

Computational Fluid Dynamics:  
from zero to guru

Alexander Yun

# Computational Fluid Dynamics: from zero to guru

Author: A. Yun, Dr.-Ing., PhD

Technical editors: A. Maltsev, Dr.-Ing.  
C. Semler, PhD

Editors: V. Makerova  
O. Varnavskaya, PhD.

Technical consultants: D. Dankin, Dipl. Eng.  
M. Shcherbakov, Dipl. Eng.

Illustrators: V. Stolyarova  
O. Sytnik

Typographer: G. Yun

No part of this book may be reprinted, reproduced, transmitted or utilized in any form by any electronic, mechanical photocopying, recording, scanning, or otherwise, without prior written permission of the author.

All products names, logos, and brands are the property of their respective owners.

# Contents

	Preface	vi
	Nomenclature	viii
<b>I</b>	<b>Zero level...</b>	
	0. Welcome or bla-bla-bla!	1
	<b>Theoretical part - I</b>	
	1. The language	4
	2. Grammar	11
	<b>Practical part - I</b>	
	3. Mesh and boundary conditions	15
	4. First project (WB + CFX)	18
<b>II</b>	<b>Beginner and a bit more...</b>	
	5. Level up!	53
	<b>Theoretical part - II</b>	
	6. Mathematical language	56
	7. Conservation laws	67
	8. Turbulence	76
	9. Simulation based on the statistical averaging	79
	9.1. Boussinesq approximation	81
	9.2. Algebraic models	83
	9.3. One-equation models	88
	9.4. Two-equation models	92
	9.5. Heat and mixing modeling	106
	9.6. Near-wall treatment	107
	10. Introduction to structural analysis	114
	10.1. Linear static analysis	118
	10.2. Nonlinear static analysis	119
	10.3. Fatigue	120
	10.4. Buckling	128
	11. Analytical methods	129
	11.1. Hagen-Poiseuille flow	129
	11.2. Couette flow	131
	11.3. Beam fixed at one end	134
	11.4. Column buckling	137
	12. Experimental methods for flows and structures	139
	12.1. Flow parameters measurements	140
	12.2. Structural parameters measurements	146

	<b>Practical part - II</b>	
	13. Geometry creation: Design Modeler, Space Claim	151
	14. Mesh generation: ANSYS Mesh	176
	15. Solution: Fluent, Icepak, Mechanical	190
	16. Post-processing: CFD-Post	252
<b>III</b>	<b>Experienced and a bit less...</b>	
	17. Up, up, up!	258
	<b>Theoretical part - III</b>	
	18. Simulation based on the statistical averaging (cont.)	259
	18.1. Nonlinear models	259
	18.2. Explicit algebraic Reynolds stress models	262
	18.3. Reynolds stress models	270
	18.4. Low-Re number effects	277
	18.5. Unsteady RANS	283
	18.6. Large Eddy Simulation	284
	18.7. Hybrid modeling	291
	18.8. Direct Numerical Simulation	293
	19. Non-Newtonian fluids	298
	20. Two-phase flows	304
	21. Radiation	311
	22. Acoustic	324
	23. Electro-magnetism	330
	24. Combustion	332
	25. Dynamic structure analysis	364
	<b>Practical part - III</b>	
	26. Mesh generation: ICEM CFD	377
	27. Solution: Fluent, Rigid Body, Modal, Harmonic	405
	28. Post-processing: CFD-Post	434
	29. Review of CAD programs	440
	30. Review of mesh generation programs	443
	31. Review of CFD programs	447
	32. Review of structural analysis programs	452
	33. Review of post-processing programs	454
	34. Review of other programs	456
<b>IV</b>	<b>Guru and much less...</b>	
	35. I know that I know nothing!	459

## **Theoretical part - IV**

36. Numerical procedure for flow modeling	461
36.1. Finite volume method	462
36.2. Coordinate transformation	465
36.3. Discretisation form for diffusion term	469
36.4. Discretisation form for convective term	471
36.5. Discretisation form for unsteady term	472
36.6. Discretisation form for source term	473
36.7. Interpolation methods	475
36.8. General system equations	478
36.9. Pressure-velocity coupling	479
36.10. Boundary conditions	483
36.11. Solution method	488
37. Numerical procedure for structural analysis	492
38. Numerical procedure for mesh generation	494
39. Alternative ways to describe turbulence	498

## **Practical part - IV**

40. Solution: FSI, scientific configurations	500
41. Post-processing: Gnuplot	544
42. Open source programs	548
References	552
Name index	564
Demonstration cases index	569
Appendix	572

## **PREFACE**

Computational Fluid Dynamics (CFD) and structural analysis play a significant role in the development of technical devices, building construction, weather predictions, biochemistry processes modeling, and in many other fields. With regard to increase computational power increase and improvements in computer modeling techniques, it is expected that the numerical simulations will prevail the traditional methods, such as the experiments and analytical solutions, in the near future. Behind computer modeling, there are complex mathematical apparatuses, physical theories, chemical reactions, etc. Together, these factors make it difficult to understand and use CFD and structural analysis. This book attempts to systematize and provide an easy explanation of computer modeling.

The book consists of four parts. Each part has two sections: theoretical and practical. The first part represents the absolute beginner level. All explanations are made by words without equations. A simple CFD simulation is introduced in the practical section. The second part has more sophisticated introduction to CFD. The basic equations and theories are considered. The usage of CFD and structural analysis software for general physics is shown in the practical section. The third part gives the information about advanced modeling and introduces different physics, such as combustion, radiation, two-phase flows, acoustics, etc. Simple demonstrations of advanced models, extended physics and review of existed software packages to model flows and structures are shown in the practical section. The last part concentrates on the numerical solution of equations described in previous parts. The practical section has a demonstration of fluid and structure interaction, scientific examples. The important equations, dimensionless numbers and the map of demonstration cases are at the end of book.

The author thanks the technical editors Dr.-Ing. A. Maltsev and Dr. C. Semler for review and helpful comments, editors V. Makerova and Dr. O. Varnavskaya for text correction, technical consultants D. Dankin and M. Shcherbakov who went through all demo cases and made many suggestions and corrections, illustrators V. Stolyarova and O. Sytnik who prepared the illustrations, and typographer G. Yun who typed the text and formulas. The author thanks TU Darmstadt (Germany) for providing the scientific package FASTEST-3D, Simutech Group (Canada) and Dr. C. Semler for ANSYS package, Mentor Graphics (US) and A. Kharitonovich for FloEFD. Many thanks go to Dr. S. Akramov, the Alizadas, G. Drappel, Dr. S. Kim, the Kubekovs, K. Woo, the Zamyatins, and A. Yun Jr. for moral support and advices. Finally, special thanks to my family for continuous support and giving time to write this book.

All recommendations, wishes and corrections please send to email: [contact@yun.su](mailto:contact@yun.su).

Alexander Yun

## PART I. Zero level...

### Chapter 0. Welcome or bla-bla-bla!



Welcome to *CFD* world. Here, we will study *CFD* and a bit of *structural analysis*. To begin with, we need to answer two main questions. What is *CFD* and why do we need it? *CFD* is an abbreviation. *C* is for Computational, in other words, a calculation made by using computers. *F* is for Fluid, something that can flow. *D* is for Dynamics the behavior of the flow. Therefore, *CFD* is a modeling of fluid motions made by using computers. The second question might have two contrasting responses, the negative one implying that we do not need *CFD* and the positive one suggesting that it is very necessary. The answer depends on the context. In general, there are other methods to research or observe the flow motion. However, to gain a deep understanding of the flow behavior *CFD* might be used. Moreover, *CFD* modeling is necessary to reduce the development cost and increase the production output when the flow plays an important role.

Apart of *CFD*, there are two other possible ways to research the fluid behavior: *experimental* and *analytical* methods. In the first method, natural experiments are used instead of flow modeling on computers. For example, to create a new car, it is first necessary to create a prototype. Thereafter, for example, some parameters, such as fuel consumption, and a stable car movement have to be checked through experiments. During this process, some ideas to improve the car performance may appear. Therefore, it is necessary to apply these ideas in the development of the next prototype, and recheck various parameters until a final design is obtained that satisfies all requirements



## Chapter 0. Welcome or bla-bla-bla!

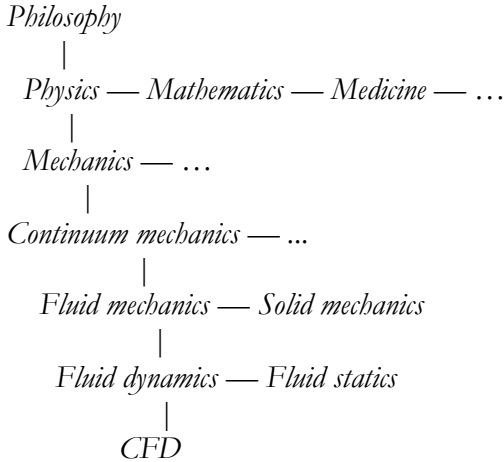
and has the best performance. In recent times, this method is rather expensive and difficult since technical devices have become more complex and advanced. However, for some simple devices, the experimental method can be used alone. The second method is to arrive at an analytical mathematical solution or a law that describes the flow motion. This method is usually used for standard or simple cases. It is very difficult to apply the analytical method for complex cases. However, it can be used as a preliminary research with some assumptions or simplifications. In practice, researchers and engineers use all the three methods mentioned above to study fluid behavior or develop technical devices. The pros and cons all three methods are summarized in Table 1. The table does not include many specific cases. With work experience, it would be possible to distinguish and choose the optimal method or a combination of methods.

Method	pros	cons
<i>CFD</i>	cheap, fast, detailed information	not real, can have a modeling error
<i>Experimental</i>	real	expensive
<i>Analytical</i>	cheap, detailed information	not possible for complex configuration

Table 1. Advantages and disadvantages of different methods to describe fluid behavior.

Now, we will define the place of *CFD* in the scientific world. *CFD* is interconnected with many other sciences, such as mathematics, physics, computer science, and chemistry. The genealogy tree below describes *CFD* relationships with other sciences.

## Chapter 0. Welcome or bla-bla-bla!



*CFD* is a part of *fluid dynamics* that describes fluid motion. In contrast, *fluid statics* studies fluid without motion. *Fluid dynamics* and *fluid statics* together form *fluid mechanics*. *Mechanics* comes from the ancient Greek word μηχανικός (mēkhanikós) that refers to the art of building a machine. Apart from *fluid mechanics*, there is *solid mechanics* that studies the behavior of solid materials. Similar to *CFD*, *structural analysis* is applied to solid materials by using computers for modeling solid structures. In subsequent chapters, we will discuss some basic modeling principles of *structural analysis* and the interaction between solid materials and fluid as well. *Fluid mechanics* and *solid mechanics* together comprise *continuum mechanics*. The basis of *continuum mechanics* is *mechanics* that studies the behavior or motions of bodies and their effects on each other. *Mechanics* is a part of fundamental science - *Physics*. There are other fundamental sciences, such as *Mathematics*, *Medicine*, and *Chemistry*. Finally, the root of all sciences is *Philosophy* (from the ancient Greek φιλοσοφία (philosophia) – “love of wisdom”). *Philosophy* studies the general and fundamental problems.

## THEORETICAL PART - I

### Chapter 1. The language

In this chapter we will introduce some of the vocabulary used to describe flow motion. The usage of some of these terms is slightly different from that in common language. There are several basic mathematical and physical terms to be understood. Let us start with simple basic terms that we use very often in our life. To simplify the writing of terms, one or two Latin or Greek letter notations are used. Sometimes, the same letters may be used for different terms. All notations are presented on pages viii to xii.

**Velocity** is a simple term. It is the rate of movement or how fast a body or flow moves. So, the *velocity* unit can be expressed in units, such as meter per second ( $m/s$ ), and kilometer per hour ( $km/h$ ). For convenience, we will use mainly the SI system. SI comes from the French term *Syste'me International d'Unite's* (International System of Units). This system has been created to simplify the units system for international use. The unit of *Velocity* in SI is meter per second ( $m/s$ ).

The Latin letters  $u$ ,  $v$ , and  $w$  are usually used to denote velocity.

**Mass** has many definitions. We will use a simple one that defines *mass* as the amount of matter. Sometimes mass is confused with weight. Unlike mass, weight is the force that a body exerts to maintain its position against gravitational force. So, the *mass* of body on the Earth or on the Moon would be equivalent, but the weight would be different since the gravitational force is not the same on both bodies. Mass unit in SI is a kilogram ( $kg$ ), and it is denoted by the Latin letter  $m$ . **Mass flow** is more apt for fluid motion. It is the mass of gas or liquid that is passing through a given area in unit time. The unit of mass flow in SI is a kilogram per second ( $kg/s$ ). A similar term is **volume flow**. It is the volume of fluid that passes through an area. This parameter is often used to evaluate fans. Instead of the SI unit a cubic meter per second

## Chapter 1. The language

( $m^3/s$ ) the unit cubic feet per second (*CFM*) is often used in the electronic industry.

Another basic term is a **density**. It is defined as a mass per cubic meter ( $kg/m^3$ ). The Greek letter  $\rho$  is used to denote density.

For *time*, we will use a simple common definition: **time** is just *time* and it always progresses in one direction. The unit of time in SI is a second (*S*) and it is usually denoted by Latin letter *t*.

**Temperature** is the measure of the heat of a substance. Different scales are used to measure *temperature*, including Celsius, Kelvin, and Fahrenheit. In SI, the unit of temperature is Kelvin (*K*). However, Celsius (*C*) is commonly used as well. To denote temperature, the capital letter *T* is used. Now, we will look at more abstract terms.

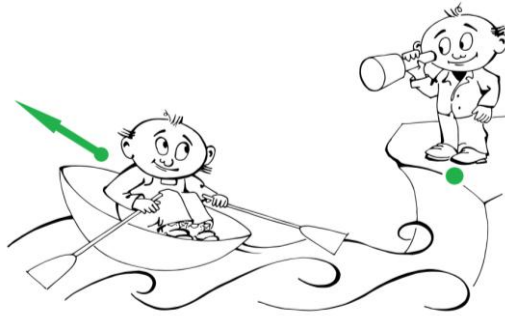
The first term is **scalar**. *Scalar* comes from the Latin word – “scalaris” (stepping). *Scalar* is described by a single number and does not have any other characteristics. Some examples of *scalar* are length (*l*), temperature (*T*), area (*A*), time (*t*), and mass (*m*). Strictly speaking, there are other different definitions of scalar in fields such as Math and Classical Physics. The next term is **Vector**. The *vector* is a more complex term than a scalar, and it is described by one number and a direction. A good example of a *vector* is velocity. There is a value and direction in which a body or fluid moves. Vectors are usually denoted by a letter with an arrow above, for example for velocity  $\vec{u}$ ,  $\vec{v}$ , and  $\vec{w}$ .



Let us discuss the terms **Fluid** and **Flow**, which we have already used in the previous page. *Fluid* is a physical term that refers to something that can change form (deform) or *flow* (fluid motion) by forces or can stay under the same conditions (for example, water in

## Chapter 1. The language

glass). Some examples of *fluid* are the air around us, the water in rivers, and the blood in our bodies. Therefore, fluid can be liquid or gas. *Fluid* or *fluid* motion can be described by scalars and vectors. For instance, the water in the river has a temperature (scalar) and moves with some velocity (vector). If we measure both these two parameters at all points a chosen river area simultaneously, we can get the temperature and velocity fields that describe *fluid* movement and state. In fact, there are two ways to observe flow. The first way is to observe fluid motion by staying on a shore at fixed position. This method is called the *Eulerian* method. The second way is to observe flow by sitting in the boat that moves due to the river flow. This way is the *Lagrangian* method. These methods will be described in details in Chapter 6.

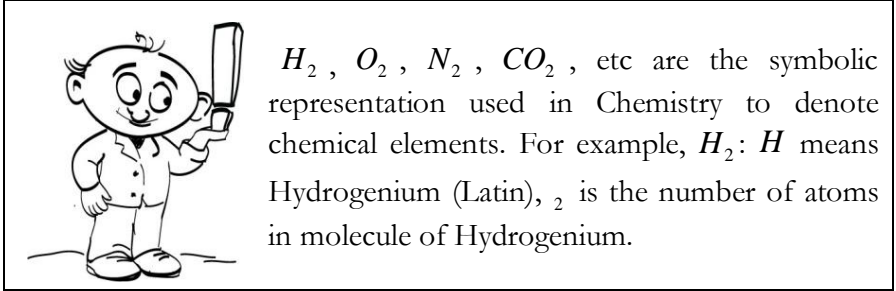


Several well-known words have strange connotation. To break the monotony, let us consider two practical definitions. Although not very important, they will help understand different ways to use *CFD* or *structural analysis*. In general, modeling is used by both scientific researchers and engineers. Scientific researchers consider small details of flow. For this purpose, a complete understanding of theory, equations, etc. is necessary. For engineers, the theoretical background is less important. When modeling is applied in the engineering field, many theoretical aspects can be neglected. Engineers use simplified geometries and accept simulation results with lower precision.

Now let us move on to other terms. We will discuss the different types of flow. The air around us can be named as a *one-phase fluid* since it has only a single gas form. Generally, air can be decomposed into

## Chapter 1. The language

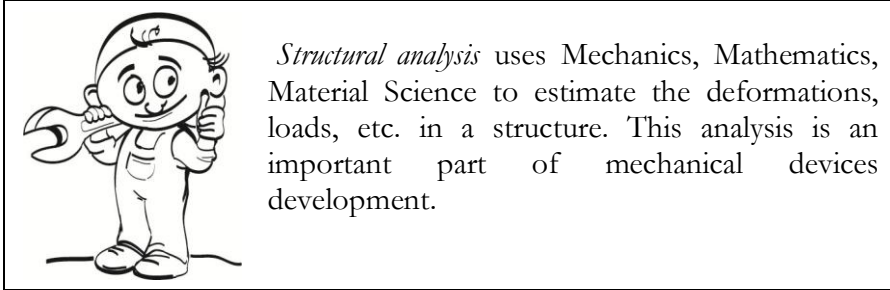
oxygen ( $O_2$ ), nitrogen ( $N_2$ ), carbon dioxide ( $CO_2$ ), etc. Such a composition of same-phase elements is called a *multicomponent fluid*. If there is a reaction between the chemical elements, for example, combustion of methane ( $CH_4$ ) in air ( $O_2$ ,  $N_2$ ,  $CO_2$ ), the fluid motion could be called *flow with chemical reactions*.



Go back to the air around us. In case of rain, we have a *two-phase flow*: air (gas) and water drops (liquid). Let us consider the burning of wood in *CFD* terms. The logic is simple. There is wood (solid) and air (gas). Therefore, there are two phases. Wood burns, and so, it means that certain chemical reactions occur. Thus, we get the term *two-phase flow with chemical reactions* or *combustion*. In addition, in this case, there is temperature change or heat transfer. So, we might call it: *two-phase flow with chemical reactions and heat transfer*. It sounds a bit long. The shorter term, *two-phase combustion*, is used for such cases since the term combustion indicates the occurrence of *heat transfer* and *chemical reactions*. Another example of flow with *heat transfer* is a working computer. There is “cold” surrounding air and “hot” heating chips inside a computer. Sometimes, a simple flow without heat transfer is called “cold” flow. Other possible physics concepts that may be taken into consideration are noise (acoustic), electro-magnetics, radiations, etc. Even if we are talking here about *CFD* in general, the flow often interacts with solid bodies not only by flow around, but, for example, deforms it. In this case, we have to take such an influence into account. This type of modeling is called **Fluid Structure Interaction (FSI)**. In this case, a *structural analysis* has to be provided in addition to *CFD*. In many cases, it is necessary to perform only *structural analysis*. We will examine some

## Chapter 1. The language

basic examples of *structural analysis* in further chapters. Sound knowledge of the definitions of flow type will help navigate in the *CFD* world freely.



Now, let us talk about flow in detail. There are 3 modes of flow: **laminar**, **turbulent** and **transitional**. *Laminar* comes from the Latin word “*lamina*”, which refers to a plate or a stripe. Fluid layers do not mix each other at low-flow velocity. The example of *laminar flow* in a channel is shown in Figure 1.1. To visualize a flow path, smoke with air has been used.

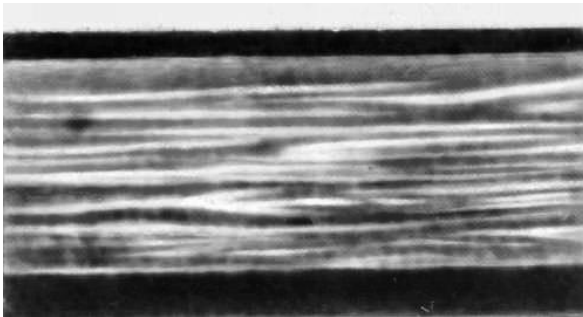


Figure 1.1. Laminar flow in channel.

By increasing the speed, the flow layers start to mix each other. This flow mode is called *transitional*. When the flow is fully converged to chaotic, non-structured behavior, we have *turbulent flow* (Figure 1.2).

## Chapter 1. The language

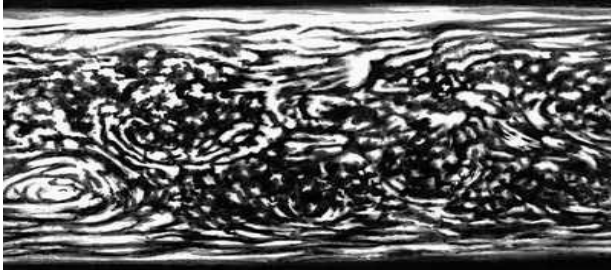


Figure 1.2. Turbulent flow in channel.

Flow can be **compressible** or **non-compressible**. These terms are related to density change. For instance, water flow is *non-compressible* since there is no density change of water. Air could be taken as *non-compressible* at low speed as well, but at high velocity, it should be considered as *compressible* flow. In reality, all flows are **unsteady**; in other words, flow always changes in time. However, in some cases when the flow is rather stable, we can consider flow as a **steady** one.

Finally, we will introduce some physical terms. **Force** is something that can cause a motion, or strictly speaking, cause a change in speed. *Force* is usually denoted by the Latin letter  $\vec{F}$ . It is with an arrow since it is a vector. The unit is Newton ( $N$ ) that is derive from  $\text{kg} \cdot \text{m} / \text{s}^2$ . Another term related to *force* is **Pressure**. *Pressure* could be calculated as a *force* pushed to the surfaces. So, the unit is Newton per square meter ( $N / \text{m}^2$ ). The Latin letter  $p$  is used to denote *pressure*.

One of the important terms related to fluids is **viscosity**. *Viscosity* comes from Latin “*viscum*”, which refers to a glue used to catch birds. In Latin, it used to describe high-viscosity glue. *Viscosity* is similar to friction, but it is an internal property of fluid that resists motion from one part to other part of fluids. Fluids with zero *viscosity* are called *inviscid fluids*. Such flow conditions are possible for some materials (super fluids) at low temperature. The Greek letters  $\mu$  and  $\nu$  are used to denote *viscosity*.

**Momentum** is a measure of the mechanical movement of body. It is a product of velocity and mass of body. Since it has a velocity, it is a



## Chapter 1. The language

vector as well. Let us explain it in other words. There are two balls with identical shape, but one of them is made of steel and the other one is of plastic. If we roll them and get the same speed for both balls, the steel ball will roll further than the plastic ball after releasing as it has more mass. We can say that the steel ball has more *momentum*. In fact, we need to apply more force to the steel ball than to the plastic one to achieve the same speed. Another thing is that if we take two identical steel balls and roll one with higher speed than other; the higher speed ball will roll further. So, in this case, velocity and mass can be interchangeable. If we create the same *momentum* for different balls, they will move the same distance, for example, 2 kg ball with 3  $m/s$  and 3 kg ball with 2  $m/s$ .

## Chapter 2. Grammar

Now we are coming to a more complex area: grammar of *CFD*. To describe a fluid motion, certain equations have to be solved. In this chapter, we will discuss the main equations in words. In Math, the equations are usually abstract (not real). For instance,  $A$  plus unknown number  $X$  is equivalent to  $B$ . It is an equation. So, to find unknown number  $X$ , we need to subtract  $A$  from  $B$ . This action is a solution. There is no physical meaning behind this equation except other than the value of the numbers. If we put some objects such as apples behind the numbers, it would be more realistic. Similarly, if we add some physical meaning, such as velocity and density to numbers, we might get the equations that describe some physical processes, for example, a fluid motion.

Physical processes can be described by laws. Equations are the mathematical formulation of laws. Laws and equations together create the *CFD* grammar. The basis of *CFD* is the *conservation laws*. Let us discuss them. The first law is **mass conservation**. This law is easy to understand. For instance, there is a 1 liter bottle of water, and we fill a 0.2 liter glass from this bottle; thus, we have 0.8 liter of water in the bottle and 0.2 liter in the glass. The total amount of water remains the same, at 1 liter. Now, there is a real case that is more closed to fluid motion. Take a channel system with two supplies and one exit; introduce 2 liters per second into one and 3 liters per second into the other supplier. Thus, we will get 5 liters per second at the exit. In general, the total mass flow at the exit will be the total of the mass flowing at the suppliers (Figure 2.1).

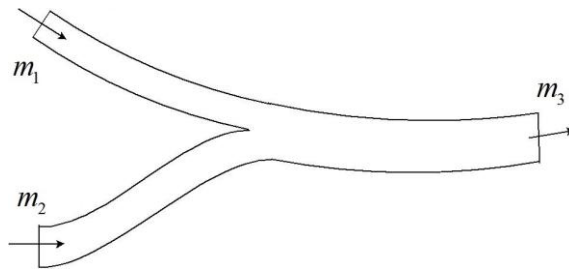


Figure 2.1. Mass flows in the channel.

## Chapter 2. Grammar

The second main law is **conservation of momentum**. According to this law, the total momentum of a closed system does not change. There is an exact method to get this law in equation, but here, we will try to understand it in a simple way. For example, there are two balls with the same mass and velocity moving toward each other that collide (Figure 2.2).

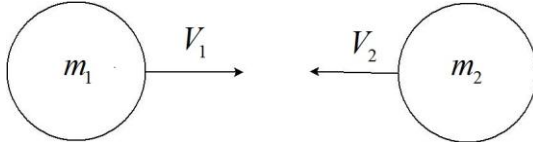


Figure 2.2. Balls with same mass and velocity before collision.

After collision, they will go to in the opposite directions (Figure 2.3).

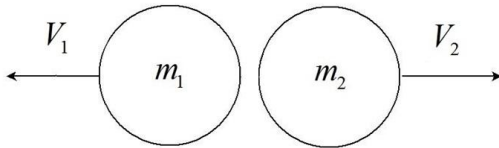


Figure 2.3. Balls with same mass and velocity after collision.

Since there are no other influences, they will have the same velocity with an opposite direction. Therefore, the total momentum of a system will be the same as that before collision. Applying it to the channel used before (Figure 2.4), we can say that the total momentum at two suppliers will be the same as that at the exit. In these examples, we have to assume that there is no friction.

## Chapter 2. Grammar

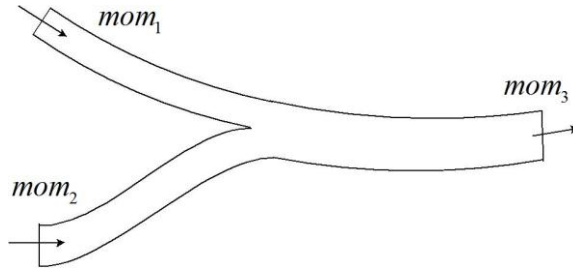
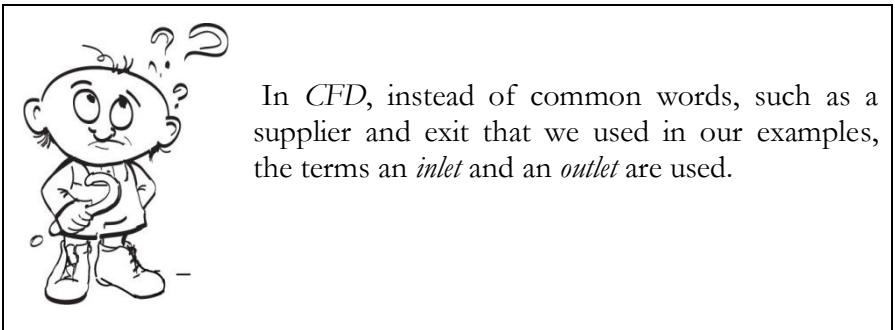


Figure 2.4. Momentums in the channel flow.



The third main law is **conservation of energy**. This law states that the total energy of a closed system is always the same. In thermodynamics, this law is called the first law. A simple explanation is that if we take an isolated pipe and supply it with water at  $20^\circ C$ , we will get the same temperature at the end. Another interesting thing coming from this law is that it rules out the possibility of creating perpetual motion machines of the first kind that generates all time an energy without supplying energy. A long time ago, the ingenious Leonardo da Vinci attempted to create such a machine without any success. In the earliest 1900s, some people even patented such devices; however, none of them worked. One example of such a machine is shown in Figure 2.5.

## Chapter 2. Grammar

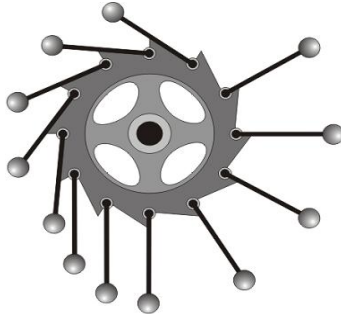
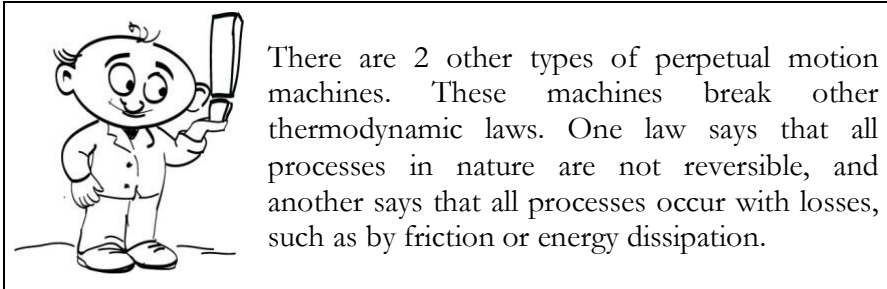


Figure 2.5. A perpetual motion machine.

The geometry of cogs is created to have the balls on the left side more closed to the axis than on the right side. Therefore, we might think that it will rotate since the leverage seems lower on left than on the right side, but in reality, the wheel will be without any motion as the forces on the left and right sides are balanced by the different number of balls.



The above *conservation laws* are valid for isolated (closed) systems where there is no interaction with the surrounding environment, for example, heat loss through channel walls, friction between balls and ground, etc. The isolated systems are not possible in real life. However, in many cases the interaction with environment can be neglected and the system can be considered as isolated one. The three fundamental *conservative laws* are valid not only for *CFD*, but for all other sciences. Now, we are ready to move to a practical part.

## Chapter 8. Turbulence

A majority of flows in technical applications are rather turbulent than laminar. Peter Bradshaw in his introduction to *Turbulence* wrote: “the one uncontroversial fact about turbulence is that it is the most complicated kind of fluid motion”.

Since 1883, following Osborne Reynolds, it has been possible to characterize the state of flow using *Reynolds number*:

$$\text{Re} = \frac{l_0 \cdot u_0}{\nu}, \quad (8.1)$$

where  $l_0$  is a typical length,  $u_0$  a typical bulk velocity and  $\nu$  a kinematic viscosity. The *Reynolds number* expresses the ratio between inertial and viscous (or molecular) forces. If this ratio is small, the viscous (or molecular) forces are comparable to the inertial forces and the flow keeps its regular structure (laminar flow). If the ratio becomes larger, the viscous forces do not suffice to compensate the inertial forces. The flow turns unstable, and small initial perturbations destroy the regular flow structure, causing turbulence (turbulent flow). The turbulent flows can be imagined as collections of eddies. Turbulence increases the rate at which conserved quantities are stirred, i. e., parcels of fluid with different contents of conserved quantity (momentum, energy, concentration, etc.) are brought into contact. This is often called turbulent mixing or turbulent diffusion. The molecular viscosity reduces the velocity gradients, causing a destruction of turbulent eddies and a dissipation of kinetic energy into the internal energy of fluid. The bigger eddies dissipate into smaller ones, transferring the kinetic energy of turbulent fluctuations. This process, first revealed by Kolmogorov [54], is called *energy cascade*. Small eddies can form into a bigger one as well. This reverse process is called *back scattering*. Richardson (1922) wrote a small poem describing the cascade character of turbulence:

Big whirls make little whirls,  
That feed on their velocity;  
Little whirls have smaller ones,  
And so on into viscosity.

Recent investigations have shown the turbulent flows of coherent structures as repeatable and essentially of deterministic character. The random part that is dominant in turbulent flows causes these events to differ in size, strength, and time interval between occurrences. There are, however, some flows that feature coherent structures with clear periodicity, and certain frequency can be referred to such a periodical motion.

To summarize, it appears clearly that turbulent flows are:

- random in time and space
- unsteady
- three-dimensional
- dissipative
- vortical.

It is possible to describe the turbulent motion by *laws of probability* following a *probability density function (PDF)* approach [162], *chaos theory*, etc. In general, some traditional ways used in CFD modeling include *Reynolds Averaged Navier-Stokes (RANS)*, *Large Eddy Simulation (LES)* and *Direct Numerical Simulation (DNS)*. The differences among the three methods to study turbulent flow in time are illustrated in Figure 8.1.

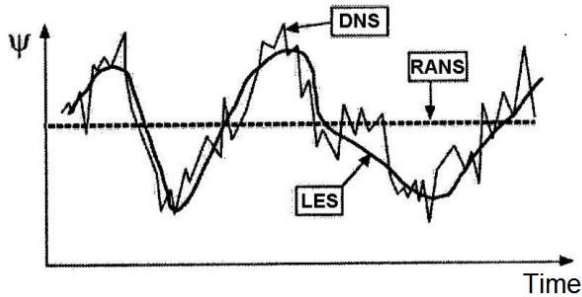


Figure 8.1. Methods to describe turbulent flow in time.

Here  $\psi$  is a variable in some fixed point, for example, velocity. In case of a turbulent flow, the velocity fluctuates in time, and *DNS* can precisely catch such behaviors. *LES* provides information about the unsteady effects of turbulent flow generated by large scales. *RANS*

## *Chapter 8. Turbulence*

operates with averaged data. To model the transient effects in *RANS*, the *unsteady RANS* (*URANS*) can be used [170]. The *URANS* method will be described in Chapter 18.5. The alternative methods to describe turbulent flows will be considered in part IV.



## Chapter 14. Mesh generation: ANSYS Mesh

ANSYS Mesh (AM) was introduced in the first practical part. The upper part of AM is shown in Figure 14.1.

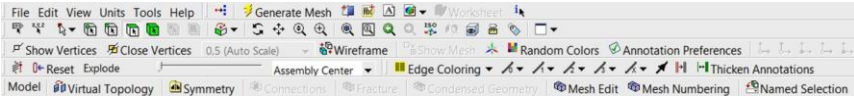








Figure 14.1. ANSYS Mesh interface.

On the top of AM there is a *Menu* bar with general functions: *File* - operations with files (*Save*, *Export*, *Clear Generated Data* and *Close*), *Edit* - operation with mesh actions (*Copy*, *Cut*, *Delete*, etc.), *View* - settings with user view, *Units* - operations with units (*Meter*, *Millimeter*, *Foot*, *Inch*, *Degree*, *Radian*, etc.), *Tools* - operations with additional options like macros and third part addins (*Addins*, *Options*, etc.) and *Help* - integrated help. On the right side of the *Menu* bar, there is a toolbar with several icons as *Generate Mesh* , *Create a New Section*  to analyze mesh at the cross section, *New Figure or Image*  to create an image, *Selection Information*  to see the information about selected element, etc. There are three lines of toolbars below: the selections, manipulation, view toolbars, etc with similar functions as DM. At the last line, there is the specific mesh toolbar for frequently used operations with mesh, such as *Virtual Topology*  to combine surfaces with similar topologies for smooth meshing, *Symmetry*  to work with symmetrical conditions, etc. Depending on section in the *Outline* tree (Figure 14.2), the icons in toolbars are changed according to the section function.

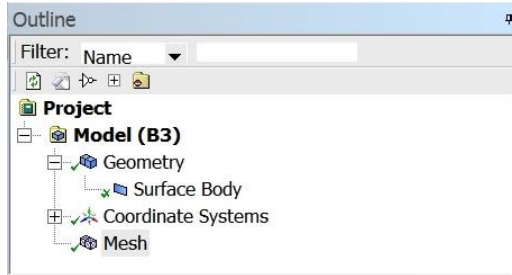


Figure 14.2. Outline in AM.

The *Outline* tree window on the left side keeps the parameters about *Geometry*, *Coordinate systems*, *Mesh* and *Named Selection*. The functionality is similar to DM. It is possible to suppress or hide some geometrical elements in *Geometry* section. There are the manipulations with coordinate systems in the *Coordinates Systems*. The section *Mesh* includes all necessary operations to generate mesh. The name for boundary conditions can be set up in the *Named Selection*. In case of selecting an operation in the *Outline* tree, the settings can be modified in the *Details View* window (Figure 14.3).

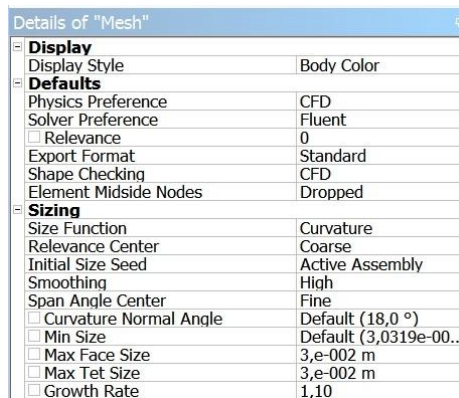


Figure 14.3. Details view in AM.

As usual there is the working area in the center. Under the working area, there is the *Messages* area with important messages about errors or

## Chapter 14. Mesh generation: ANSYS Mesh

recommendations during meshing process. Finally, on the bottom there is the *Status* bar that shows a current state and some geometrical information. Let us generate a mesh for 2D/3D Ahmed body.

### Prerequisites

Software: Workbench, Ansys Mesh (ANSYS 17.0)

Input: 2D/3D CAD models made in Chapter 13

Experience level: beginner

### Goal

Introduction to ANSYS Mesh (AM)

### Description

Open the project *ahmed\_body.wbpj* that we created before. We will generate a mesh for this geometry. Take the *Mesh* in the *Component System* and drop it to the *Geometry* cell of Ahmed body 2D in the *Project Schematic* of *WB* (Figure 14.4).

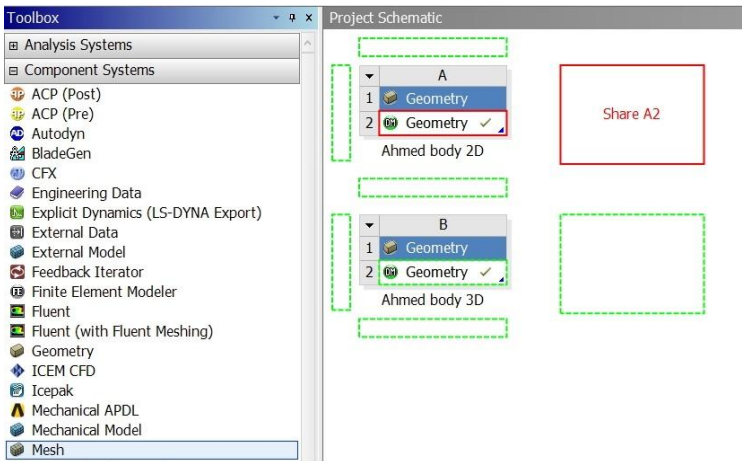


Figure 14.4. Attaching Mesh in Workbench.

## Chapter 14. Mesh generation: ANSYS Mesh

Double-click on the *Mesh* cell to start AM (Figure 14.5).

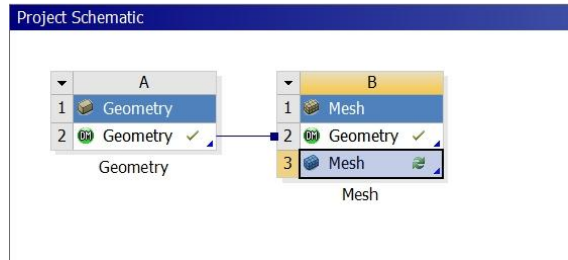


Figure 14.5. Starting ANSYS Mesh in WB.

First, we will generate the mesh with default parameters. Click on the *Mesh* in the *Outline* to see the mesh parameters (Figure 14.3). Click on the *Generate* button to see the default mesh (Figure 14.6).

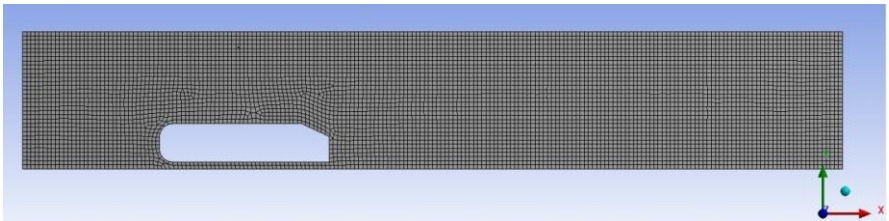


Figure 14.6. Default mesh for 2D Ahmed body case.

Scale the mesh to see the mesh around Ahmed body. The mesh is not fine enough for aero-dynamic simulations (Figure 14.7).

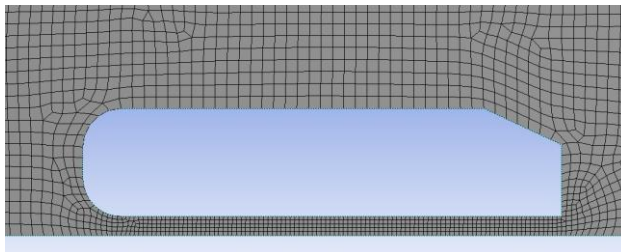


Figure 14.7. Mesh around Ahmed body.

In the *Sizing* section of the *Details* window, change the *Size function* from *Curvature* to *Curvature and Proximity* and enter 5 in the *Num Cell across Gap*. It means that any gap will be meshed at least with 5 elements across it. Now, we will turn up the mesh by using local settings. Select all edges, except for the bottom line of Ahmed body, by left mouse click with pressed Ctrl button, call pop-up menu (RMB), and select *Insert* → *Sizing* option. Enter *Element Size 0.01 m* and *Growth Rate 1.1* in the *Details of "Edge sizing"* window (Figure 14.8). Retain other parameters as default.

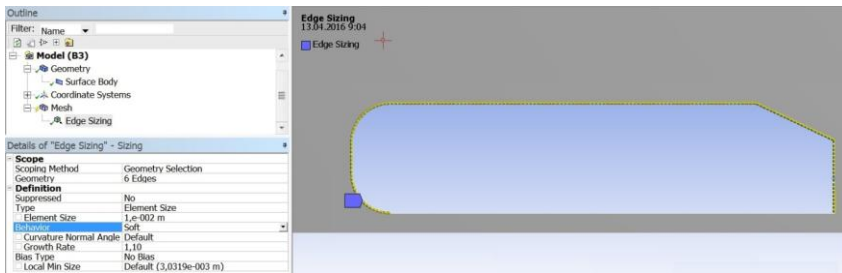


Figure 14.8. Design Modeler interface.

After completing *Setup*, click on the *Generate* button to re-generate mesh. The new mesh is shown in Figure 14.9. This mesh is not perfect, but finer than the first one. The information about the mesh can be seen in the *Statistics*. The mesh has 8201 nodes and 7801 elements. The number of nodes and elements can be slightly different. The solvers operate with nodes or elements, for example, Fluent operates with elements and CFX with nodes. In a 3D model and by using tetra/triangles based mesh, the elements number is usually higher than nodes. The number of elements/nodes is important for preliminary estimation of simulation time and computer requirements.

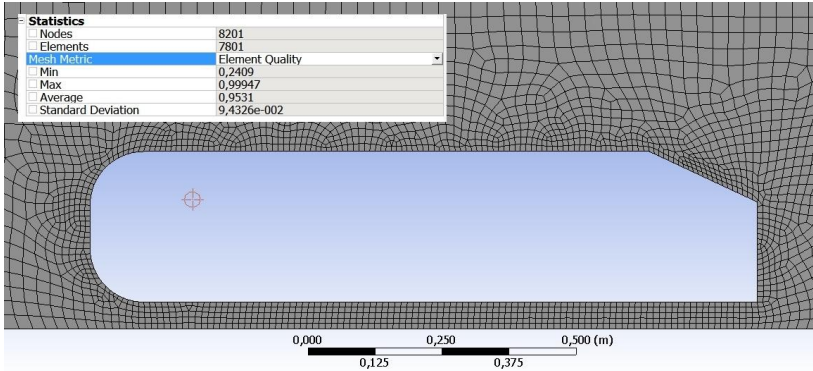


Figure 14.9. Refined mesh.

Select *Mesh Metric* → *Element Quality* in the *Metric statistic*. The minimal *Element Quality* is relatively low *0.2409*. It is possible to check other important quality parameters such as *Aspect ratio* (*Max. = 1.93*), *Skewness* (*Max. = 0.72263*) and *Orthogonal Quality* (*Min. = 0.64972*). The *Element Quality* option will call the additional *Mesh Metrics* panel below the *Main* window (Figure 14.10).

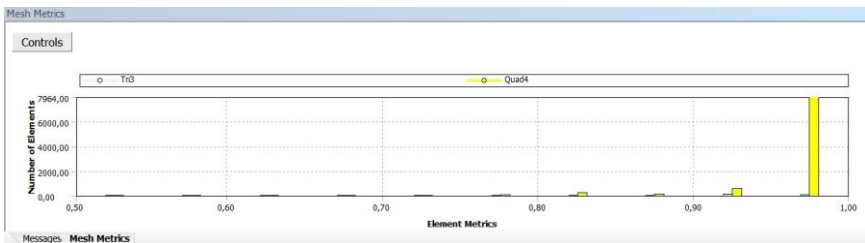


Figure 14.10. Mesh metrics panel.

The elements with low quality are not well visible in the *Mesh Metrics*. Click *Control* and change the range for *X-Axis*: *Min 0.2 - Max 0.5* and for *Y-Axis*: *Min 0 - Max 10* (Figure 14.11).

Chapter 14. Mesh generation: ANSYS Mesh

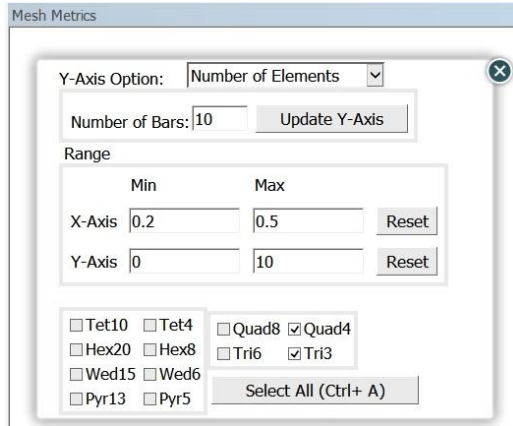



Figure 14.11. Control panel.

Close the *Control panel* . Now, the elements with low quality are visible in the *Mesh Metrics* window. Select all bars (Ctrl + LMB) to see these elements (Figure 14.12).

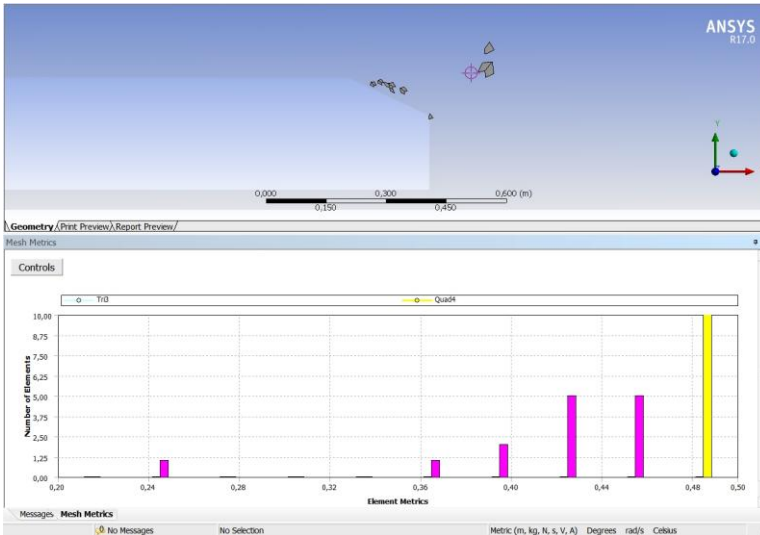


Figure 14.12. The worst elements.

## Chapter 14. Mesh generation: ANSYS Mesh

Let us improve it by using other mesh methods. Click on *Mesh* → *Insert* → *Method* (Figure 14.13).

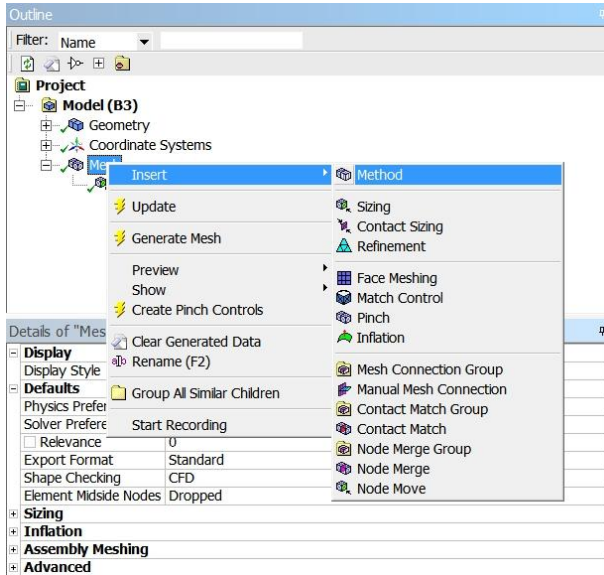


Figure 14.13. Changing the mesh method.

Select the *Body* in the main window for *Geometry* and method *MultiZone Quad/Tri Method* in the *Details* window (Figure 14.14).

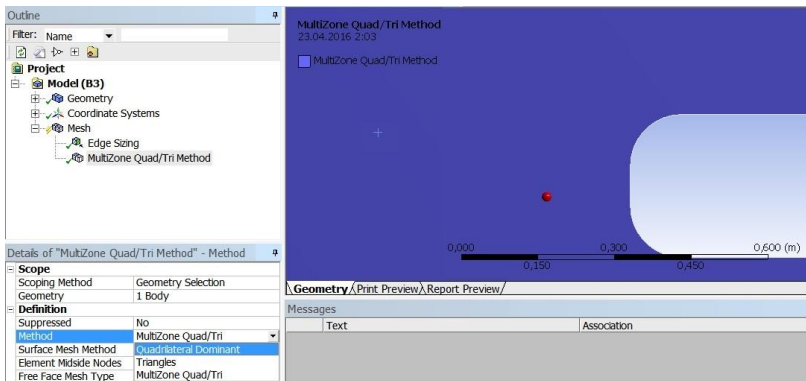


Figure 14.14. Setup MultiZone Quad/Tri Method.



## Chapter 14. Mesh generation: ANSYS Mesh

Click on the *Generate* button and check again *Quality* (Figure 14.15).

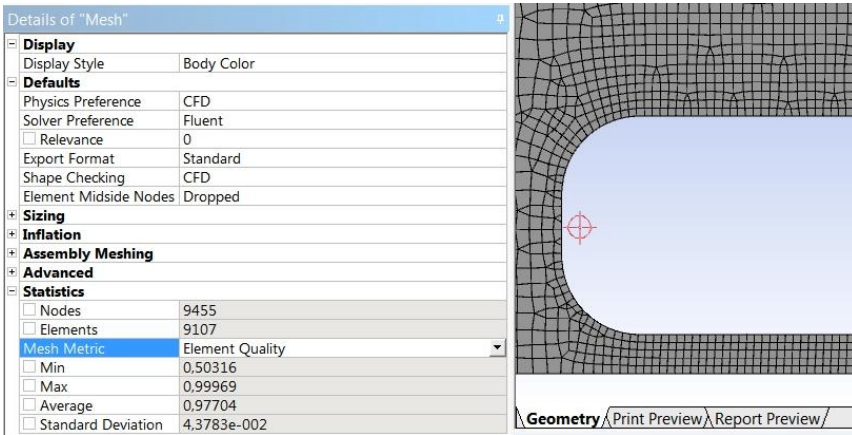


Figure 14.15. Statistics panel.

The mesh quality was improved. The minimal *Element Quality* is 0.50316. Other important quality parameters, such as *Aspect ratio* (*Max.* = 1.63), *Skewness* (*Max.* = 0.51446) and *Orthogonal Quality* (*Min.* = 0.84179) can be checked as well.

The last operation is the naming of the future boundary conditions. Select the front edge of domain, use RMB to call the context menu and select *Create Named Selection* (Figure 14.16).

## Chapter 14. Mesh generation: ANSYS Mesh

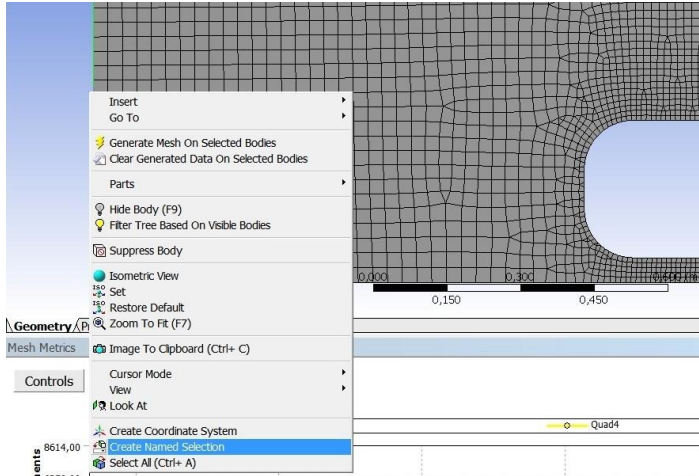


Figure 14.16. Create Named Selection option.

Enter the name for boundary condition: *Inlet* in the new pop-up window (Figure 14.17).

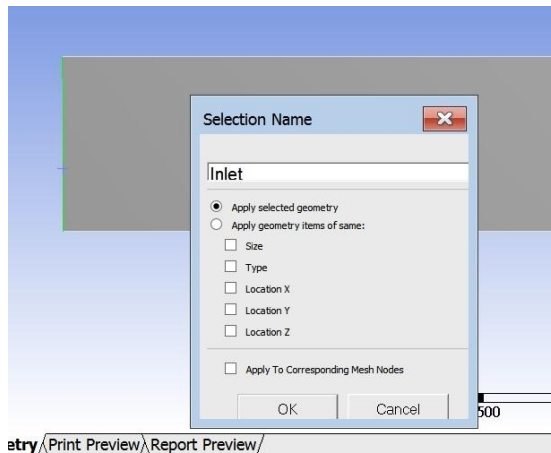


Figure 14.17. Naming for selected edge.

Make the similar operations for other boundary conditions (Figure 14.18). Select '*Outlet*' for the right line, '*Symmetry*' for the upper line,

'Ground' for the bottom line, 'Wall' for lines limiting Ahmed body and 'Fluid' for surface.

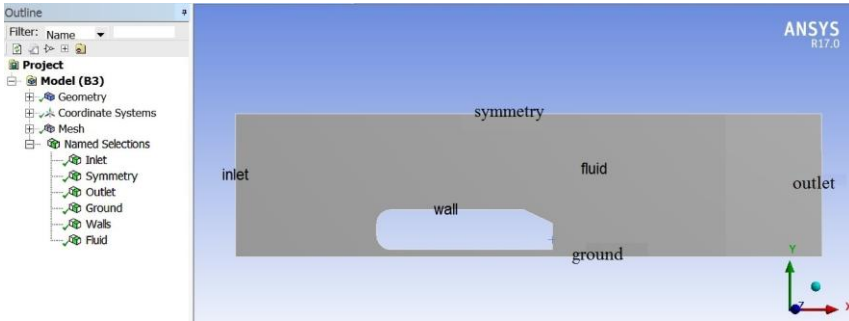


Figure 14.18. Setup of names for 2D Ahmed body (cut in X direction).

Finally, save the project, close AM and return to WB.

Now, we will generate a mesh for 3D Ahmed body. Similar to 2D example, attach the *Mesh* to *Geometry* cell in *WB* and open it. In the *Menu* bar, change the *View* to *Wireframe* to see the inside geometry (Figure 14.19).

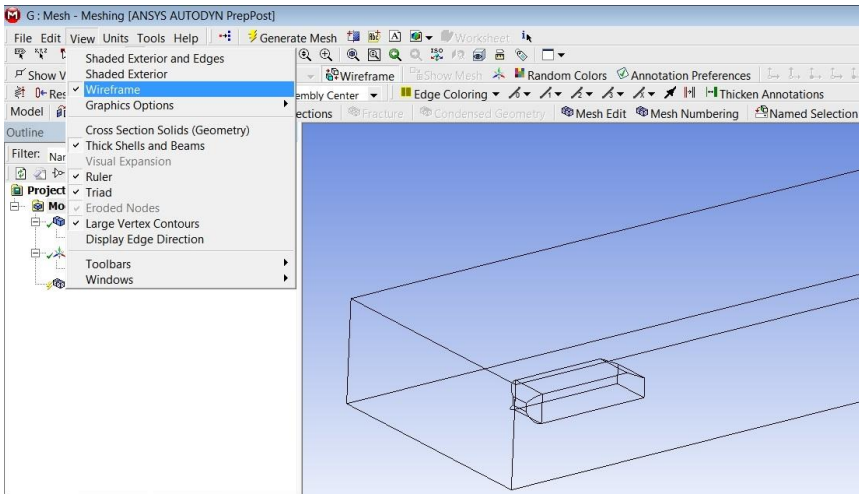


Figure 14.19. Wireframe view for 3D Ahmed body.

## Chapter 14. Mesh generation: ANSYS Mesh

Enter the names (Figure 14.20) for boundary conditions by using *Name selection*, similarly to the previous example. Select “*Open*” for a surface, “*Symmetry*” for the left, right and top surfaces, “*Ground*” for the bottom surface, “*Wall*” for surfaces limiting Ahmed body and “*Fluid*” for the body. Use *Hide Faces (F8)* option to hide outer surfaces.

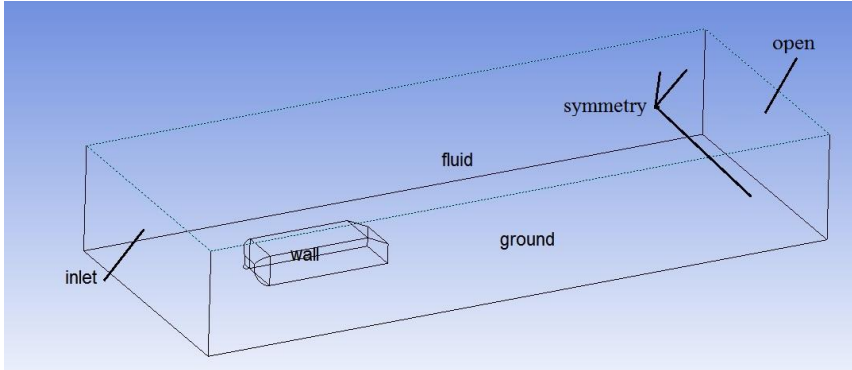


Figure 14.20. The names for boundary conditions.

We will use the prism layers near the wall for more precise aerodynamic simulation. Open the *Inflation* option in the *Details* window (Figure 14.21).

## Chapter 14. Mesh generation: ANSYS Mesh

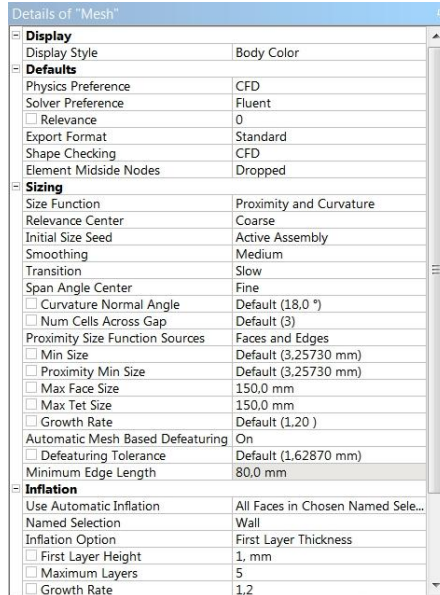



Figure 14.21. The global mesh parameters.

Select *Use Automatic Inflation* with *All Faces in Chosen Named Selection* in the inflation section and specify the *Wall* in the *Named Selection*. Select *First Layer Thickness* in the *Inflation Option*, enter *1mm* in the *First Layer Height*, and add *5* in the *Maximum Layers* and *1.2* in the *Growth Rate*. AM will generate *5* prism layers with the first *1 mm* layer and the next layers will be enlarged to *1.2* times. The inflation operation in AM gives an option to generate the prism layers around chosen surfaces. After it, click on the *Generate* button. Change the view to *Shaded exterior and Edges* again. Now, we will see the mesh inside by creating *New section plane* . This option can be called from the *Top* toolbar. Select *YZ plane* and create the line that cuts the domain in vertical direction (Figure 14.22).

## Chapter 14. Mesh generation: ANSYS Mesh

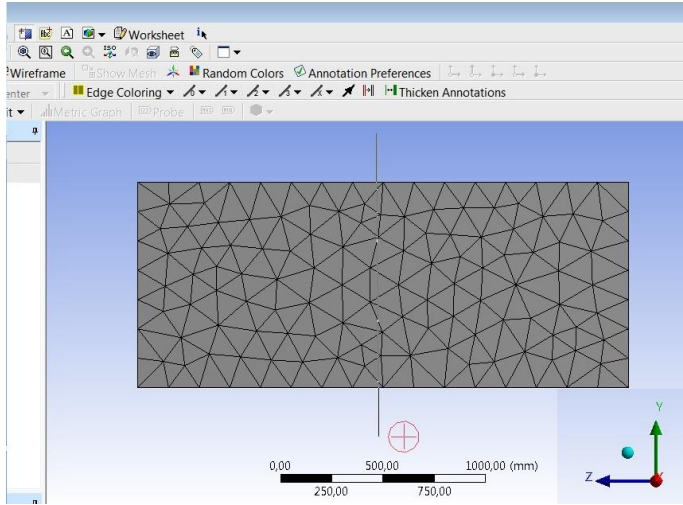


Figure 14.22. The use of section plane.

The section plane view (enlarged) is shown in Figure 14.23. The prism layers can be seen around 3D Ahmed body.

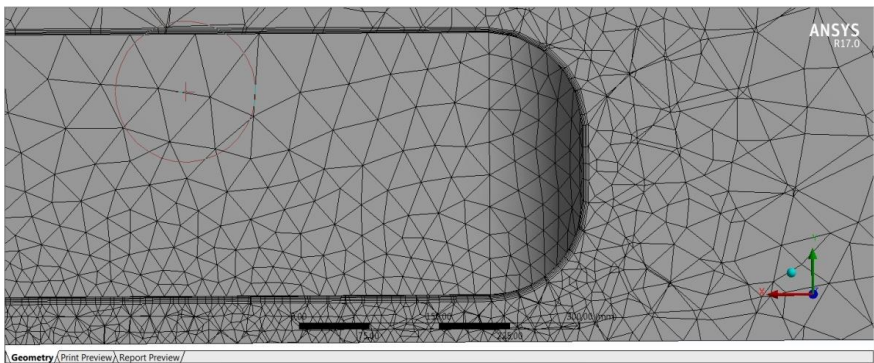


Figure 14.23. The section plain view.

Similar to 2D, it is possible to check the mesh quality and improve it by using other methods. We will retain the current mesh for future simulation. Close AM and save the project in WB.

## Chapter 22. Acoustics

A simulation of noise generation plays a significant role during development of devices from a home appliance to the gas turbine engines. The acoustic processes can not only generate a noise, but can also influence the combustion processes in engines, cause the vibrations, etc. The study of acoustic propagation and appearance is called acoustics. *Acoustics* comes from Greek word  $\acute{\alpha}\kappa\omicron\upsilon\sigma\tau\iota\kappa\acute{\omicron}\varsigma$  (*akoustikos*) that means “ready to hear”.

Lighthill, a pioneer in aeroacoustics, derived a *wave equation* from the *continuity* and *Navier-Stokes equations* [64]. It is written as

$$\frac{\partial^2 \rho'}{\partial t^2} - a_0^2 \frac{\partial^2 \rho'}{\partial x_i \partial x_i} = \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j}, \quad (22.1)$$

where  $a_0$  is the speed of sound and  $T_{ij}$  is the Lighthill tensor defined as

$$T_{ij} = \rho u_i u_j + (p' - a_0^2 \rho') \delta_{ij} - \sigma_{ij}. \quad (22.2)$$

In general,  $T_{ij}$  represents the acoustics sources: the first source is Reynolds stresses; the second term is nonlinear acoustics generation; and the third term is the sound generated by shears. The noise sources can be divided into several types such as monopole, dipole, and quadrupole. A monopole is a noise source that induces the sound waves uniformly in all directions. A dipole source has two monopoles of the same power that induce the sound waves with opposite phases. The distance between monopoles is comparable with the wavelength of the sound. Therefore, adjacent to the two monopoles, there is a zone where the fluid moves back and forth and generates a noise. A quadrupole has 4 monopoles with opposite phases. The different types of noise source are shown in Figure 22.1.

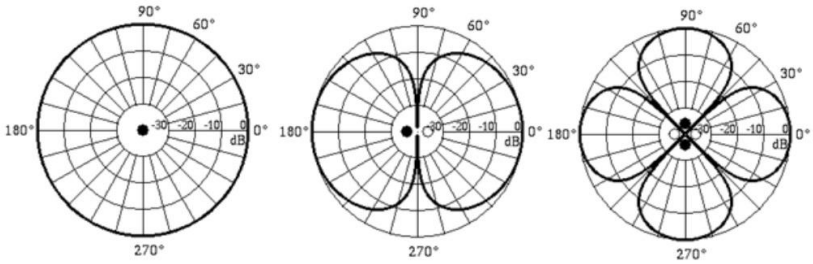


Figure 22.1. Monopole, dipole, and quadrupole noise sources.

The noise is estimated by *Sound Pressure Level (SPL)* that is usually measured in *decibel (dB)* with logarithmic scale as

$$SPL = 10 \log \left( \frac{P}{P_{ref}} \right), \quad (22.3)$$

where  $P$  is a power of noise and  $P_{ref}$  is a reference power usually taking equivalent to  $10^{-12} \text{ W/m}^3$ .

Since the acoustics energy is small in general, it is negligible for most flows. Therefore, usually, acoustic simulation is considered as post-processing of flow simulation. Firstly, the simulation of flow is provided to get a velocity, pressure, and other characteristics of flow. From this data, the noise level can be extracted. In some cases, the two directional simulations can be provided, for instance, to study the acoustics influence to combustion.

There are many methods to calculate the noise level, ranging from empirical to direct modeling. The most complex method is *direct modeling*. In this model, the appearance and propagation of acoustic waves are resolved directly. The method is called *Computational Aeroacoustics (CAA)* and uses *main unsteady equations (DNS)* describing flow. This method is expensive and utilizes a high computational power, since it is necessary to satisfy *Courant number*  $C \leq 1 \dots 2$ . The *integral method*, suggested by Lighthill [64], is simpler. In this method, the



## Chapter 22. Acoustics

flow is first modeled by any *transient simulation* (DNS, LES, and URANS), and then, the data are used for the integral solution of wave equation. To obtain a model wave equation, a control surface covering the solid can be used. For the control surface, the function  $f$  is introduced as

$$f \begin{cases} > 0, & \text{out surface} \\ = 0, & \text{on surface} \\ < 0, & \text{in surface,} \end{cases} \quad (22.4)$$

By using this function, Navier-Stokes and continuity equations, the *general heterogeneous equation* of Ffowcs-Williams, & Hawking [34] can be written as

$$\begin{aligned} \frac{1}{a_0^2} \frac{\partial^2 p'}{\partial t^2} - \nabla^2 p' &= \frac{\partial^2}{\partial x_i \partial x_j} \{T_{ij} H(f)\} \\ &- \frac{\partial}{\partial x_i} \{[P_{ij} n_j + \rho u_i (u_n - v_n)]\} \\ &+ \frac{\partial}{\partial t} \{[\rho_0 v_n + \rho (u_n - v_n) \delta(f)]\}, \end{aligned} \quad (22.5)$$

where  $u_i$  is the velocity component of flow in  $x_i$  direction;  $u_n$  is the velocity component normal to surface;  $v_i$  is the velocity component of surface in  $x_i$  direction;  $v_n$  is the velocity component of surface normal to speed;  $\delta(f)$  is Dirac delta function;  $a_0$  is the sound speed;  $p'$  is the sound pressure; and  $H(f)$  is the function of Heaviside defined as

$$H(f) = \begin{cases} 1, & f > 0 \\ 0, & f < 0 \end{cases} \quad (22.6)$$

where  $T_{ij}$  are the components of Lighthill tensor:

Chapter 22. Acoustics

$$T_{ij} = \rho u_i u_j + P_{ij} - a_0^2 (\rho - \rho_0) \delta_{ij}, \quad (22.7)$$

and  $P_{ij}$  is the stress tensor defined as

$$P_{ij} = p \delta_{ij} - \mu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right]. \quad (22.8)$$

The solution of eq. (22.5) is introduced below. The detailed solution process is described in [34].

$$p'(\vec{x}, t) = \underbrace{p'_T(\vec{x}, t)}_{\text{monopole}} + \underbrace{p'_L(\vec{x}, t)}_{\text{dipole}} + \underbrace{p'_Q(\vec{x}, t)}_{\text{quadropole}}, \quad (22.9)$$

where the first source is described by

$$4\pi p'_T(\vec{x}, t) = \int_{f=0} \left[ \frac{\rho_0 (\dot{U}_n + U_n)}{r(1-M_r)^2} \right] dS + \int_{f=0} \left[ \frac{\rho_0 U_n \{ r \dot{M}_r + a_0 (M_r - M^2) \}}{r^2 (1-M_r)^3} \right] dS, \quad (22.10)$$

the second source can be found by

$$4\pi p'_L(\vec{x}, t) = \frac{1}{a_0} \int_{f=0} \left[ \frac{\dot{L}_r}{r(1-M_r)^2} \right] dS + \int_{f=0} \left[ \frac{L_r - L_M}{r^2 (1-M_r)^2} \right] dS + \frac{1}{a_0} \int_{f=0} \left[ \frac{L_r (r \dot{M}_r + c (M_r - M^2))}{r^2 (1-M_r)^3} \right] dS, \quad (22.11)$$

and the third source by

$$4\pi p'_Q(\vec{x}, t) = \frac{1}{a_0^2} \frac{\partial^2}{\partial t^2} \int_{f>0} \left[ \frac{Q_{rr}}{r|1-M_r|} \right] dV +$$

$$\frac{1}{a_0} \frac{\partial}{\partial t} \int_{f>0} \left[ \frac{3Q_{rr} - Q_{ii}}{r^2|1-M_r|} \right] dV + \int_{f>0} \left[ \frac{3Q_{rr} - Q_{ii}}{r^3|1-M_r|} \right] dV.$$

(22.12)

If the control surface aligns with the solid surface, the first term on the right side of eq. 22.9,  $p'_T(\vec{x}, t)$  represents the monopole surface source that is defined by a solid shape and it is kinematic motion. A dipole surface source  $p'_L(\vec{x}, t)$  is formed by the force acting to fluid from body. A quadruple volume source  $p'_Q(\vec{x}, t)$  is defined by the volume sources outside of the solid body. At low speed, the last term is negligible.

In many flows, the noise does not have well-defined tones, and the acoustic energy is distributed continuously in the bright range of frequencies. In such cases, it is possible to use the turbulent statistics from the steady equations describing the flow. For example, it is sufficient to have a velocity field and the turbulent characteristics, as kinetic energy and dissipation. Such models are called the broadband models and present the semi-empirical equations that have been extracted from experiments or direct modeling. A popular *model of Proudman* [87] is presented below using a semi-empirical equation:

$$P_A = a\rho_0 \left( \frac{u^3}{l} \right) \frac{u^5}{a_0^5}. \quad (22.13)$$

For the  $k - \varepsilon$  model, it can be re-written as

$$P_A = a_\varepsilon \rho_0 \varepsilon \left( \frac{\sqrt{2k}}{a_0} \right)^5, \quad (22.14)$$

## Chapter 22. Acoustics

where  $a_\varepsilon = 0.1$  is the model constant.

The main advantages, disadvantages, and limitations of acoustics models are summarized in Table 22.1.

Model	Properties
<i>CAA</i>	Excessive computational demands, reflection and scattering, transient, high accuracy
<i>Integral</i>	High computational demands, no reflection and scattering, transient, high accuracy
<i>Based on turbulent statistics</i>	Low computational demands, no reflection and scattering, steady, limited accuracy

Table 22.1. Advantages, disadvantages and limitations of acoustics models