hybrid mesh is a positive combination of both structured and unstructured mesh and possesses the advantages of both.

### 1.3.3. Classification of mesh generation techniques

As discussed before, the mesh generation techniques can be divided to two major categories of *structured* and *un-structured* mesh. Strictly speaking, a structured mesh can be recognized by all interior nodes of the mesh having an equal number of adjacent elements. For our purposes, the mesh generated by a structured grid generator is typically all quad or hexahedral. Algorithms employed generally involve complex iterative smoothing techniques that attempt to align elements with boundaries or physical domains. Where non-trivial boundaries are required, ***block structured*** techniques can be employed which allow the user to break the domain up into topological blocks. Structured grid generators are most commonly used within the CFD

field, where strict alignment of elements can be required by the analysis code or necessary to capture physical phenomenon.

Unstructured mesh generation, on the other hand, relaxes the node valence requirement, allowing any number of elements to meet at a single node. Triangle and Tetrahedral meshes are most commonly thought of when referring to unstructured meshing, although quadrilateral and hexahedral meshes can also be unstructured. While there is certainly some overlap between structured and unstructured mesh generation technologies, the main feature which distinguish the two fields are the unique iterative smoothing algorithms employed by structured grid generation.

### **Structured Meshes**

- Complex Variables (Restricted to 2D)
- Algebraic Techniques (TFI)
- PDE Methods (PDE)

### **Unstructured Meshes**

- Delany Triangulation
- Advancing Front
- Octree Method
- Polyhedral Meshes
- Overset Meshes
- Cartesian Meshes

### **Hybrid Meshes**

#### **Adaptive Meshes**

- Structured
- Unstructured

### 1.3.4. Mesh quality

A mesh is considered to have higher quality if a more accurate solution is calculated more quickly. Accuracy and speed are in tension. Decreasing the mesh size always increases the accuracy but also increases computational cost.

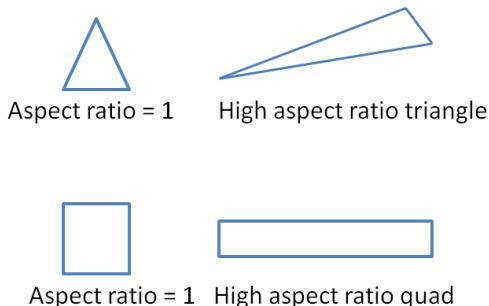
Generally, there are four criteria to evaluate the qualities of the solid mesh, which are aspect ratio, skewness, orthogonality, and smoothness. Aspect ratio is the most important criteria to evaluate the qualities of each individual element. On the other hand, skewness, orthogonality, and smoothness, show the quality prediction for two adjacent elements sharing the same inner face. The definition of each quality is explained below.

#### 1.3.4.1. Aspect ratio

Aspect ratio is the measure of a mesh element's deviation from having all sides of equal length. A high aspect ratio occurs with long, thin elements. Entering an overly large value for the minimum element size mesh control may cause the mesh generator to create solid elements with high aspect ratios.

Having a large aspect ratio can result in an interpolation error of unacceptable magnitude.

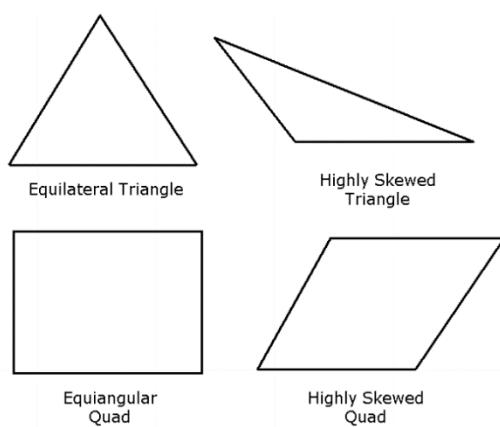
$$AR = \frac{\text{Long edge length}}{\text{Short edge length}} \quad (1.1)$$



**Figure 1.4 :** Aspect ratio visualization of formula

### 1.3.4.2. Skewness

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (i.e., equilateral or equiangular) a face or cell is.



**Figure 1.5 :** Ideal and skewed triangles and quadrilaterals

Table 1.1 lists the range of skewness values and the corresponding cell quality. According to the definition of skewness, a value of 0 indicates an equilateral cell (best) and a value of 1 indicates a completely degenerate cell (worst).

**Table 1.1 :** Skewness ranges and cell quality

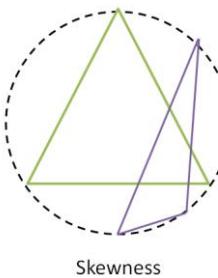
Value of Skewness	Cell Quality
1	Degenerate
0.9-<1	Bad(sliver)
0.75-0.9	Poor
0.5-0.75	Fair
0.25-0.5	Good
0>-0.25	Excellent
0	Equilateral

Highly skewed faces and cells are unacceptable because the equations being solved assume that the cells are relatively equilateral/equiangular.

Two methods for measuring skewness are:

Based on the equilateral volume (applies only to triangles and tetrahedra).

$$\text{Skewness} = \frac{\text{optimal cell size} - \text{cell size}}{\text{optimal cell size}} \quad (1.2)$$



**Figure 1.6 :** Visualization of skewness

Based on the deviation from a normalized equilateral angle. This method applies to all cell and face shapes, e.g., pyramids and prisms.

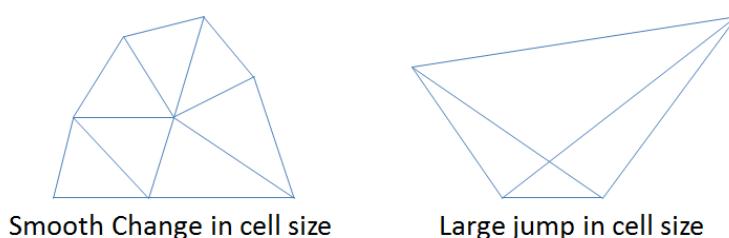
#### 1.3.4.3.Orthogonality

The concept of mesh orthogonality relates to how close the angles between adjacent element faces (or adjacent element edges) are to some optimal angle (depending on the relevant topology). An example for orthogonality is presented.

The orthogonality measure ranges from 0 (bad) to 1 (good).

#### 1.3.4.4.Smoothness

In a high-quality mesh, the change in size from one face or cell to the next should be gradual (smooth). Large differences in size between adjacent faces or cells will result in a poor computational grid because the differential equations being solved assume that



the cells shrink or grow smoothly.

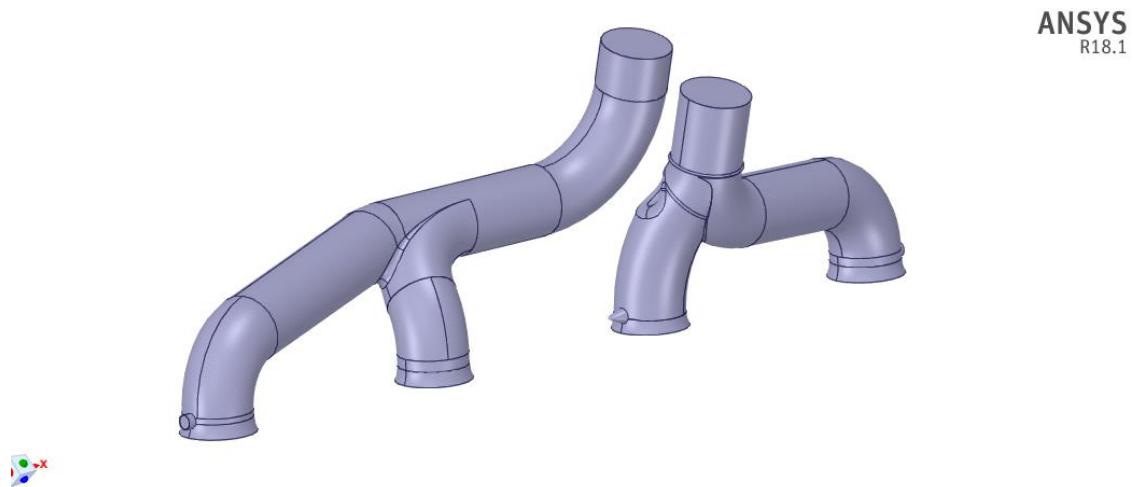
**Figure 1.7 :** Demonstration of smoothness

## CHAPTER 2

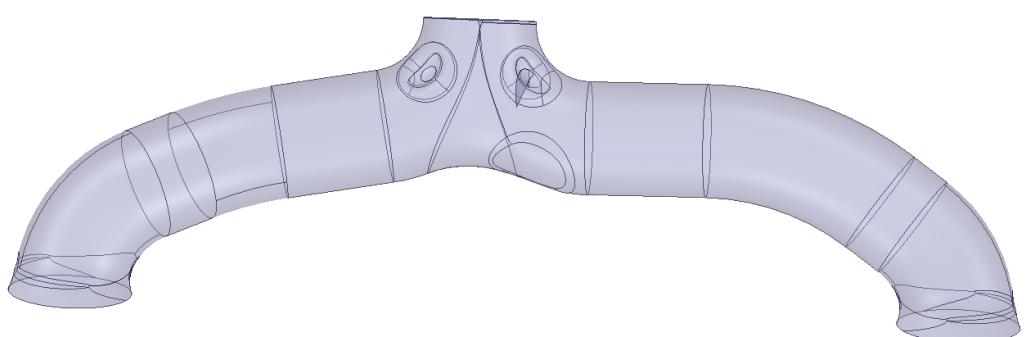
### 2. GEOMETRY

#### 2.1.Exhaust Manifold Parts

Fluid domain geometries extracted from the exhaust manifold parts shown below. These exhaust manifold parts belongs to a V16 locomotive engine.



**Figure 2.1 :** Fluid domain of exhaust manifold (1)



**Figure 2.2. :** Fluid domain of exhaust manifold (2)

## 2.2.Geometry Cleanup

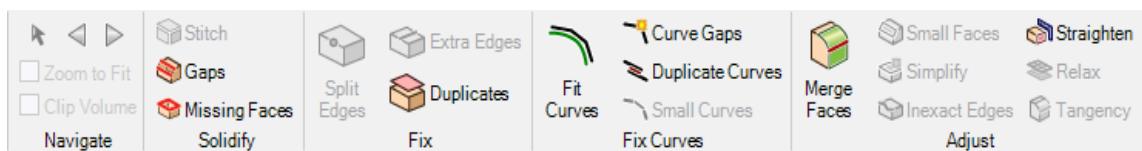
Cleaning up the geometry is one of the most important step before the mesh generation. The unwanted parts should be removed and if there is a problem with the geometry such as gaps, missing edges, complex surfaces should be fixed. In case the geometry is not cleaned completely, the problem is that there might be problems when generating mesh. So, to avoid problems and achieve high quality mesh, the geometry has to be simplified.

In this project, the geometry has been cleaned in chosen programs such as Ansys Space Claim, Design Modeler, Solidworks, Salome and other CAD software.

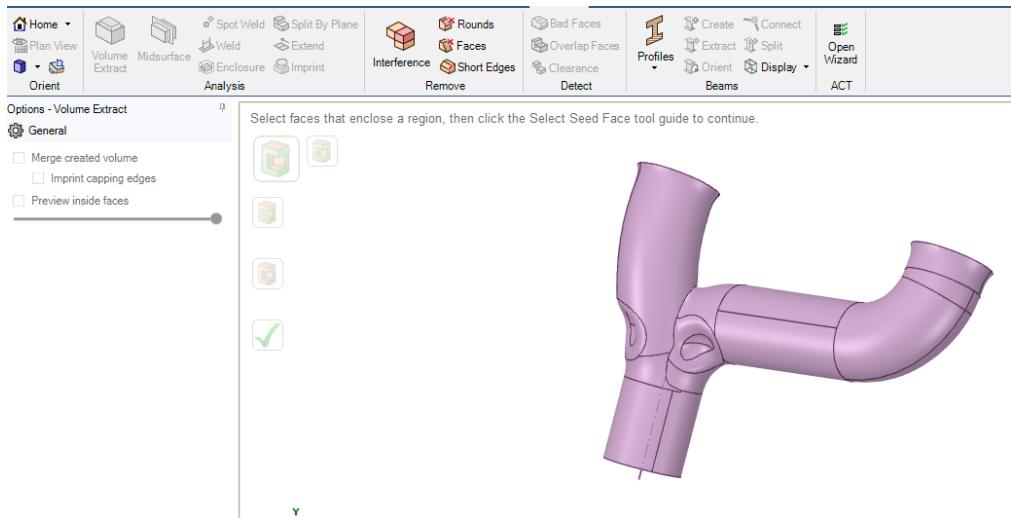
Some of the cleaning operations that are made on the geometry are as following on Ansys SpaceClaim.

For the mesh generation, the interior volume is used. It is extracted by the “volume extract” command by selecting the boundary edges. Initially the geometry has meshed and in the geometry some set of errors occurred such as gaps, missing edges.

As shown below the gaps, missing faces, extra edges, split edges, duplicates, curve gaps were very useful for cleaning the geometry for mesh under the repair section. One of the reasons that we come across with errors in mesh operation is small faces that occurs on the geometry. These small faces also fixed in the repair part. Some faces had to be merged to get rid of complexity.



**Figure 2.3 :** SpacemClaim repair operations section

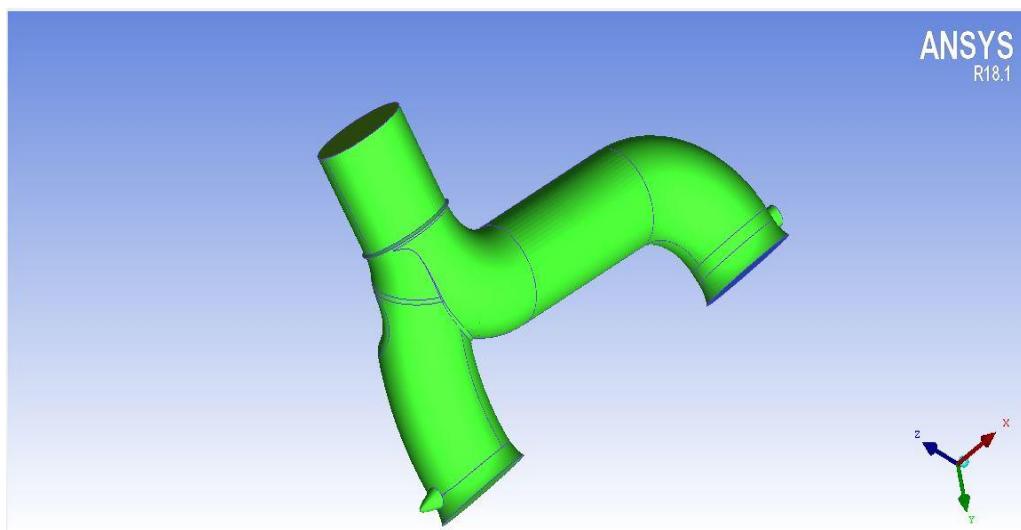


**Figure 2.4 :** Repaired exhaust manifold part

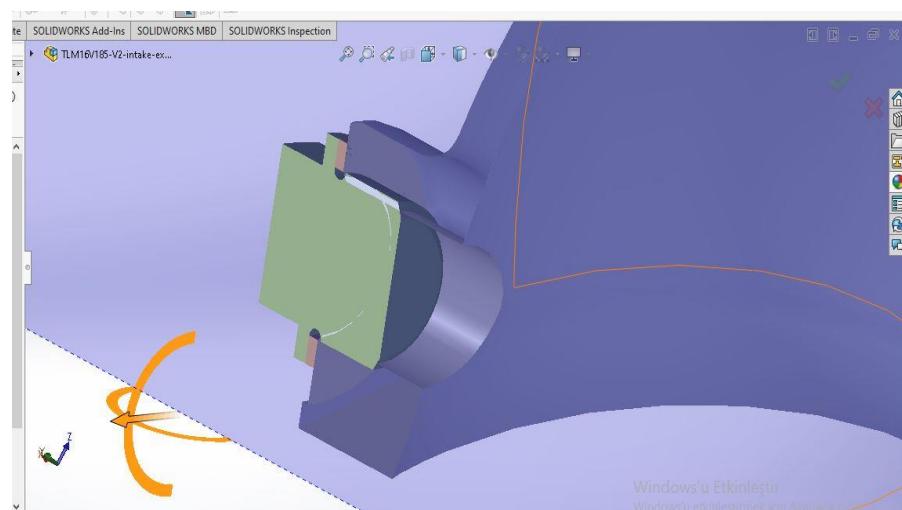
The first part of the exhaust manifold is going to be explained in this part.

Initially, the part was extracted from the whole engine geometry and the interior flow region was obtained.

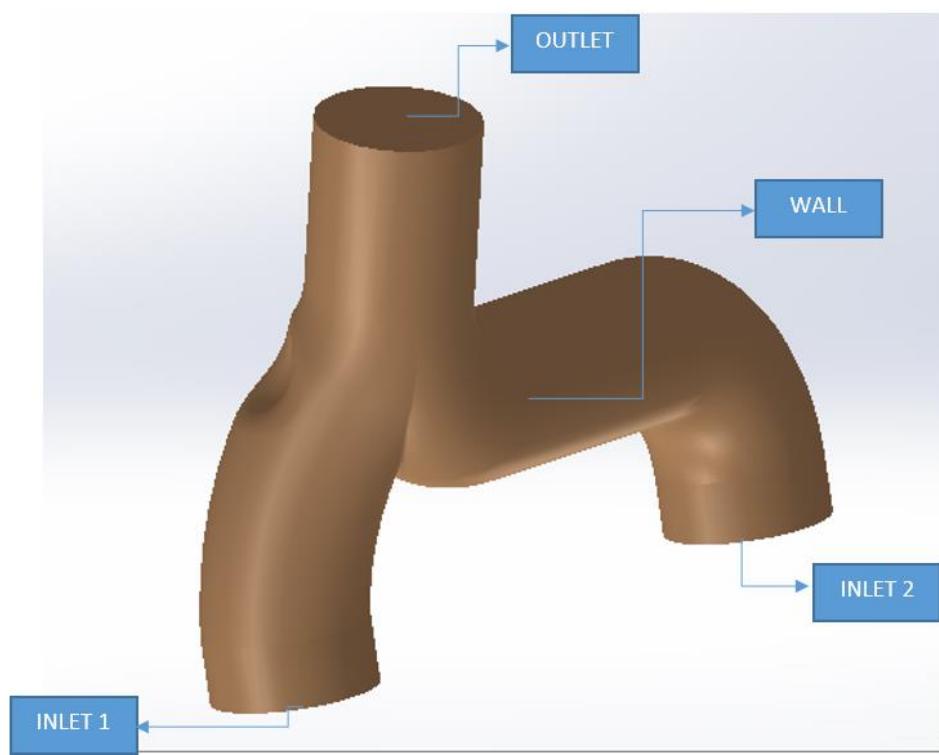
Before the mesh generation phase, by consulting with the professor and our research advisor, it was decided that there needs to be cleaning on the geometry. As it is seen below (Fig.1), there are some unnecessary parts on the geometry. Initially, these parts were removed and the ideal geometry was acquired on Solidworks design software. The second geometry (Fig.2) is going to be used for the mesh generation process.



**Figure 2.5 :** Interior geometry with unremoved parts



**Figure 2.6 :** Repairing process of unnecessary parts



**Figure 2.7 :** Interior geometry with removed unnecessary parts

## CHAPTER 3

### 3. MESH GENERATION

Once a mathematical model is selected, we can start with the major process of a simulation, namely the domain discretization process. Since the computer recognizes only numbers, we have to translate our geometrical and mathematical models into numbers which of course called discretization. The first action is to discretize the space, including the geometries and solid bodies present in the flow field or enclosing the flow domain. This set of points, which replaces the continuity of the real space by a finite number of isolated points in space, is called a grid or a mesh. The process of grid generation is in general extremely complex and requires dedicated software tools to help in defining grids that follow the solid surfaces (this is called ‘body-fitted’ grids) and have a minimum level of regularity. We wish already here to draw your attention to the fact that, when dealing with complex geometries, the grid generation process can be very delicate and time consuming. Grid generation is a major step in setting up a CFD analysis, since, as we will see the outcome of a CFD simulation and its accuracy can be extremely dependent on the grid properties and quality. Please notice here that the whole object of the simulation is for the computer to provide the numerical values of all the relevant flow variables, such as velocity, pressure, temperature, etc., at the positions of the mesh points. Hence, this first step of grid generation is essential and cannot be omitted. Without a grid, there is no possibility to start a CFD simulation.

#### 3.1. ICEM CFD

**Ansys ICEM CFD** is a popular proprietary software package which provides advanced geometry and mesh generation as well as mesh diagnostics and repair functions useful for in-depth analysis. Its design is centered around aerospace, automotive and electrical engineering applications with a specific focus on computational fluid dynamics and structural analysis.

ICEM CFD offers mesh generation with the capacity to compute meshes with various different structures depending on the users requirements. It is a powerful and highly

manipulative software which allows the user to generate grids of high resolution. This is a requirement as mesh generation is an inherently geometry dependent problem meaning there is no singular meshing method which can be used for every problem. ICEM CFD allows the following different types of grid structures to be created:

- Multi-block structured meshes
- Unstructured meshes
- Hybrid meshes

ICEM CFD provides a user interface which includes several components to make mesh generation intuitive and easy to use once its full capabilities are understood. The user interface contains a complete environment to create, modify and manage computational grids:

- Main Menu: Create/Open/Save/Close projects, Geometry/Mesh/Blocking options and parameters, Import/Export Model/Geometry/Mesh
- Utilities for visualization purposes: View/Zoom/Refresh screen, Undo/Redo commands, Wireframe/Solid Simple Display
- A hierarchical Display Control Tree: Model Geometry/Mesh/Blocking/Parts
- Function Tabs to modify the mesh: Geometry/Mesh/Blocking, Edit Mesh, Properties/Constraints/Loads/FEA solver options, Output Mesh
- Selection Toolbar
- Data Entry Zone
- Message Window
- Histogram Window

The user interface window can also be personalized in the settings menu.

### **3.1.1. Exporting mesh**

ICEM CFD allows the user to export their mesh into various different formats for compatibility with other external solvers. The meshing topology, associated parts, boundary conditions and loads should all be predefined before this stage. Some of the possible supported output solvers are:

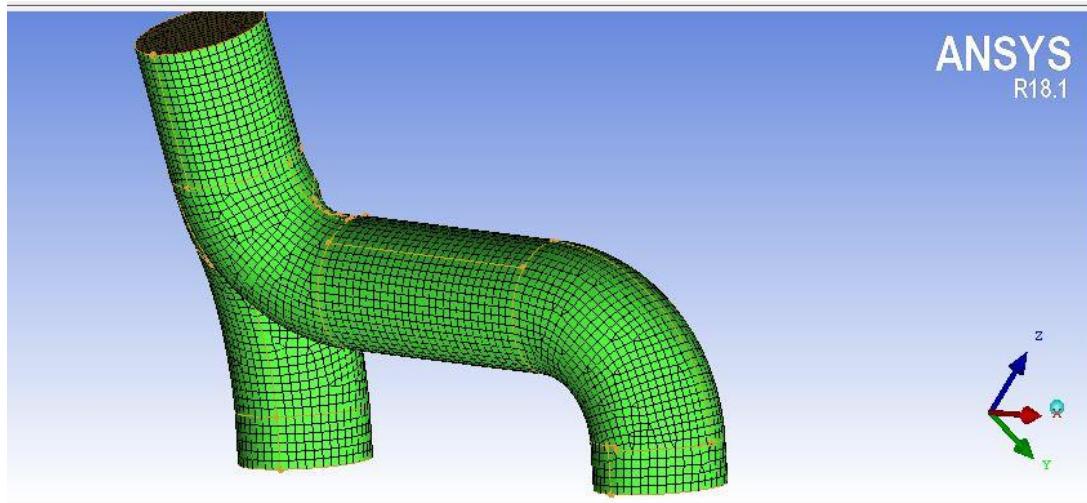
- ANSYS CFX
- ANSYS Fluent
- CGNS
- Plot3D
- STAR-CCM+

### 3.1.2. Mesh Generation

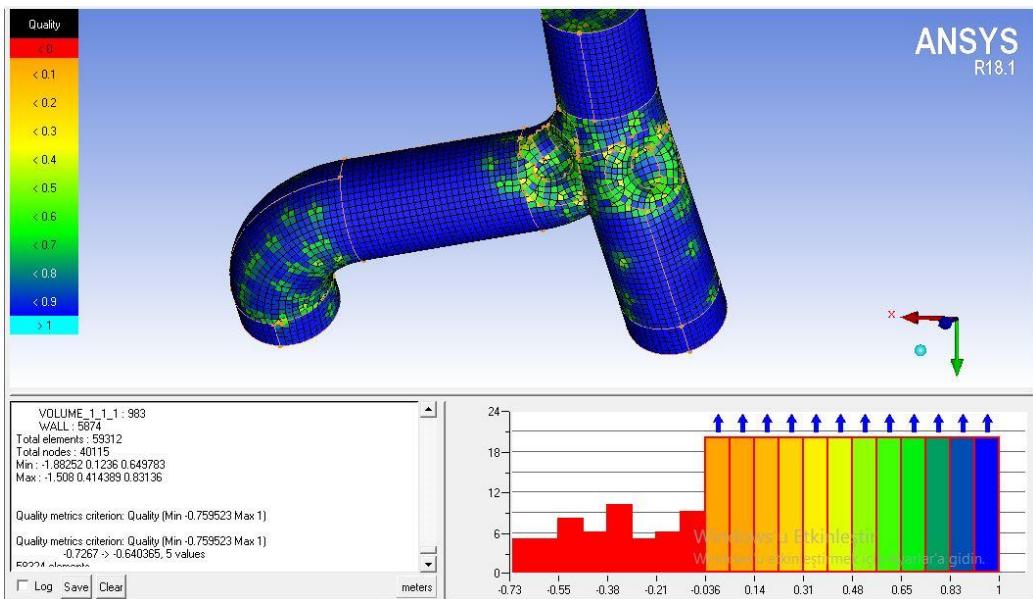
After creating names for the body boundaries, we proceed with the mesh generation. In ICEM CFD, first we have to create 2D shell mesh for the surfaces and then we can create the volume mesh. If the shell mesh is obtained with high quality and no errors, that means the volume mesh is probably going to be qualified.

In every step the quality, aspect ratio, orthogonal quality and other parameters are being tested and if there is an error we seek for the solution. In some cases, the mesh needs to be repaired to get rid of errors.

The first shell mesh is quadrilateral with very few triangular elements that is created on the surface in 2D. The mesh information is also provided below.



**Figure 3.1 :** Surface mesh



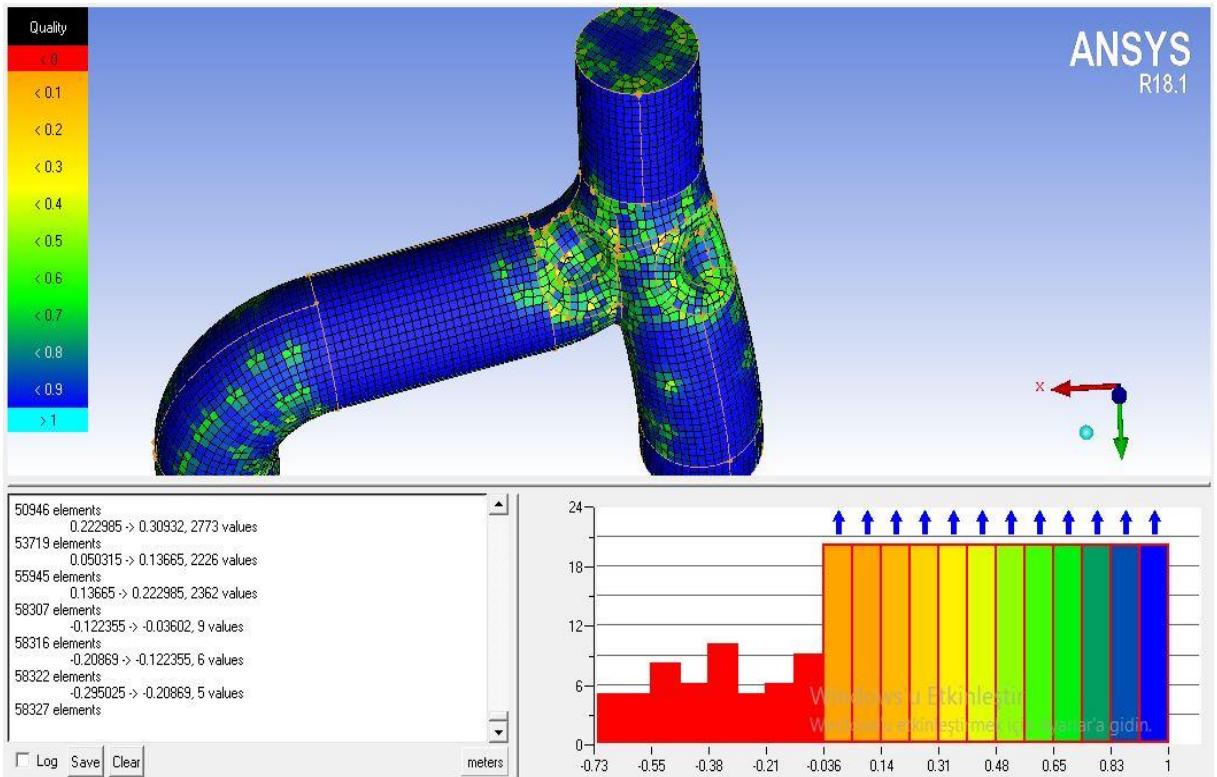
**Figure 3.2 :** Mesh quality information

As we observe, the mesh information is as follows:

Total Element: 59312

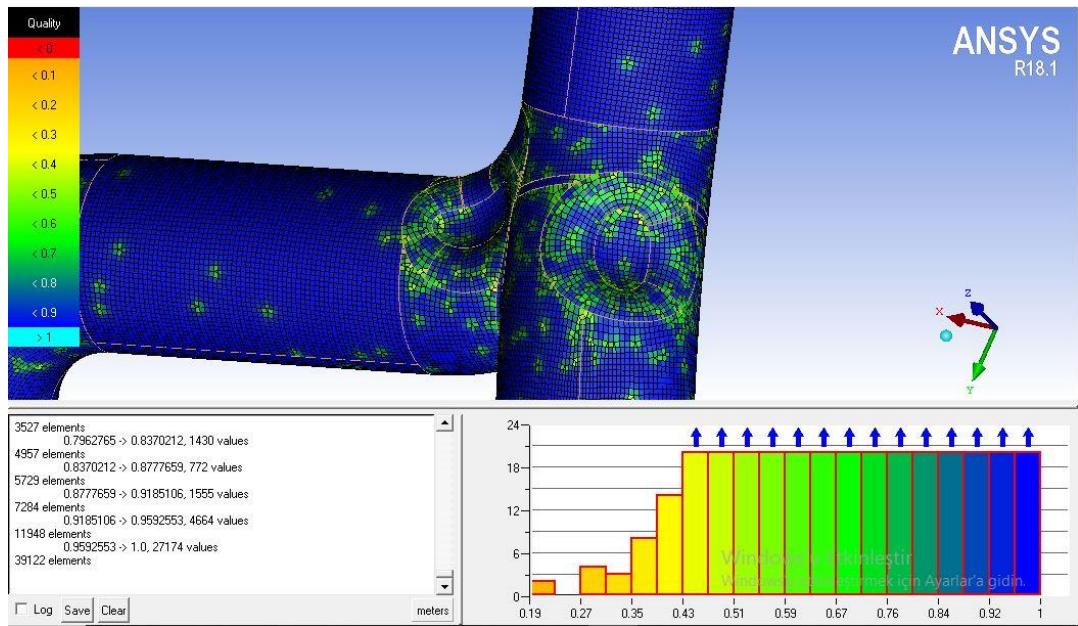
Total Nodes: 40115

The quality ranges between 0 and 1. The colored graph shows the overall quality of each part of the mesh. Once clicked on the column, the element numbers that are in this range pop up on the screen. That way, we can determine how many elements are belonged to the interested range.



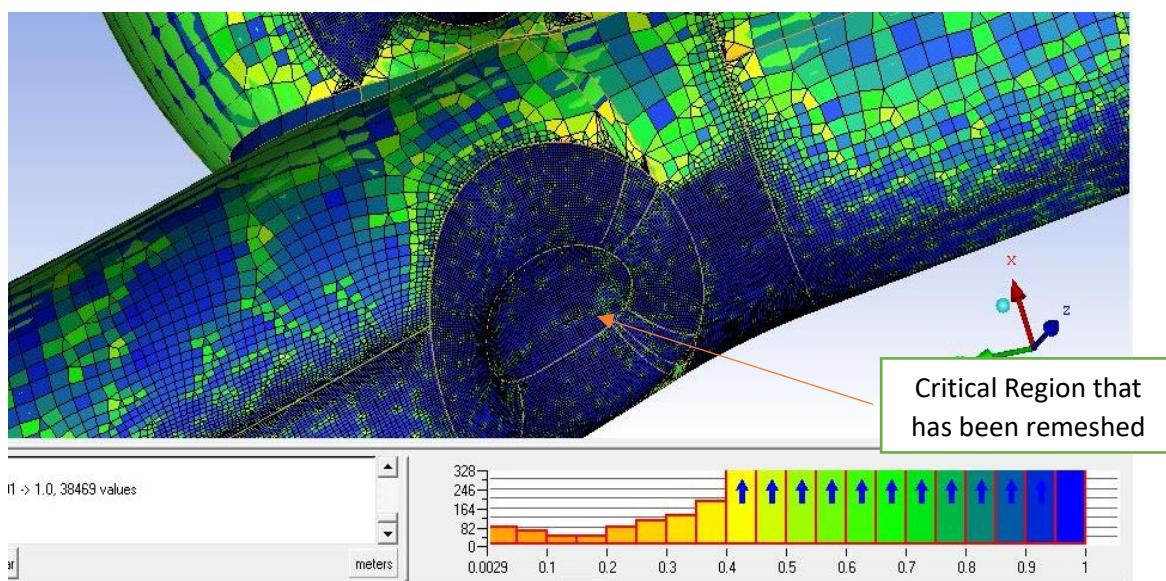
**Figure 3.3 :** Overall quality of the mesh

We are also able to detect which part of the mesh has the lowest quality. In the first mesh it is observed that, the lowest quality is obtained on the joint of the body where the manifolds connect. For most of the body, the quality is between 0.83 – 1 whereas in this part, it decays. So, we have to work on this part specifically to get rid of lower qualified elements. One way is to decrease the global element seed size. For example, changing it from 0.01 to 0.007 results in more qualified mesh map as it is seen in the following figure.



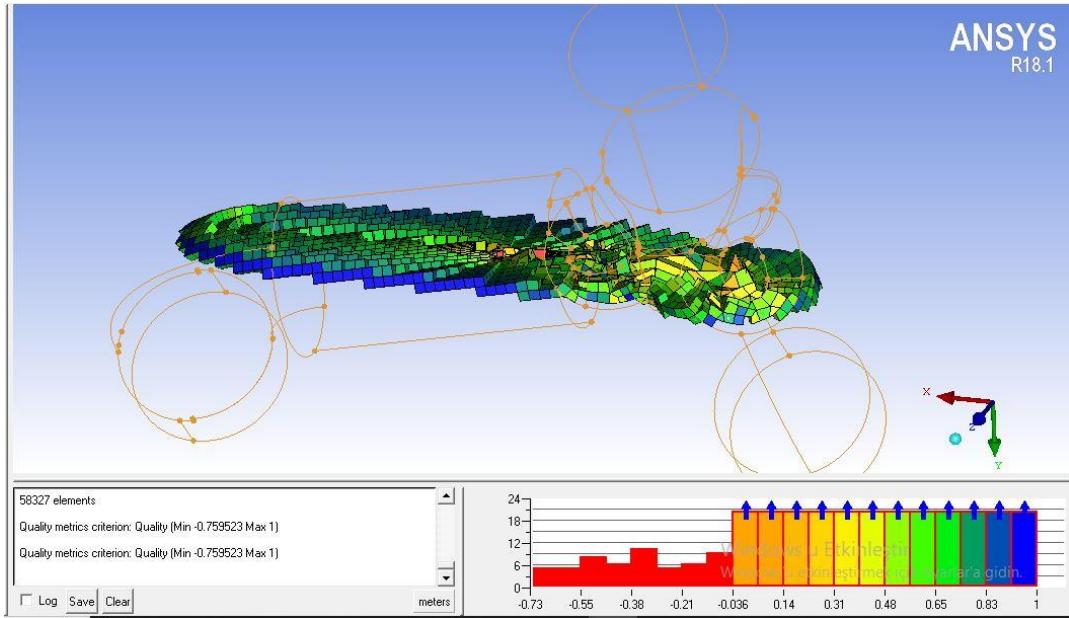
**Figure 3.4 :** Mesh quality information for smaller element size

ICEM CFD allows us to redefine mesh parameters such as mesh size, mesh patch dependency, mesh type and so on. For the lower quality regions, the parameters were set again and the mesh was computed. As it is seen below, with lower element size higher quality mesh is obtained.

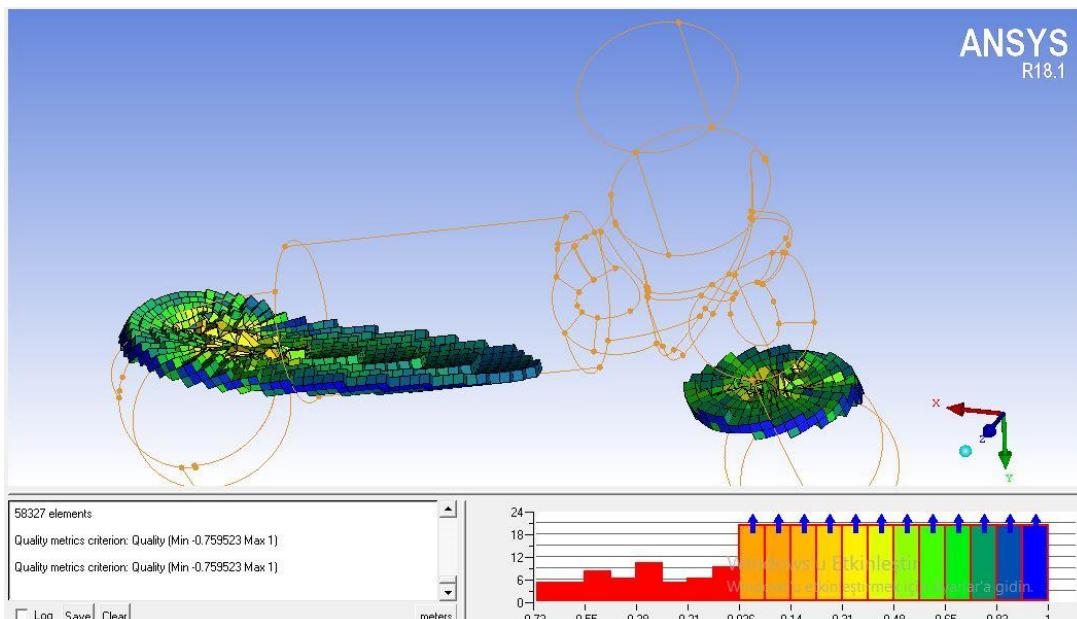


**Figure 3.5 :** Critical region of the exhaust manifold

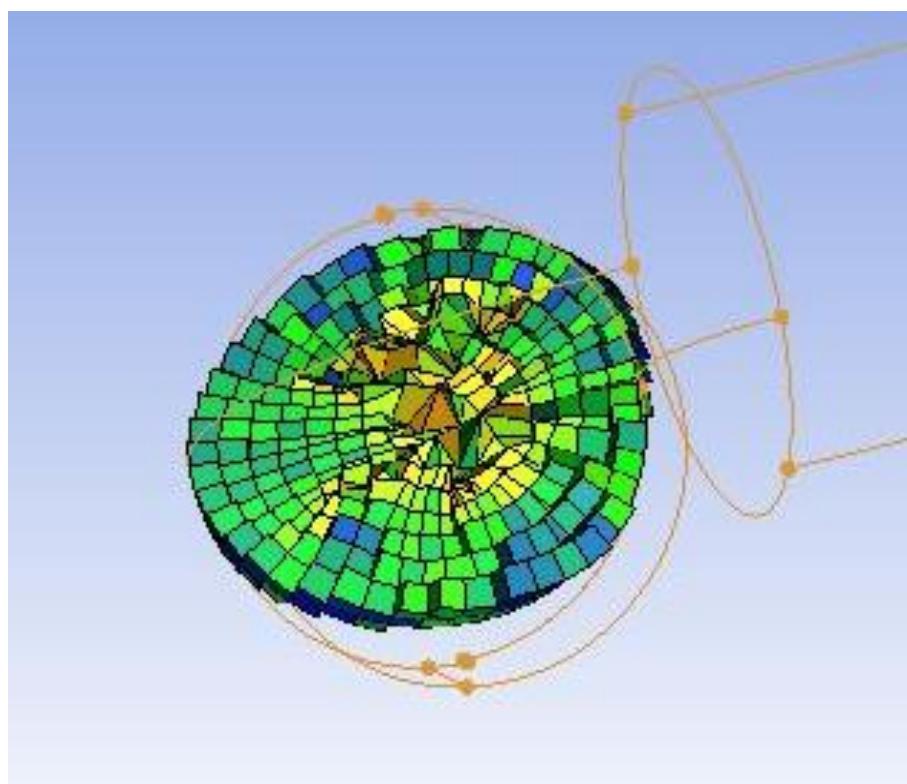
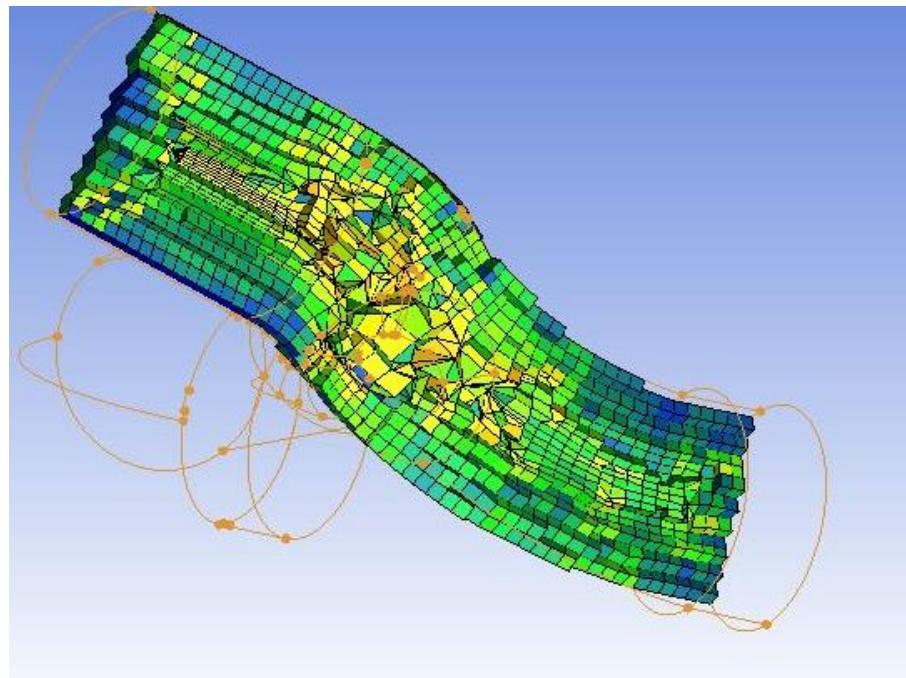
After the shell mesh was generated, the next step is volume mesh. The volume mesh type is hexahedral and it is set to be patch dependent. Some cross sections of the mesh are as followings:



**Figure 3.6 :** Hexahedral volume mesh cross-section (1)

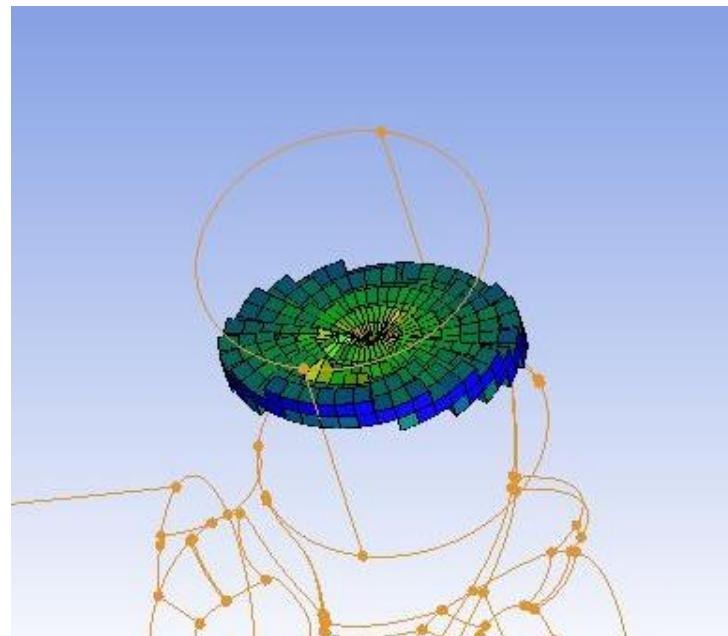


**Figure 3.7 :** Hexahedral volume mesh cross-section (2)



**Figure 3.8 :** Hexahedral volume mesh cross-section (3)

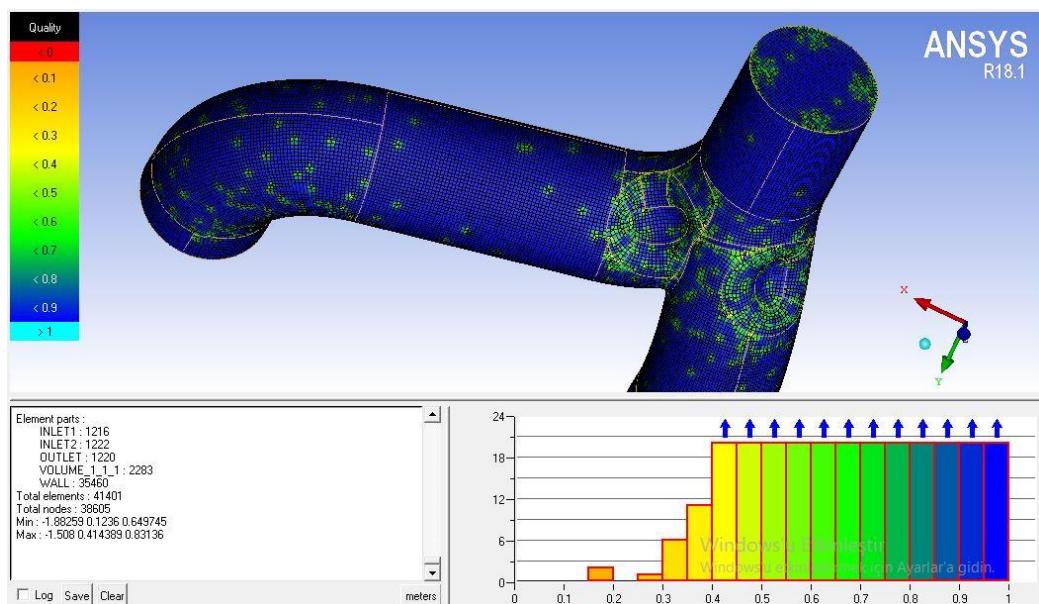
**Figure 3.9 :** Hexahedral volume mesh cross-section (4)



**Figure 3.10 :** Hexahedral volume mesh cross-section (5)

It is observed that, around the centerline of the part the quality decreases. Density box could be created to increase the element number in the interested critical points to provide a more superior mesh.

Another surface mesh type is generated on the geometry. This type is quadrilateral mesh. In this case, a lower global element seed size was chosen and higher number of elements were created. The surface mesh was checked for any error and the quality range was considered.



**Figure 3.11 :** Shell mesh information

As it is seen above, the shell mesh information is as following:

**Table 3.1 :** Shell mesh information

<b>Inlet1</b>	1216
<b>Inlet2</b>	1222
<b>Outlet</b>	1220
<b>Volume</b>	2283
<b>Wall</b>	35460
<b>Total Elements</b>	41401
<b>Total Nodes</b>	38605

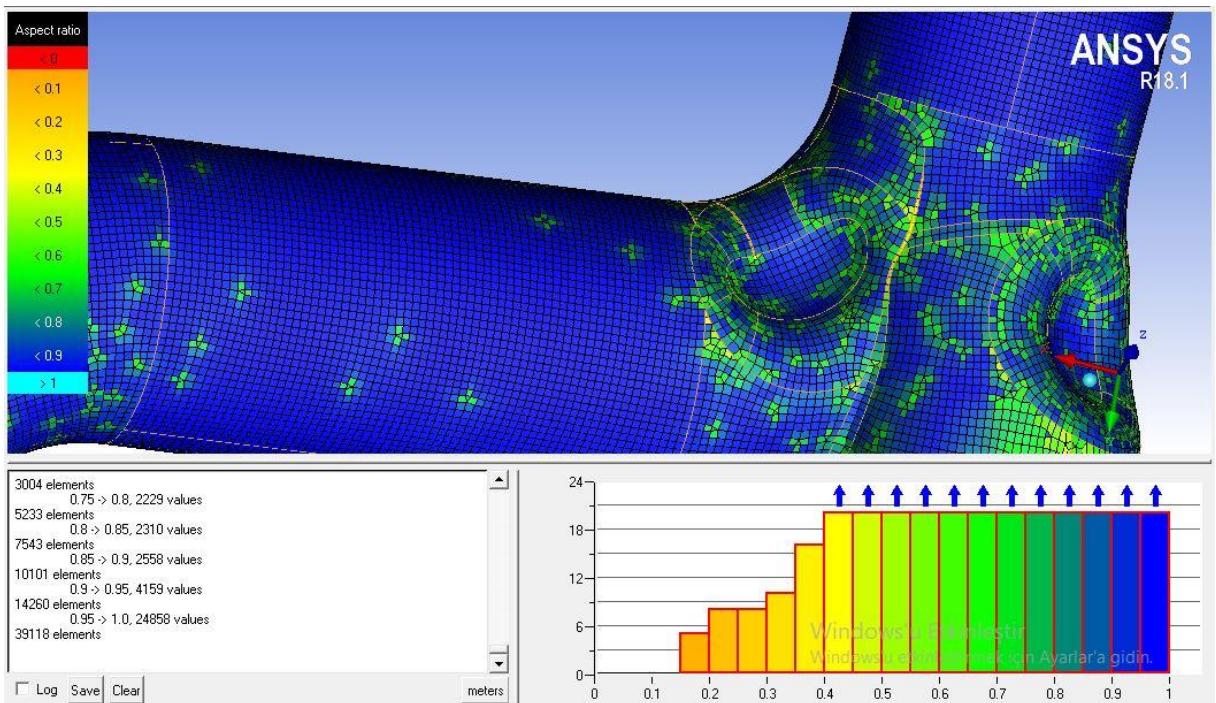
Additionally, the mesh was checked and the bad elements were minimized. The check results is obtained as following by the program output.

- No problems were found for duplicate elements.
- No problems volume elements were found for uncovered faces.
- No problems were found for missing internal faces.
- No problems were found for multiple edges.
- No problems were found for single edges.
- No problems were found for overlapping elements.
- 0 unconnected vertices were found.
- Unconnected vertices are okay

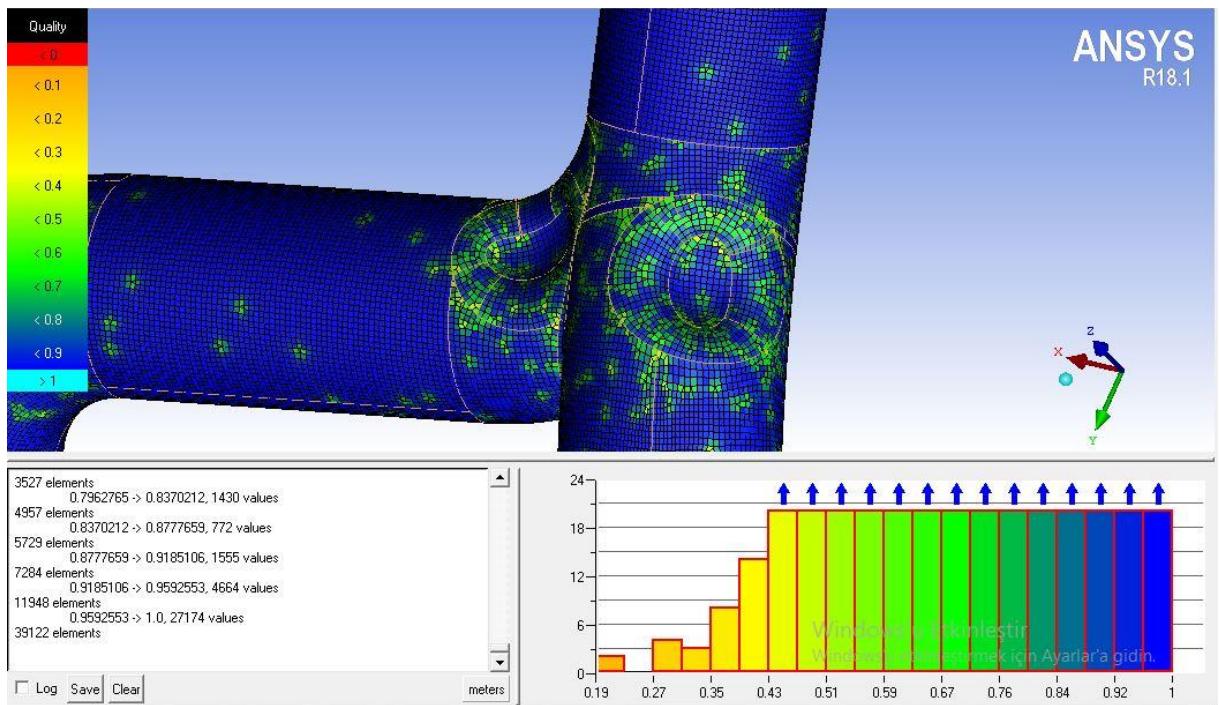


**Figure 3.12 :** Information of shell mesh

We observe the aspect ratio and the element quality of the quad shell mesh in the following outcomes. We see that, the majority of the elements have optimum aspect ratio values.



**Figure 3.13 : Quad shell mesh aspect ratio**



**Figure 3.14 : Quad shell mesh quality**

One way of increasing the quality of the shell mesh is the smoothing operation. The mesh may be smoothed to improve the overall block/mesh quality either for a certain region or for the entire model.

### **3.1.2.1.Mesh smoothing criteria**

#### **i. Determinant:**

These criteria attempt to improve the element determinant by movement of nodes, which are subject to geometry and association constraints.

#### **ii. Laplace:**

The Laplace option attempts to minimize abrupt changing the mesh lines by moving the nodes.

#### **iii. Warp:**

The warp method is based upon correcting the worst angle between two elements in the mesh.

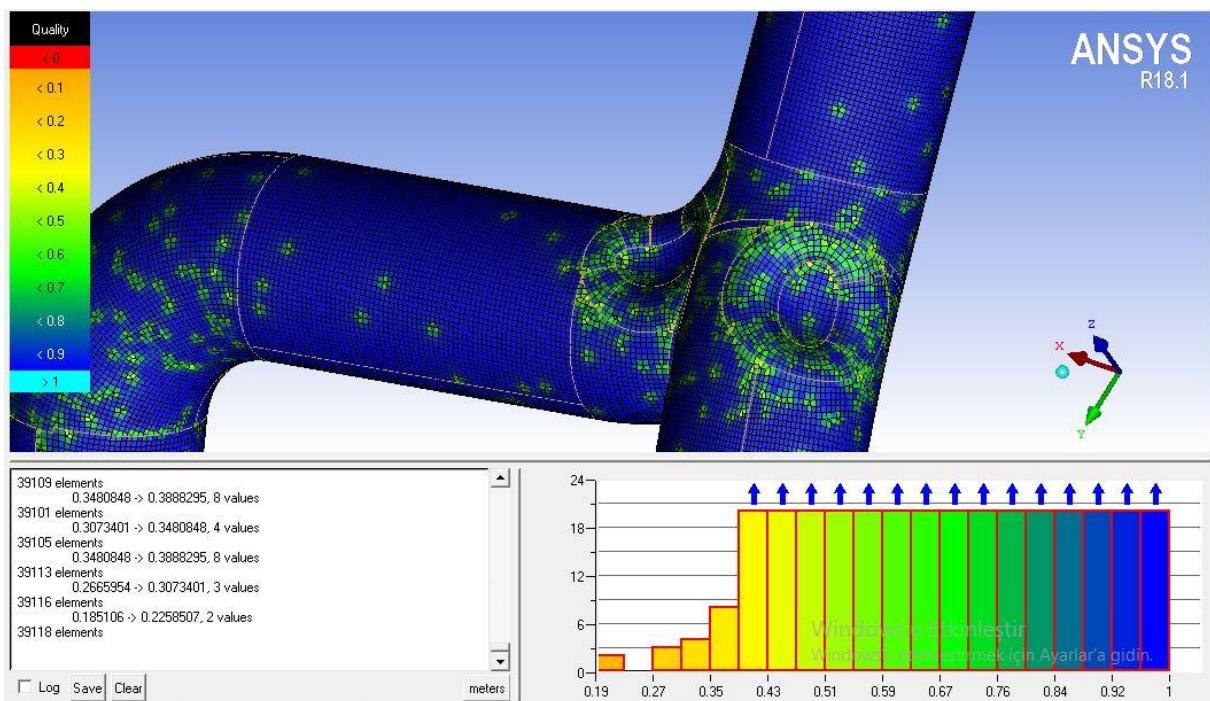
#### **iv. Quality:**

Like the determinant criteria, the quality criteria attempt to improve the elements interior angle by repositioning the nodes, which are subject to geometry and association constraints.

#### **v. Skewness:**

The skewness is defined differently for volume and surface elements. For a volume element this value is obtained by taking all pairs of adjacent faces and computing the normal. The maximum value thus obtained is normalized so that 0 corresponds to perpendicular faces and 1 corresponds to parallel faces. For surface elements, the skew is obtained by first taking the ratio of two diagonals of the face. The skewness is defined as 1 minus the ratio of the shorter diagonal over the longer diagonal. Thus 0 is perfectly rectangular and 1 represents maximum skewness.

After the smoothing applied to the shell mesh, the quality has changed and the quality is slightly increased.



**Figure 3.15 :** Quad mesh quality after smoothing

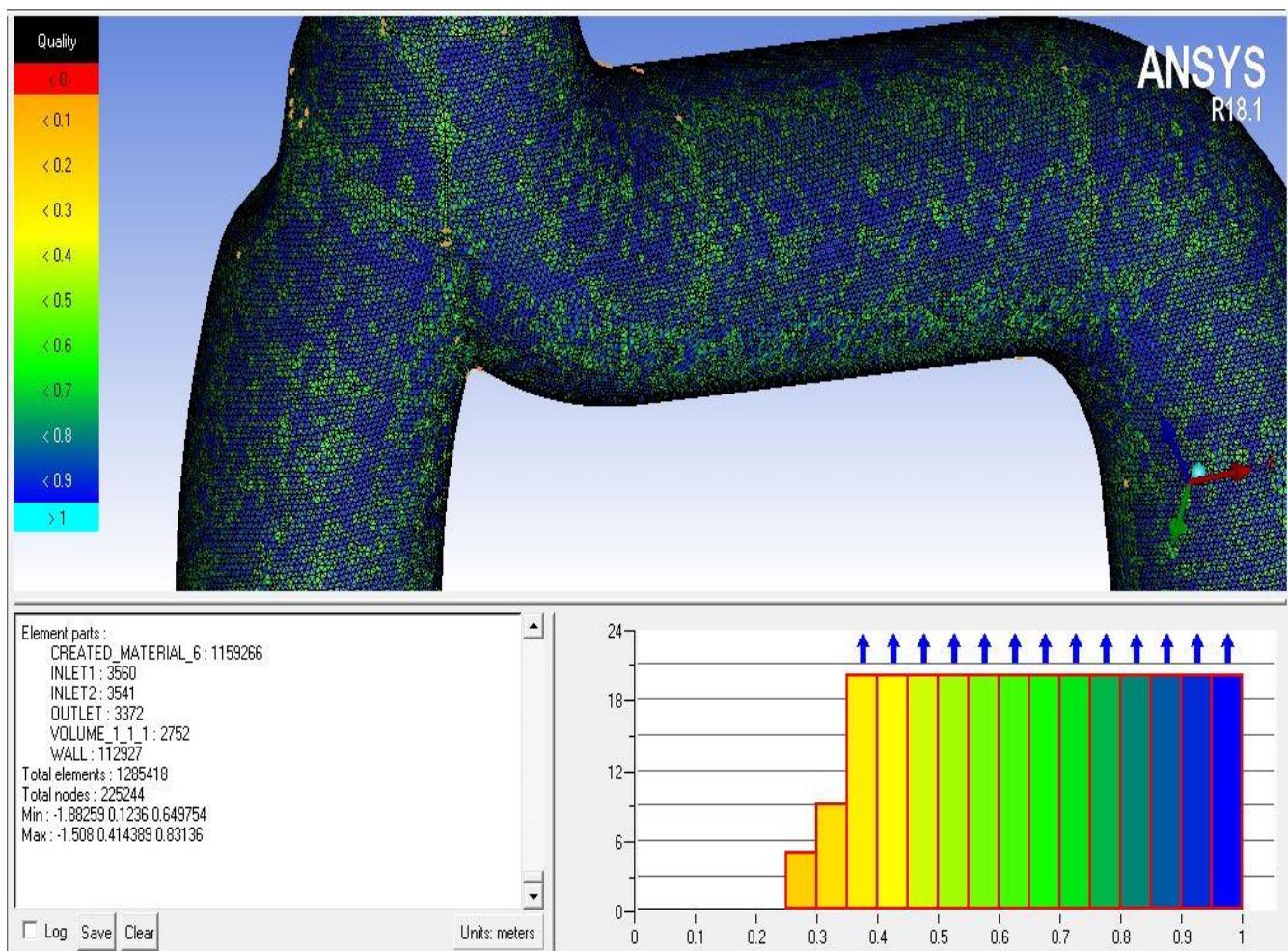
After the 2D shell meshing completed, tetrahedral volume mesh was generated. The global element seed size is set to be 0.004 for this mesh. So, the element number is obtained high. The element are set to be patch dependent and “tetra / 1 tri” option is chosen to reach the best quality. The mesh information is given as:

Inlet1	3560
--------	------

**Table 3.2 : Mesh**

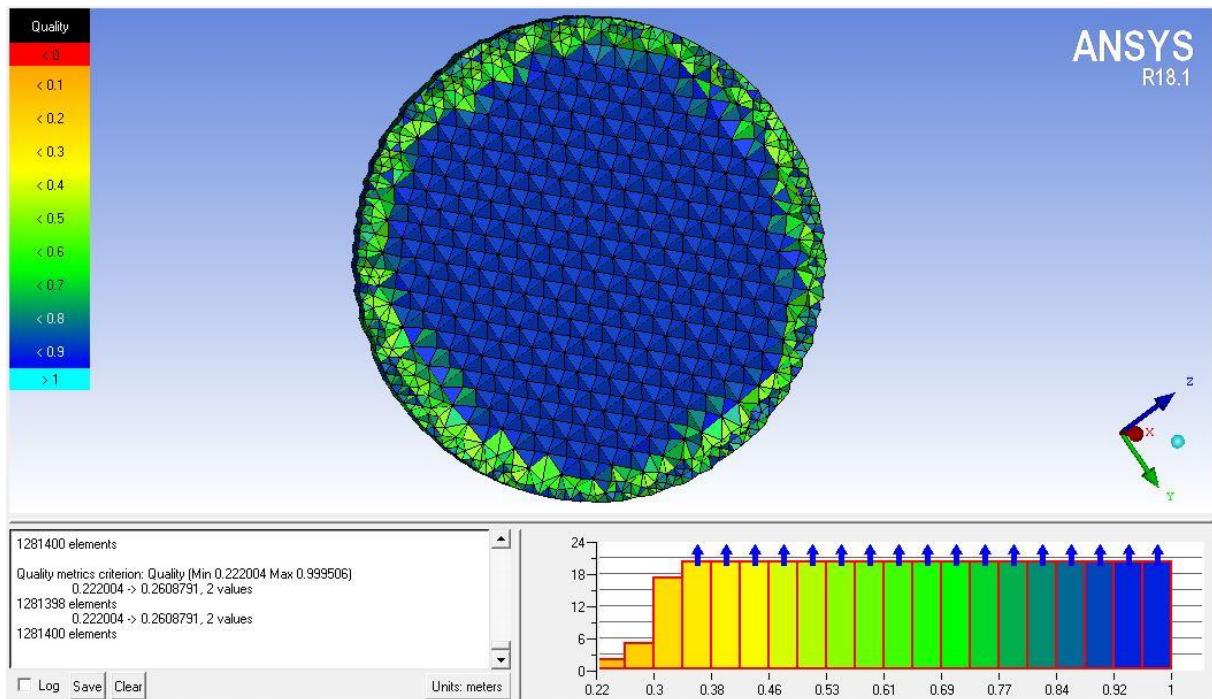
<b>Inlet2</b>	3541
<b>Outlet</b>	3372
<b>Volume</b>	2752
<b>Wall</b>	112927
<b>Total Elements</b>	1285418
<b>Total Nodes</b>	225244

information

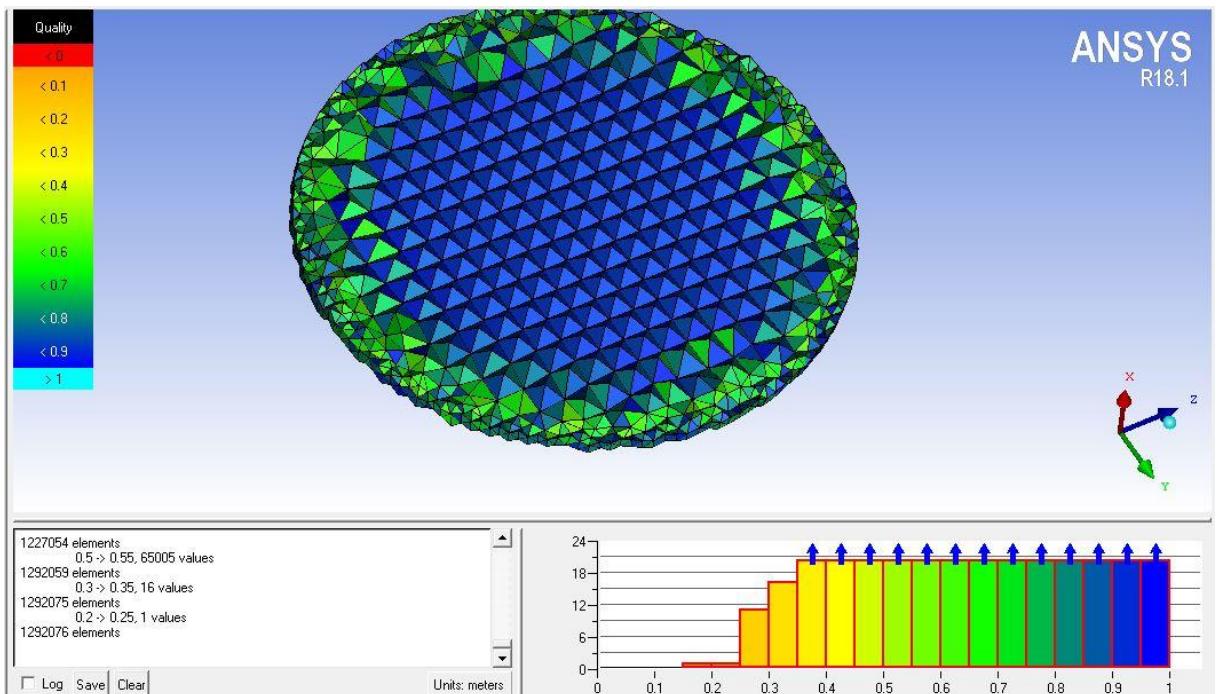
**Figure 3.16 : Tetrahedral volume mesh quality**

A section plane was created to see the volume mesh quality in more detailed. As it is seen in this part of the body, the quality is high. Along the edge of the cross section, the quality is relatively low; thus, a prism mesh application is needed on this part to make it more qualified.

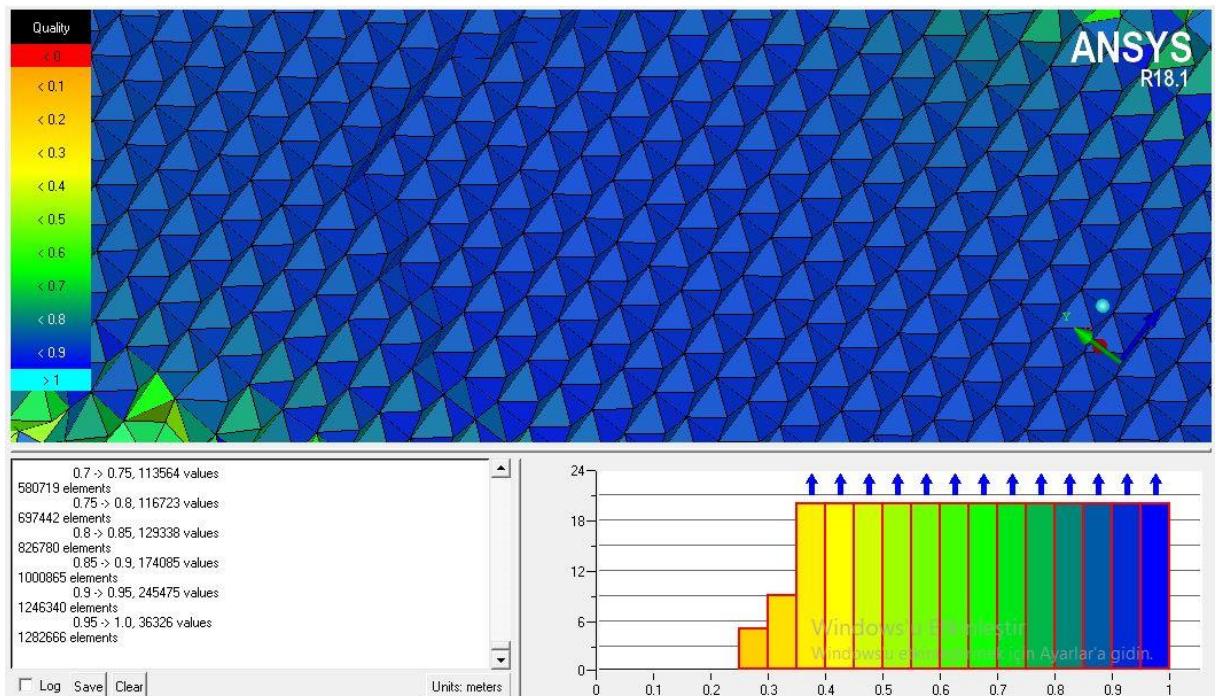
Also, smoothing needs to be applied to get the skewness and the aspect ratio better.



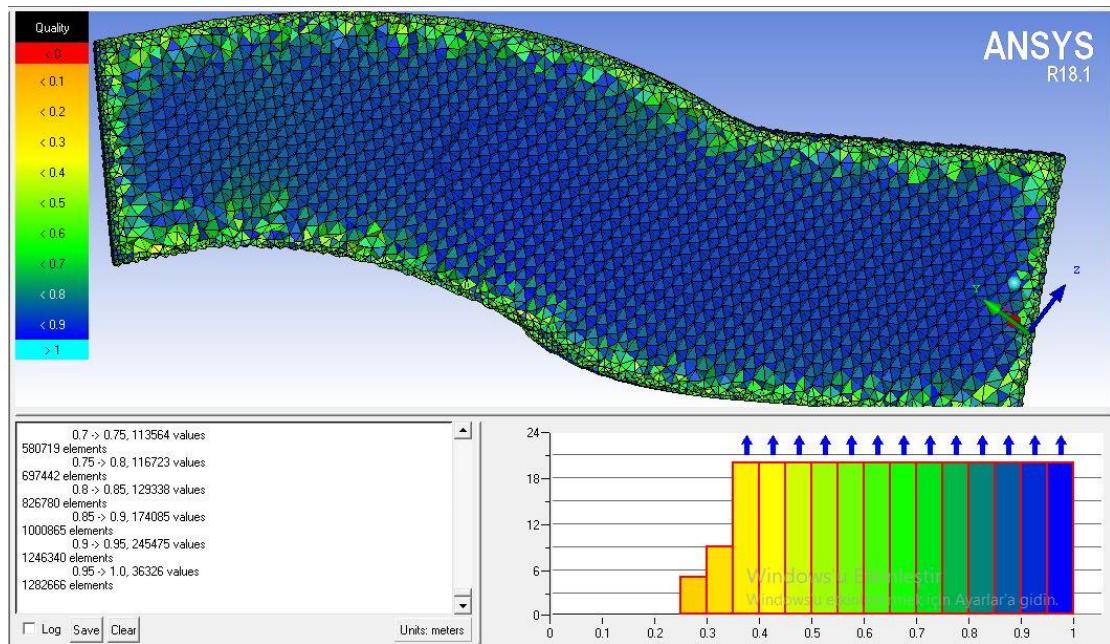
**Figure 3.17 :** Section view taken in the X plane



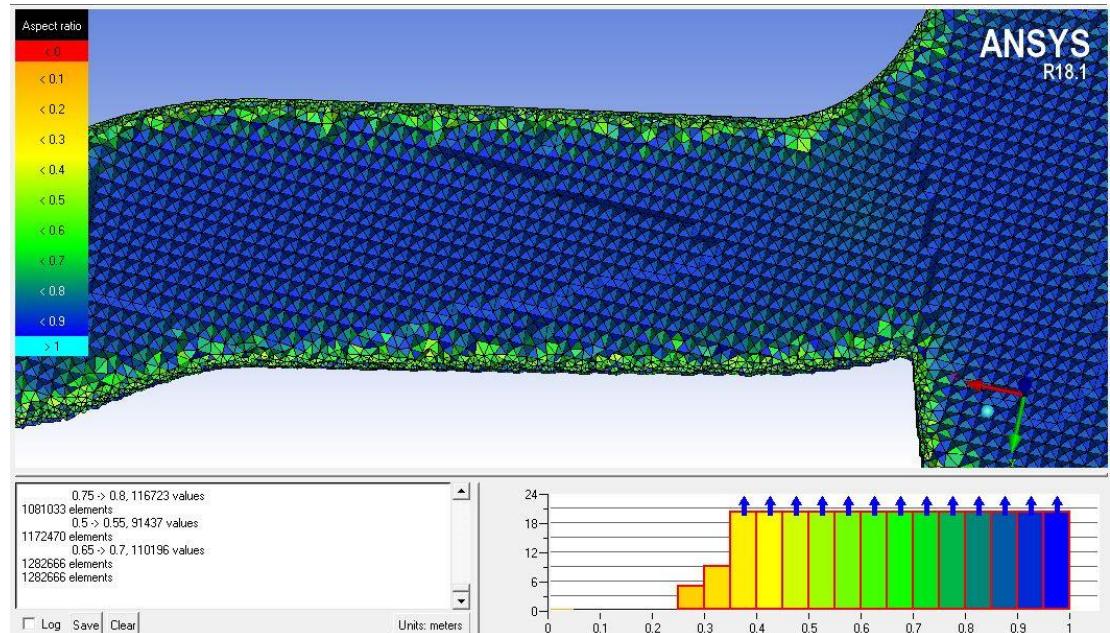
**Figure 3.18 :** Section view of volume mesh after smoothing



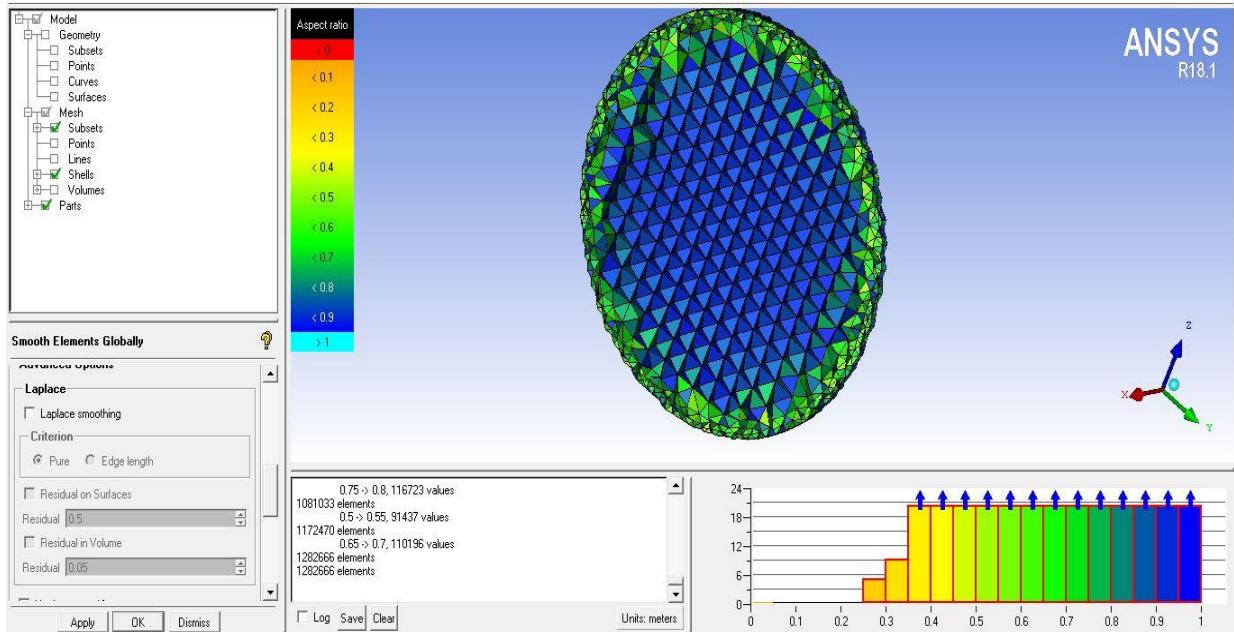
**Figure 3.19 :** Zoomed section view



**Figure 3.20 :** Tetrahedral mesh section view along the manifold tube



**Figure 3.21 :** Aspect ratio section view

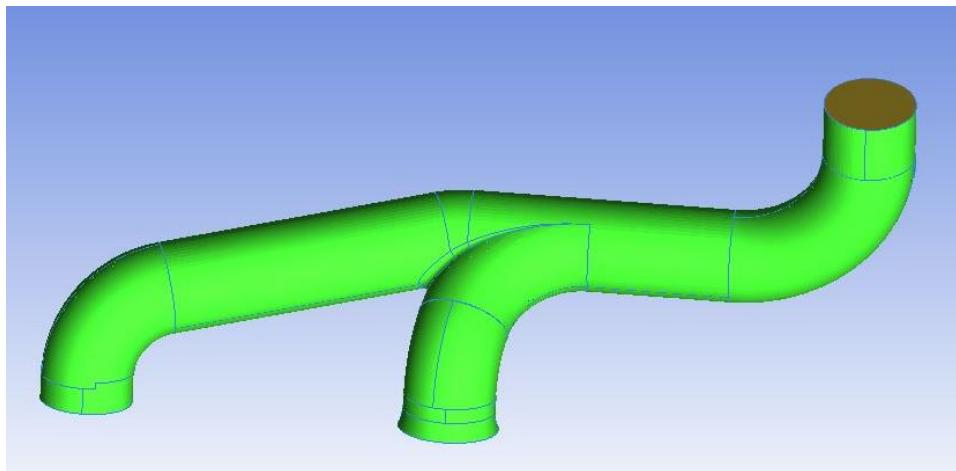


**Figure 3.22 :** Laplace smoothing applied on tetrahedral mesh

It is concluded that, for the volume mesh, more qualified tetrahedral elements could be generated in a lower amount of time. Yet, the tetrahedral mesh can be resolved in particular locations by creating prism mesh.

### 3.1.3. Mesh Generation On Part 2

The other part of the exhaust manifold was imported into Ansys Space Claim and the interior flow region was extracted. The faces were named as inlet, outlet and wall. The geometry contained unwanted parts similar to the initial geometry. Those parts were removed and their surfaces were merged with the body surface to simplify the geometry.

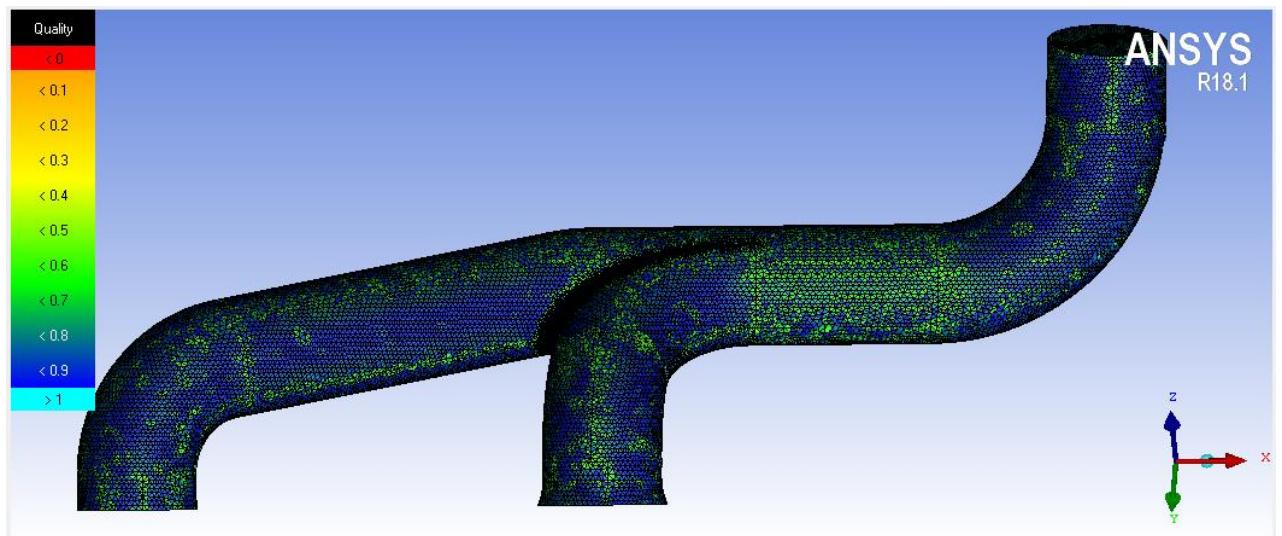


**Figure 3.23 :** Second part of exhaust manifold extracted interior volume

After simplifying and creating names for the boundaries, the meshing operation is applied. Since the other part was meshed and the results such as quality, aspect ratio, orthogonal quality, skewness and analysis results with iterations were examined, the second part was meshed quicker than the first part.

The mesh was created on ICEM CFD and the quality was checked along with errors and critical points.

The first mesh was created to be triangular on the surface and tetrahedral on volume mesh. The meshed was checked for any errors and there appeared some errors because of gaps. So, the team returned back to design modeler in order to fix the body and simplify it more. Using the *merge faces* and *delete gaps* commands, the geometry was repaired and prepared again for the mesh.

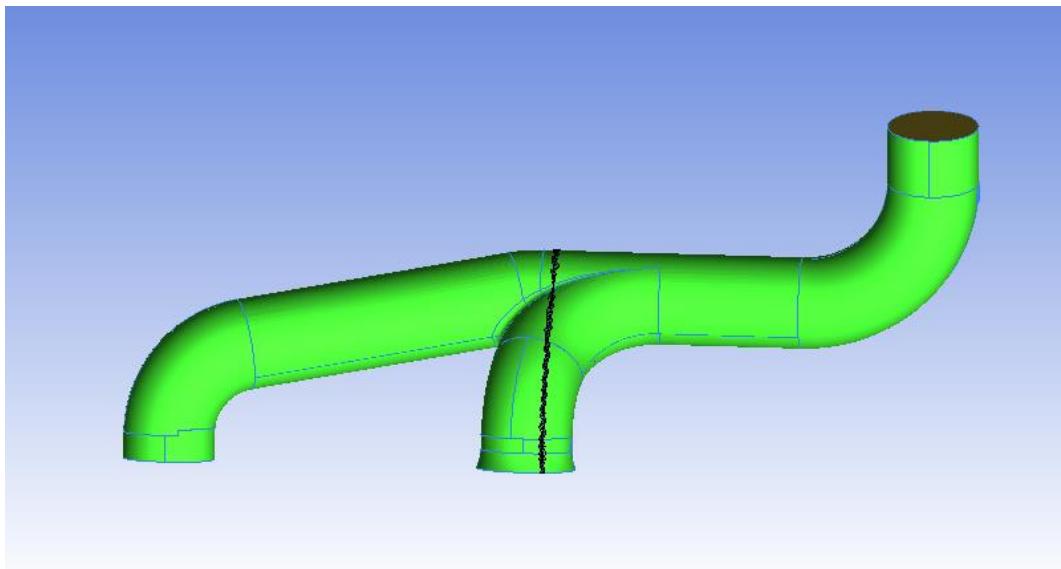


**Figure 3.24 :** Mesh generation on second part

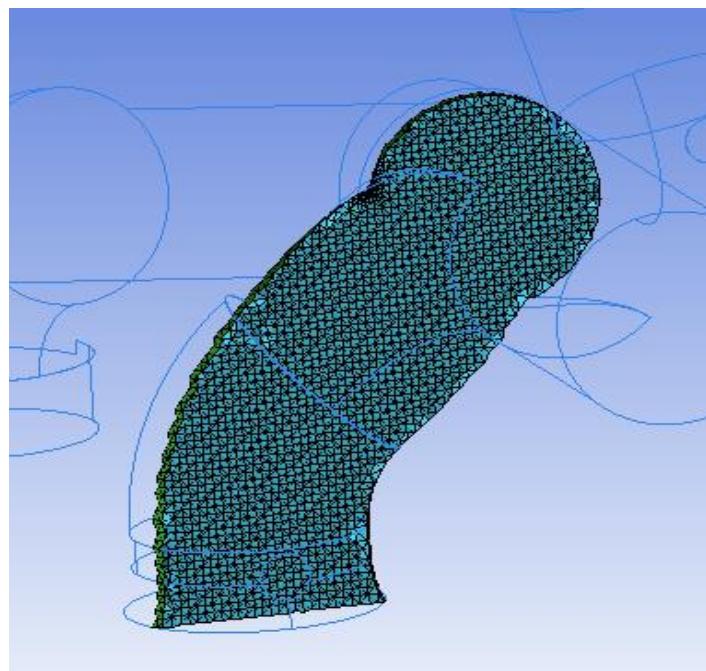
**Table 3.3 :**  
information of

Mesh  
second part

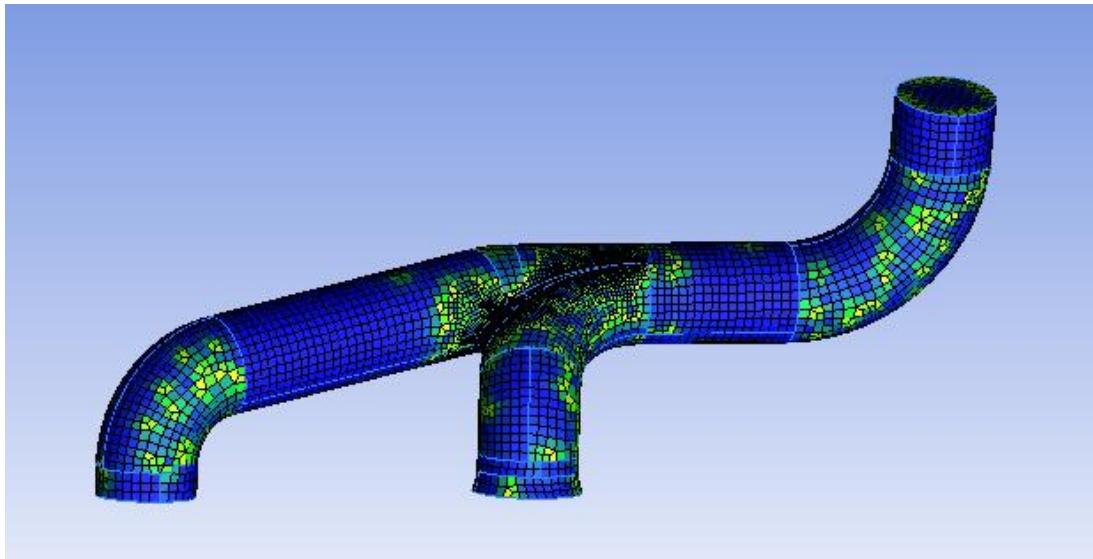
MESH INFORMATION	
<b>Inlet</b>	556
<b>Outlet</b>	871
<b>Volume</b>	2115
<b>Wall</b>	49798
<b>Total Elements</b>	1056570
<b>Total Nodes</b>	180647
<b>Global Element Seed Size</b>	0.004



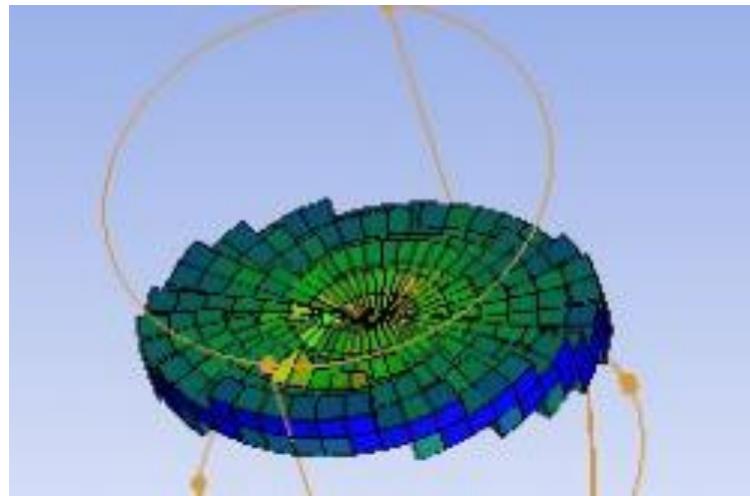
**Figure 3.25 :** Section plane to see the volume mesh



**Figure 3.26 :** Tetrahedral mesh section view on mid -x plane



**Figure 3.27 :** Quad mesh on surface and hexahedral mesh on volume



**Figure 3.28 :** Section view of hexahedral volume mesh

### 3.2.GMSH

Gmsh is an open source 3D finite element mesh generator with a built-in CAD engine and post-processor. Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input and advanced visualization capabilities.

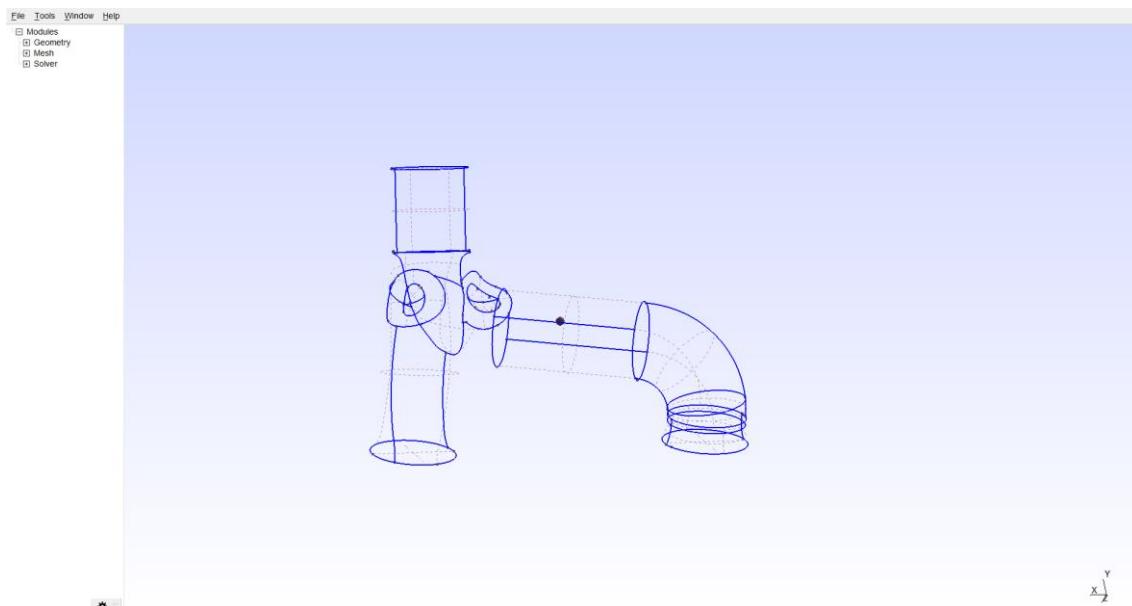
Gmsh is built around four modules:

1. Geometry
2. Mesh
3. Solver
4. Post-processing

Gmsh supports parametric input and has advanced visualization mechanisms. Gmsh supports full constructive solid geometry features, based on Open Cascade Technology. Furthermore, Gmsh can be used as a 1-, 2- and 3- dimensional mesh generator for use with Fluidity.

### 3.2.1. Starting Gmsh

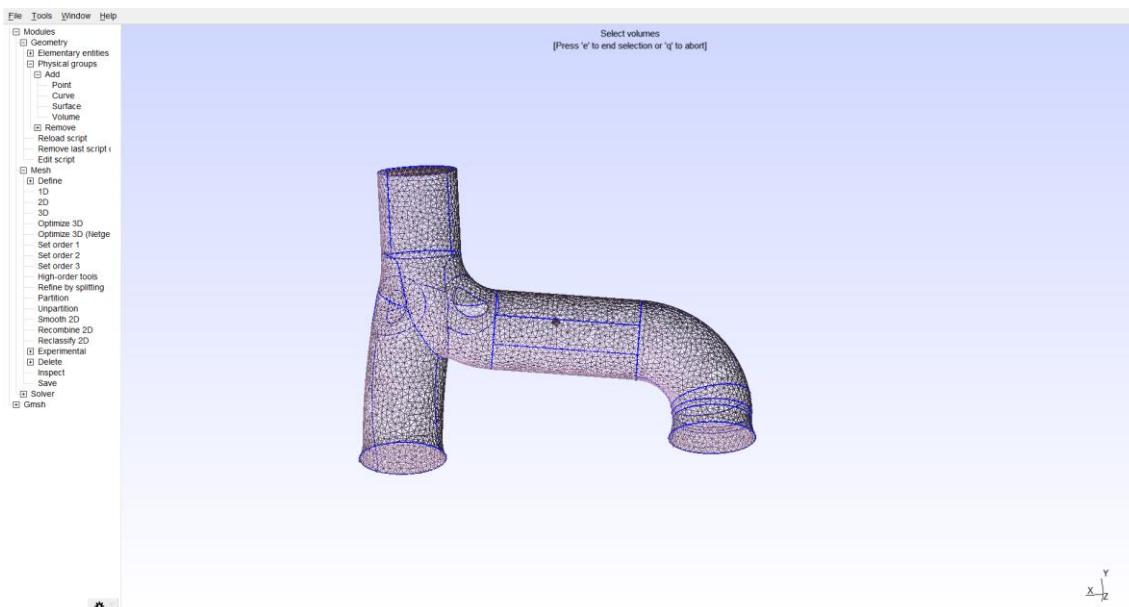
Gmsh is a free program which only occupy a place as 59.2 Mb. As we download the program from the site and open it, we come across with a simple interface involves the modules on the left panel. To start with, we import the geometry file, previously repaired through SpaceClaim, to the program as shown below.



**Figure 3.29 :** CAD data of Exhaust Manifold via Gmsh

After some adjustments the geometry has been prepared to start the mesh operations.

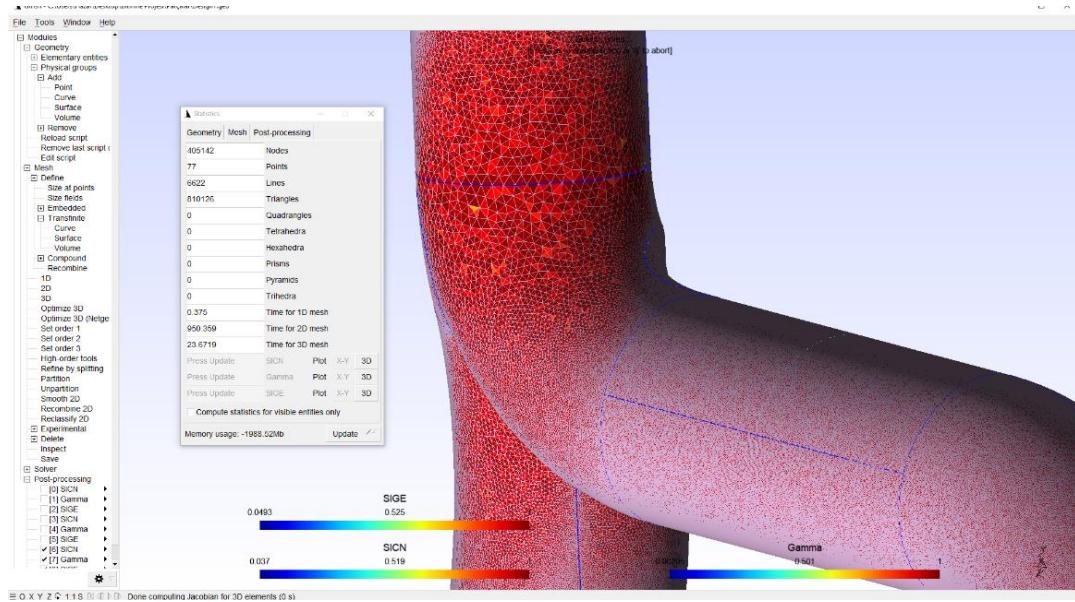
Gmsh is a mesh module regroups several 1D, 2D and 3D mesh algorithms, all producing grids conforming in the sense of finite elements. The 2D unstructured algorithms generate triangles or both triangles and quadrangles. The 3D unstructured algorithms only generate tetrahedra.



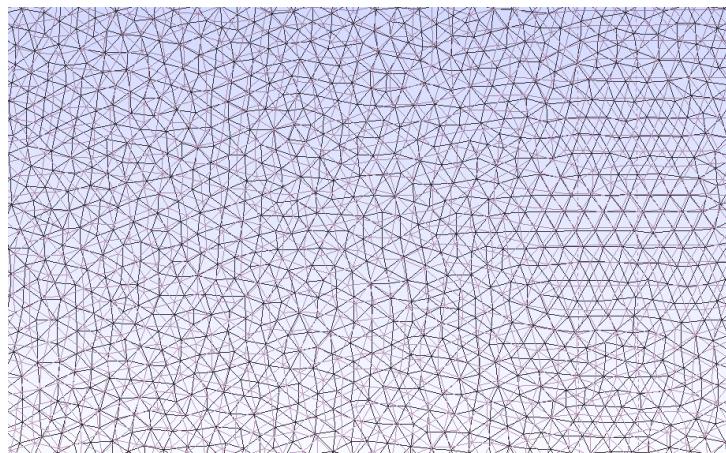
**Figure 3.30 :** Tetrahedral mesh on exhaust manifold

**Table 3.4 :** Chosen parameters for the mesh generation

2D algorithm	MeshAdapt
3D algorithm	Delaunay
2D recombination algorithm	Blossom
Subdivision algorithm	All Hexas
Smoothing steps	10
Element size factor	0.01



**Figure 3.31 :** Mesh information for tetrahedral

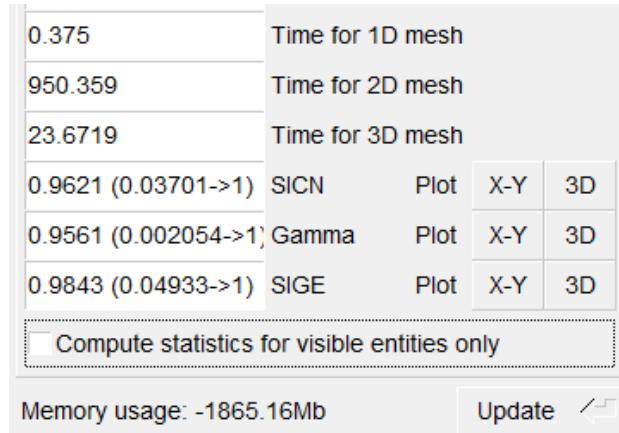


**Figure 3.32 :** Closer look to the tetrahedral mesh generated

After 3D meshing operations node number becomes 405142.

Also this program shows how much time required to complete the meshing process and some quality parameters beside. Choosing the right kind of mesh, applying the right density to critical regions and selecting the right kind of coarseness or looseness affect the accuracy and speed of the simulation process.

Total time spent on arranging the geometry and creating the mesh is approximately 10 minutes.



**Figure 3.33 :** Mesh information panel

### 3.3.Salome Software

Salome is free software that provides a generic platform for pre- and post-processing for numerical simulation. It is based on an open architecture made of reusable components. It is open source, released under the GNU Lesser General Public License, and both its source code and binaries may be downloaded from its official website.

#### 3.3.1. Features

- Supports interoperability between CAD modeling and computation software (CAD-CAE link).
- Makes easier the integration of new components on heterogeneous systems for numerical computation.
- Sets the priority to multi-physics coupling between computation software.
- Provides a generic user interface, user-friendly and efficient, which helps to reduce the costs and delays of carrying out the studies.
- Reduces training time to the specific time for learning the software solution which has been based on this platform.

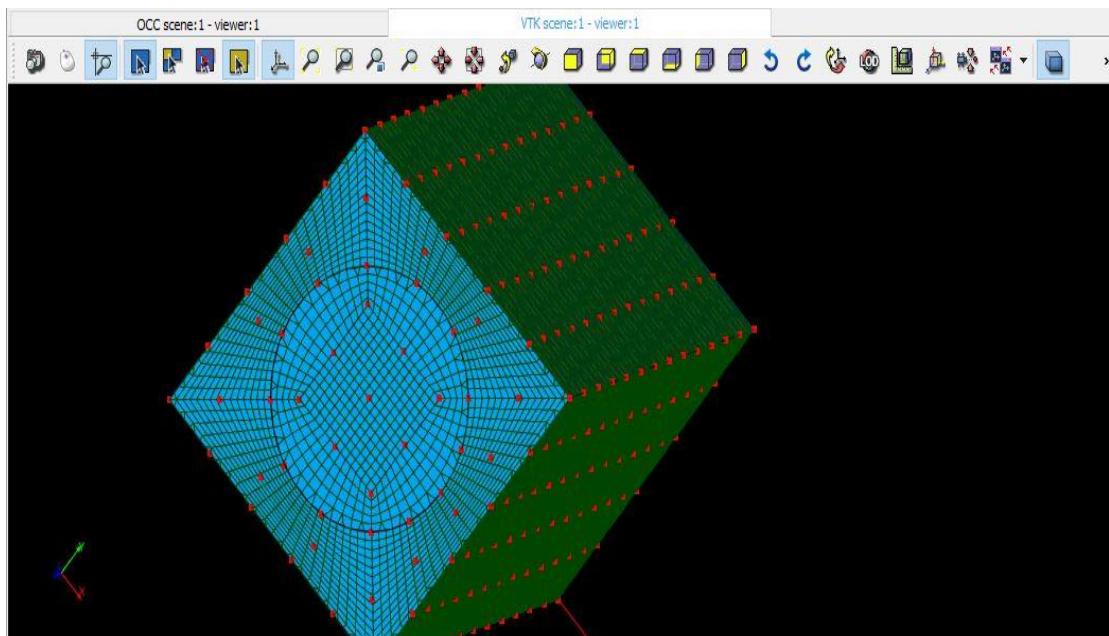
- All functionalities are accessible through the programmatic integrated Python console.

In this project, Salome software has been used to generate mesh. However, it has been a big challenge to deal with the geometry before the mesh operation, especially for the exhaust manifold. Some set of mesh was created on different geometries.

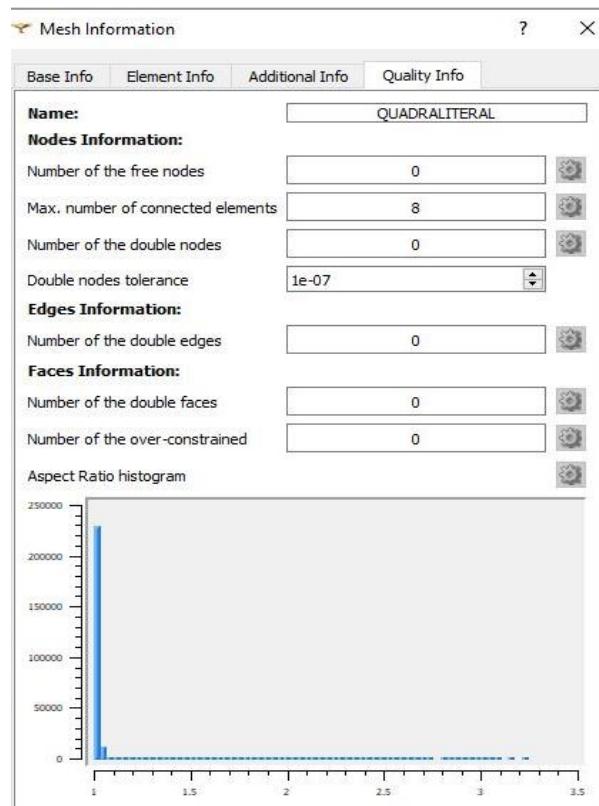
Different mesh types have been implemented on a geometry and their properties with the quality information has been saved. Some mesh types are with high quality whereas, some of them fails because of the reasons shown below the information section.

### **3.3.2. Mesh generation on a simple cubical geometry**

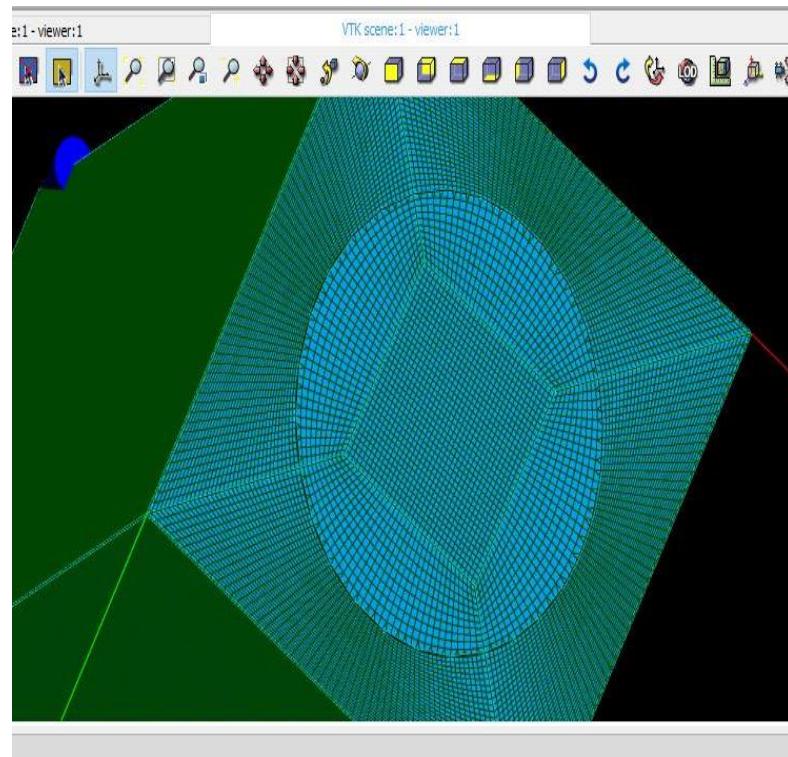
#### **3.3.2.1.Qaudraliteral mesh**



**Figure 3.34 :** Mesh generated on geometry



**Figure 3.35 :** Mesh quality information panel



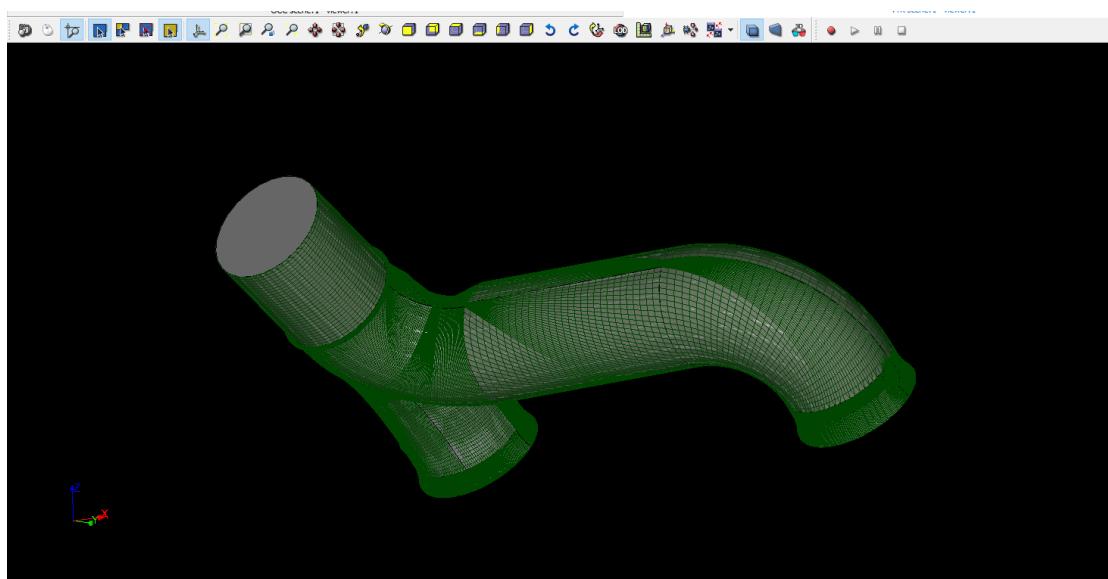
**Figure 3.36 :** Hexahedral mesh generated on cubical geometry

Mesh Information				
	Base Info	Element Info	Additional Info	Quality Info
<b>Name:</b>	QUADRALLITERAL			
<b>Object:</b>	Mesh			
<b>Nodes:</b>	1026358			
<b>Elements:</b>	Total 1042244	Linear 1042244	Quadratic 0	Bi-Quadratic 0
<b>0D:</b>	0			
<b>1Balls:</b>	0			
<b>1D (edges):</b>	10284	10284	0	
<b>2D (faces):</b>	1031960	1031960	0	0
Triangles:	1040	1040	0	0
Quadrangles:	1030920	1030920	0	0
Polygons:	0	0	0	
<b>3D (volumes):</b>	0	0	0	0
Tetrahedrons:	0	0	0	
Hexahedrons:	0	0	0	0
Pyramids:	0	0	0	
Prisms:	0	0	0	
Hexagonal Prisms:	0			
Polyhedrons:	0			

**Figure 3.37 :** Mesh base information panel

The mesh quality and aspect ratio have increased as the element size is decreased and higher quality mesh is obtained.

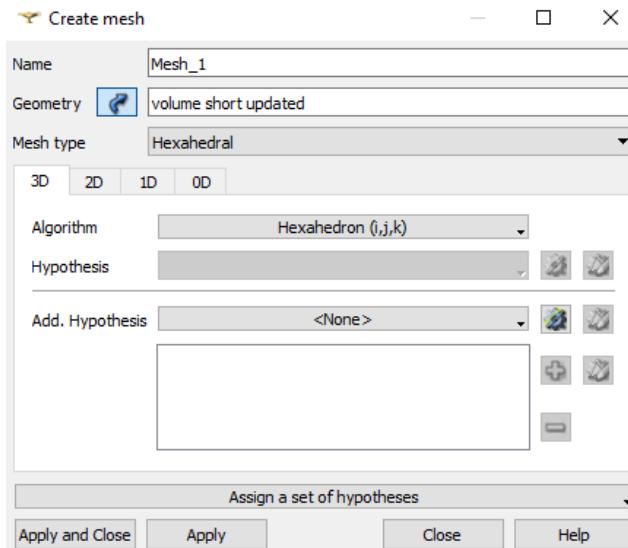
### 3.3.3. Mesh Generation Approaches on Exhaust Manifold



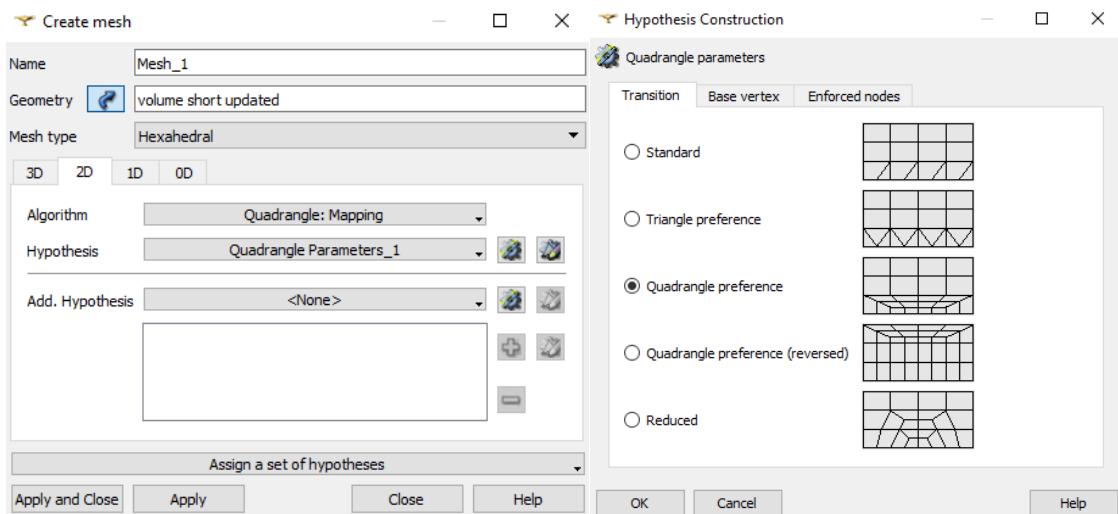
**Figure 3.39 :** Mesh generation demonstration on exhaust manifold

To create the mesh on any volume in Salome, when click on the create mesh option a window pops up that we can select the mesh type and parameters. After choosing the parameters for the geometry, we click on the apply and close and then when we click on compute mesh creates.

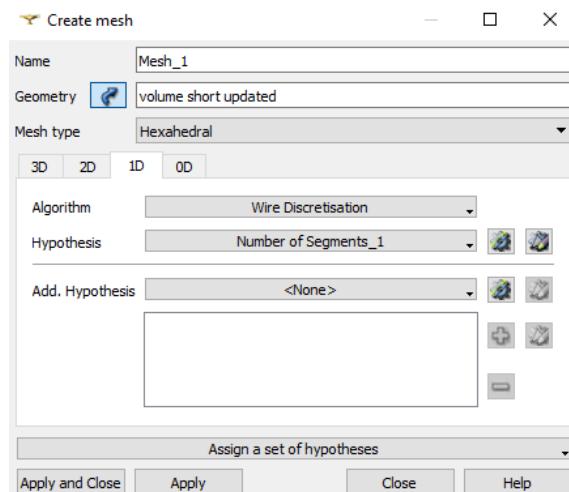
Parameters needed to create the mesh are consist of 3 section which are 1D, 2D and 3D parameters. Mesh parameters selected for the exhaust manifold shown below step-by-step.



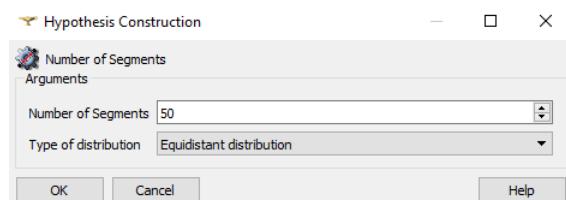
**Figure 3.40 :** Mesh generation parameter selection for 3D



**Figure 3.41 :** Mesh generation parameter selection for 2D

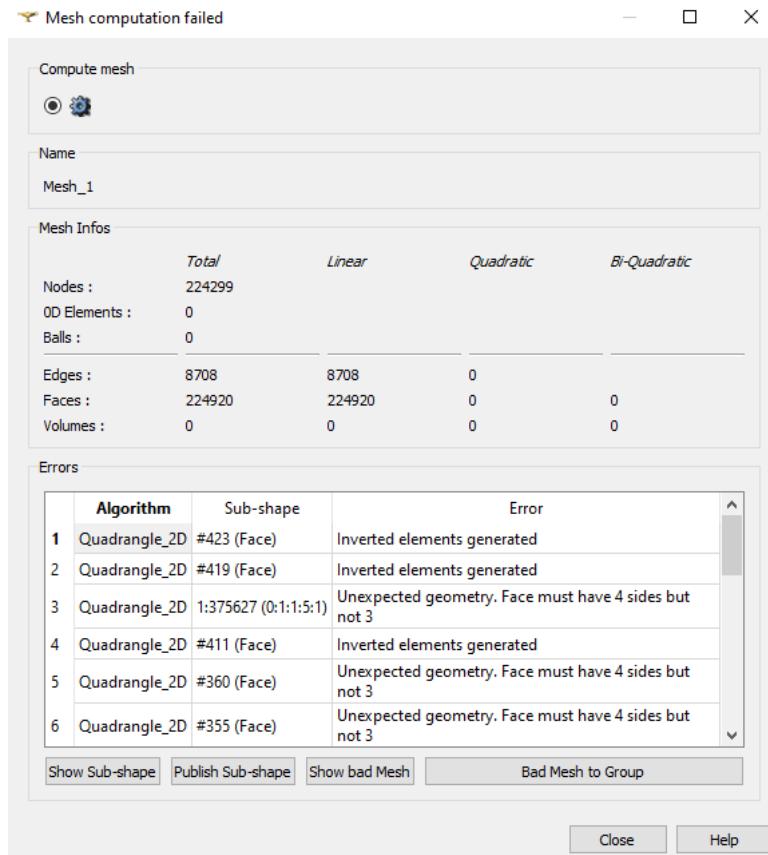


**Figure 3.42 :** Mesh generation parameter selection for 1D



**Figure 3.43 :** Wire discretization parameter selection

The exhaust manifold geometry imported to Salome software after geometry repair operations on SpaceClaim but as shown in Geometry Cleanup topic in Chapter 2 geometry could not repaired completely. Because of this disordered geometry problem, we came across some specific problems as shown in Fig1.



**Figure 3.38 :** Mesh computation error panel

Mesh generation time period is approximately 15 minutes. This time variance effected from the computer's cpu, geometry complexity of the geometry et cetera. The time spent on mesh generation could be lowered using a more powerful computer.

Salome is able to create proper hex-dominant mesh but even small defects on the part can be a problem to create the mesh on the geometry.

## **CHAPTER 4**

### **4. CONCLUSION**

This study project covered the investigation of grid generation of an exhaust manifold. The aim of the project was to generate technics to optimize the grid generation process with the best quality and lowest amount of time. The geometry of the V16 Locomotive engine exhaust manifold consists of three main parts of which their interior volume extracted to get the flow volume. Before all the procedures, the geometries needed to be cleaned up as it is crucial before the meshing operation. On each geometries there occurred some unwanted parts that were not related to volume mesh operation. These redundant parts were removed using CAD software which are Solidworks and Ansys SpaceClaim. Also the geometry was repaired before each meshing operation for the purpose of simplification and acquiring higher quality mesh.

Different types of meshes has been generated on different software to be able to compare the qualities and the time spent. For the surface shell mesh, quadrilateral, triangular and their deviations; for the volume mesh tetrahedral, hexahedral and hybrid mesh types were generated. Gmsh, ICEM CFD, Salome, Mesher are the software used throughout the project. The mesh quality of each software examined and illustrated on the detailed parts. Section views of the volume mesh was taken to see the quality distribution of mesh cell elements. The elements with errors were detected and repaired using some set of technics.

The sequence of generating the mesh is shell, volume and boundary layer. The quality of volume mesh is dependent on the shell mesh quality. .For that reason, Mesher used to generate the surface mesh automatically with minimized errors. Afterwards, the files were transferred to ICEM CFD which is capable of generating high quality volume mesh.

One way of understanding the quality of mesh was to run it and observe the analysis results. Ansys Fluent was used for the analysis where the boundary conditions were

kept constant for each iteration. As an example, the pressure distribution on the manifold was converging for a constant value as the mesh quality was increased.

## 5. REFERENCES

- [1] Liu, Hengjin (2006) 3-d mesh generation for finite-element modelling of complex natural structures, Master thesis, Mc Gill University Biomedical Engineering Department, Ottawa, Canada
- [2] Monan Wang, Jian Gao & Xinyu Wang (2017) High-quality mesh generation for human hip based on ideal element size: methods and evaluation, Computer Assisted Surgery, 22:sup1, 212-220
- [3] Walton, Kirk S. (2003) Sculpting: an improved inside-out scheme for all hexahedral meshing, master thesis, Brigham Young University Department of Civil and Environmental Engineering, Provo, Utah
- [4] Vivekanand Navadagi, Siddaveer Sangamad, (2014) CFD Analysis of Exhaust Manifold of MultiCylinder Petrol Engine for Optimal Geometry to Reduce Back Pressure, International Journal of Engineering Research & Technology (IJERT), Vol. 3 Issue 3
- [5] Deger Yasar, (2017) Coupled CFD-FE-Analysis for the Exhaust Manifold of a Diesel Engine
- [6] Kanupriya Bajpai,(2017) CFD Analysis of Exhaust Manifold of SI Engine and Comparison of Back Pressure using Alternative Fuels, Mechanical Engineering Department, BIT Durg, India
- [7] Dr. Bipin Pai, Yang Wang, Dong Fu, & Dr. Chenn Q. Zhou (2012) Analysis and Visualization of Thermal stresses in exhaust manifolds, Purdue University Calumet, Hammond, IN 46323
- [8] Sadrehaghghi, Ideen, Mesh Generation in CFD, Patch 1.86.7, CFD Open Series
- [9] Schneiders, Robert, Algorithms for Quadrilateral and Hexahedral Mesh Generation, Aachen, Germany



# **Curriculum Vitae**



## **ABDULHALİM ASLANAÇIER**

Abdurrahman Ay sok.

Sefa Apartmanı NO:4 GÜNGÖREN – İSTANBUL

Tel: 05537590863

E-posta: halimaslanacier@gmail.com

### **KİŞİSEL BİLGİLER**

Doğum Tarihi: 06.12.1996

Doğum Yeri : Mardin

Medeni Hali: Bekar

Ehliyet: B sınıfı

### **EĞİTİM BİLGİLERİ**

2015 - 2020 Marmara Üniversitesi, İstanbul

Lisans, Makine Mühendisliği(İngilizce)

Akademik Başarı: 3,00 / 4,00

### **Staj Deneyimi**

03.06.2018 – 20.08.2018 – Üretim Stajı

AGTEKS, İstanbul

CNC, Kaynak, Büküm departmanlarında çalışılıp tasarım ofisinde tecrübe kazanıldı.

## **Değişim Programı**

06.02.2019- 15.07.2019 – Erasmus Programı – Universita Degli Studi Di Brescia -  
Italya

## **Bilgisayar Bilgisi**

Microsoft Office, Matlab, Solidworks, Autocad, Inventor, Ansys Fluent, Ansys Icem  
CFD, CFX

## **Yabancı Diller**

YDS Puanı: 80 (B)

	Okuma	Yazma	Dinleme	Konuşma
İngilizce	C1	C1	B2	C1
İtalyanca	A2	A2	A2	A2

## **Projeler**

- Tubitak Uluslararası İnsansız Hava Aracı Projesi 2020, Mekanik Tasarım Ekibi
- Investigation of grid generation strategies for Exhaust Manifold – Bitirme Projesi

## **Etkinlikler**

Mechanical Engineering Society(MES)

ISMEK – İş Hayatında İletişim Kursu

Istanbul Sanayi Odası Etkinlikleri

Corso Di Lingua Italianca Sertifikası

Satranc Kulübü

## **Hobiler**

Satranc , Atletizm, Basketbol

# **Curriculum Vitae**



**GÖKSU HAZAR**

Kadıköy / İstanbul

Tel : 0534 953 06 85

E-posta : goksuuhazar@gmail.com

Linkedn : [www.linkedin.com/in/göksu-hazar-b18204117](https://www.linkedin.com/in/göksu-hazar-b18204117)

## **KİŞİSEL BİLGİLER**

Cinsiyet : Kadın

Doğum Tarihi : 07/11/1997

Doğum Yeri : Sakarya

Medeni Hali: Bekar

Ehliyet: B sınıfı

## **EĞİTİM BİLGİLERİ**

2015 – devam ediyor Marmara Üniversitesi, İstanbul

Lisans, Makine Mühendisliği(İngilizce)

Akademik Başarı: 2,53

## **STAJ DENEYİMİ**

19/08/2019–16/09/2019 Intern Toyota Otomotiv Sanayi Türkiye A.Ş., Sakarya

01/07/2019–29/07/2019 Intern Alimex Alüminyum A.Ş., Sakarya

## İŞ DENEYİMİ

11/2018, Autodesk, Student Expert

## KİŞİSEL BECERİLER

	Okuma	Yazma	Dinleme	Konuşma
İngilizce	C1	C1	C1	C1
Fransızca	A2	A2	A2	A2
Almanca	A1	A1	A1	A1

## PROJELER

- Investigation of grid generation strategies for Exhaust Manifold – Bitirme Projesi
- Jet motoru dizayn
- “Tıp için Tasarla” yarışması ve kampına katılım.
- Çek Cumhuriyetinde sanat alanında gönüllü çalışma kampına katılım.

## BAŞARI VE ÖDÜLLER

- Autodesk Dijital Tasarım Sertifikası
- Autodesk DesignNow 3B İstanbul Taksi Modelleme Yarışması 2.lük Ödülü
- Yüzme Serbest Stil Bayrak Yarışı Sakarya 2.lük Ödülü

## ORGANİZASYONEL/ YÖNETİMSEL BECERİLER

- Makine Mühendisliği Günleri Etkinlik Lideri, 2019

## DİJİTAL BECERİLER

Ansys, Autocad, Matlab, Solidworks, Fusion 360, Photoshop, Microsoft Office,

## DİĞER BECERİLER

- CMAS 1 Yıldız Dalıcı

## **ÜYELİKLER**

Makine Mühendisliği Kulübü (Başkan yardımcılığı 2018-2019)

Bilim ve Enerji Kulübü (Başkan yardımcılığı 2017-2018)

Makine Mühendisleri Odası

Mamara Üniversitesi Sualtı Sporları Kulübü

Mamara Üniversitesi Tenis Kulübü

## **İlgili Alanları**

3 boyutlu tasarım, yüzme ve dalış sporları, keman ve piyano, bisiklet, puzzle.