



University of Thessaly

Polytechnic School

DEPARTMENT OF MECHANICAL ENGINEERING

ANSYS SOLAR UPDRAFT TOWER SIMULATION

ΠΡΟΣΟΜΟΙΩΣΗ ΡΟΗΣ ΑΕΡΑ ΣΕ ΗΛΙΑΚΟ ΠΥΡΓΟ ΦΥΣΙΚΗΣ ΣΥΝΑΓΩΓΗΣ
ΜΕ ΤΗ ΧΡΗΣΗ ΤΟΥ ΠΡΟΓΡΑΜΜΑΤΟΣ ANSYS

By: Nikolaos Psarologos

Submitted in partial fulfillment of the requirements for the degree of Diploma
in Mechanical Engineering at the University of Thessaly

© 2025 Nikolaos Psarologos

All rights reserved. The approval of the presented Thesis by the Department of Mechanical Engineering, School of Engineering, University of Thessaly, does not imply acceptance of the views of the author (Law 5343/32 art. 202).

Approved by the Committee on Final Examination:

Advisor Dr. Konstantinos Ritos
Professor, Department of Mechanical Engineering, University of Thessaly

Member Dr. Vasilis Bontozoglou,
Professor, Department of Mechanical Engineering, University of Thessaly

Member Dr. Tassos Stamatelos,
Professor, Department of Mechanical Engineering, University of Thessaly

Date Approved: [September 09, 2024]

ANSYS SOLAR UPDRAFT TOWER SIMULATION

NIKOLAOS PSAROLOGOS

Department of Mechanical Engineering, University of Thessaly

Supervisor: Dr Konstantinos Ritos

Abstract:

Solar Updraft Tower Plants (SUT's) are promising technology for clean energy generation. Using the sun as a source of energy, SUT systems are offering potential for tempering climate change and reducing emissions from burning fossil fuels. There are several numerical, theoretical and experimental works which assess the performance of a SUT, therefore this thesis's purpose is to understand and follow the steps to solve a three-dimensional axisymmetric computational fluid dynamics (CFD) simulation case for a SUT plant with the same dimensions as the Manzanares Prototype plant, the first made full scale experimental structure. Also, different solar discretization parameters, turbulence models and geometries will be tested to observe if the Power output value of the plant can increase. For model validation, the numerical results are compared with other numerical studies and with the typical experimental study of the Manzanares Plant and obtain a good agreement with those. To simulate radiation, DO irradiation solar load model was used and for the turbulence models, the RNG model was chosen among the others through testing. The mesh has acceptable quality metrics and is reliable. The results for the reference geometry and at 1000W irradiation were 14.3 m/s maximum velocity which is close with the experimental 15 m/s, and 43kW of Power output. The best geometry modifications were a 2-degrees inclined collector, a 1-degree inclined chimney wall and finally an alternative curved connection between the collector and the chimney. The first achieved a power output close to 50 kW, while the chimney wall result was near 100kW of Power. All the above merged, creating Case 9 where the power output can reach its maximum value at 360 kW. Finally, the numerical findings of the bigger scaled geometry were very promising, near 270 kW of power, while the optimized bigger scaled geometry reached 461 kW of power with a chimney height at 214 meters. Apart from this particular case simulation, the goal of this thesis is the understanding of the steps to create a CFD simulation. The steps are the design of accurate geometry, the creation of a reliable mesh, the setting of the materials, environment parameters and boundary conditions, and finally the selection of the solar and turbulent models. When these steps are completed, the results of the case must agree with reference studies. When the case is validated, the changes in the geometry, models and parameters can be done to observe the different results.

Table of Contents

Abstract:.....	4
Table of Contents.....	5
List of tables:	7
List of figures:.....	7
Chapter 1. Introduction:	9
1.1 Renewable energy importance:	9
1.2 PV and CSP technologies:	9
1.3 Updraft tower principle:.....	11
1.4 Comparison between the SUT and other technologies:	13
1.5 Manzanares Solar Updraft Tower Plant:.....	15
1.6 Other small-scale plants and proposals:.....	15
1.7 CFD case in this thesis:	17
Chapter 2. Tower design:	18
2.1 Location / solar rays' direction:	18
2.2 Geometry and surfaces:.....	19
2.3 Materials:	20
2.4 Turbine type:.....	21
Chapter 3. Modeling:	23
3.1 Equations:	23
3.2 Performance parameters:.....	25
3.3 Boundary conditions:	26
3.4 Solver and radiation model:	28
Chapter 4. Domain selection and meshing:	31
4.1 Domain size selection:	31
4.2 Mesh:	32
4.2.1 Mesh quality elements explanation:.....	32
4.2.2 Mesh independence:.....	35
4.2.3 Mesh quality results in this case:	39
Chapter 5. Turbulence modeling and validation:	44
5.1 k- ω and k- ϵ models comparison:.....	44
5.2 k- ω models:	45
5.2.1 standard k- ω :	45

Table of Contents

5.2.2 GEKO $k-\omega$:	45
5.2.3 BSL $K-\omega$:	47
5.2.4 SST model:	47
5.3 $k-\epsilon$ models:	49
5.3.1 Standard $k-\epsilon$:	49
5.3.2 RNG $k-\epsilon$ model:	49
5.3.3 Realizable $k-\epsilon$ model:	50
5.4 Other turbulence models:	53
5.4.1 Spalart-Allmaras Model:	53
5.4.2 Transition SST Model:	53
5.5 Validation:	54
Chapter 6. Results and discussion:	56
6.1 Reference geometry results:	56
6.2 Geometrical variations effect:	57
6.2.1 Focus on the collector and chimney connection:	57
6.2.2 Collector slope:	62
6.2.3 Chimney walls angle:	63
6.2.4 Optimized Geometry:	65
6.2.5 Bigger scale plant:	67
6.2.6 Bigger scale plant with optimized geometry:	69
Chapter 7. Conclusions:	70
References	71

List of tables:

Table 1: Experimental research and their values:	16
Table 2: Geometrical Parameters of the Manzanares Prototype.....	19
Table 3: Physical properties of the materials used in the simulation	20
Table 4: Environment parameters used in the simulation	21
Table 5: Boundary conditions in this case.....	28
Table 6: Phi and Theta divisions and pixels CFD analysis	30
Table 7: Overall solver settings.....	30
Table 8: Velocity results of the different geometries.....	32
Table 9: Skewness metrics	34
Table 10: Overall metrics best and worst values.....	35
Table 11: Results of grid independence test for the CFD model.....	36
Table 12: Mesh properties	37
Table 13: Max, Min and default values for the GEKO model parameters	46
Table 14: Results of the different GEKO parameters.....	46
Table 15: K- ω models comparison	48
Table 16: k- ω models results	48
Table 17: k- ϵ models' comparison.....	52
Table 18: k- ϵ models results	52
Table 19: Transition SST and Spalart Allmaras results.....	53
Table 20: Results for the different connections	59
Table 21: Results of the different inclined collectors	63
Table 22: Results of the different inclined chimney walls	64
Table 23: Results of the optimized geometry	66
Table 24: Rayleigh number values at the start of the collector	67
Table 25: Results for bigger scale plant.....	68
Table 26: Different results for different scales of the optimized geometry	69

List of figures:

Figure 1: PV panels.....	10
Figure 2: PV principle	10
Figure 3: Parabolic trough plant.....	10
Figure 4: Concentrating solar tower	10
Figure 5: Exact location of the Manzanares plant in google maps	18
Figure 6 : Representation of size comparison of the plant with a human (3D model using SketchUp).....	19
Figure 7: Parts of the SUT.....	20
Figure 8: SUT's boundaries in ANSYS	21
Figure 9: Pressure staged wind turbine used in Manzanares	22
Figure 10: Boundary conditions of the case.....	27
Figure 11: Cartesian system of ϕ and θ divisions (Picture from ANSYS guide).....	28
Figure 12: Pixelation of a control volume (Picture from ANSYS guide)	29

List of figures:

Figure 13: Different geometries for the CFD case	31
Figure 14: Depiction of aspect ratio on a triangular shape	33
Figure 15: Depiction of aspect ratio on a square	33
Figure 16: Depiction of skewness on a Quad	33
Figure 17: Depiction of skewness on a triangle	34
Figure 18: Vectors for Orthogonal quality	34
Figure 19: Display of mesh in the chimney parts	36
Figure 20: Display of mesh in the side of the chimney	37
Figure 21: Element quality range of the mesh elements	39
Figure 22: Position of the elements with the poorest element quality metrics	40
Figure 23: Orthogonal quality range of the mesh elements	40
Figure 24: Position of the elements with the poorest orthogonal quality metrics	41
Figure 25: Skewness range of the mesh elements	41
Figure 26: Position of the elements with the poorest skewness metrics	42
Figure 27: Aspect ratio range of the mesh elements	42
Figure 28: Position of the elements with the poorest aspect ratio metrics	43
Figure 29: Velocity validation	54
Figure 30: Temperature validation	55
Figure 31: Power validation	55
Figure 32: Pressure contour for the standard geometry	56
Figure 33: Velocity contour for the standard geometry	57
Figure 34: Different connections between the collector and the chimney wall	58
Figure 35: Mass flow at case 1 and 2 for the 15-degree geometry	59
Figure 36: Velocity Vectors showing the recirculation area in case 2 and the difference with case 3	59
Figure 37: Connection in case 4	60
Figure 38: Gauge Pressure contour for case 4	61
Figure 39: SUT principle 1 degree	63
Figure 40: 2 degrees chimney wall angle	63
Figure 41: 1 degree chimney wall angle	63
Figure 42: 3 degrees chimney wall angle	64
Figure 43: Velocity contour for Case 9	65
Figure 44: Pressure contour for Case 9	65
Figure 45: Pressure contour for the optimized geometry	65
Figure 46: Velocity contour for the optimized geometry	65
Figure 47: Increase of Power for the bigger geometries	68
Figure 48: % change of Power in optimized bigger geometry	69

Chapter 1. Introduction:

The first chapter includes an introduction to the solar updraft tower principle, comparing it with other solar energy systems like photovoltaics and concentrating solar power plants and explains the importance of these renewable energy systems. In addition, shows information about the first SUT prototype in Manzanares and other small laboratory scaled plants that were built.

1.1 Renewable energy importance:

Nowadays, the world deals with challenges in energy demand, environmental crisis, and there is a need for reduction of greenhouse gas emissions. So, renewable sources of energy are playing an important role to stop pollution and the global warming. As the time passes, the world is more focused in the development of renewable energy technologies. [1] The total energy consumption from fossil fuels was 81.47 % during 2023 [2] and creates environmental issues. If, instead of fossil fuels, renewable sources were encouraged, the problem of polluting the environment can be solved. From all the renewable sources of energy, solar thermal energy is the most abundant form of energy which is available in direct and indirect forms. [1] One of many promising technologies is the solar updraft towers SUTs. SUTs can produce electricity, using the physics of natural convection and solar irradiation. The mechanical system of an SUT is very simple with no moving parts and it does not need water for cooling. Thus, it is very friendly to the environment and has potential for development. [3] The buoyancy effect which is the basic principle of SUTs was first described by the famous Italian Leonardo da Vinci. The use of this mechanism to create power was described first by Isidoro Cabanyes in 1903. However, the concept of the plant was first proposed by Hans Gunther in 1931, [1] until 1982 J Schlaich and his team made it reality by constructing the first big scale prototype in Manzanares. [1]

1.2 PV and CSP technologies:

Photovoltaic systems (PVs):

One of the most important renewable technologies is photovoltaic systems or PVs. [4] They transform the energy from the photons of the solar rays, to electric current and then electricity is produced directly. Discovered in 1839 by French physicist Edmond Becquerel [5] , the pv effect is the principal where solar cells convert the sunlight into electricity. [4]

A solar cell is made from 2 semiconductors. An n (negative) type silicon layer and a p (positive) type. The negative part has extra electrons and the positive has extra spaces for the electrons which are called 'holes'. [5]

The modern solar cells have extra layers and more complex materials in order to increase their efficiency. For example a passivation layer is added to the back of the area of the cell to reduce recombination of electrons and holes. [5] Also , antireflection coating are added to ensure that the most of the photons will contribute to the generation of the electricity. [5] PV systems are already very developed and are developing very fast due to their practicability, high efficiency and environmental friendliness. [4]

Different materials like crystalline silicon and thin films are under development to increase the efficiency and cost of the photovoltaic systems. [4]

Considering their environmental friendliness and cost reduction, pv systems play an important role in the global energy demand. Globally, the installed PV capacity has reached 1.2TW by the year 2022 and 1.6TW by the year 2023. [6] The most popular countries with PV installations are China, US and India. China continuous to install more than half of the worlds PV systems every year while India has 77 percent increase from the previous year and US has 55 percent. [7] There is a cost to this rising of PV systems and it is the waste that is created. It is estimated that by the year 2050, there will be more than 78 million tons of PV waste and it will be a challenge to deal with it. [4]

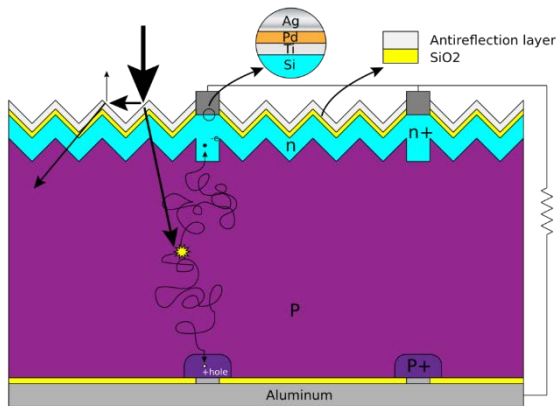


Figure 2: PV principle



Figure 1: PV panels

Concentrating Solar Power (CSP) Plants:

Concentrating Solar Power plants CSP systems convert the solar energy into thermal energy using solar collectors. [8] There are a lot of designs and ways to heat the different collectors, but they have the same basic idea to use mirrors to reflect the solar rays in one specific direction/region, called a reciever. [9] Inside the reciever, a working fluid passes and gets heated and prepared to go through a steam turbine and produce electricity. [9] The main CSP technologies are four. Parabolic troughs, linear Frensel reflectors [8], solar towers [9], and parabolic dishes. [8] All these technologies have different ways to produce electricity from concentrating solar energy. [10] The different systems are depicted in the following figures:



Figure 4: Parabolic trough plant



Figure 3: Concentrating solar tower



Figure 5: Parabolic dish



Figure 6: Linear Fresnel reflectors

Some CSP systems use thermal energy storage TES and have high-capacity factors with the use of molten salt [11], so electricity can be produced in the night. The region in which an CSP plant will be installed plays a very important role. Locations like Middle East and North Africa MENA region, Southwest US and Spain, are best suited for big scale CSP plants [10], but for the lower DNI regions, the photovoltaics are more competitive. [10]

1.3 Updraft tower principle:

The solar Updraft tower plants SUTs or known as solar chimneys convert the solar radiation energy into electricity, using simple physics of natural convection and radiation. It is made up of 3 major parts. The collector, the tower and the turbine. [3] The collector is a large area made from a transparent material like glass or plastic. [3] The larger the collector is, the more energy is available for electricity production. The next and very crucial part of the system is the tower. The height of the tower plays an important role for the output power. As the height increases, the pressure difference between the air inside the chimney and the outside air is bigger, resulting in better efficiency and power generation. [3] The final component of the plant is the wind turbine. This pressure staged type of turbine reminds more of the hydroelectric systems where static pressure is converted to power using the principle of enclosed turbine systems. [12]



Figure 7: Manzanares solar power plant



Figure 8: Manzanares Tower

An SUT plant has similar working principle as a hydroelectric power plant, with the difference that uses air instead of water. [13] Solar rays pass through the collector and heat the air and the ground beneath it. This is known as the greenhouse effect. [1] The heated air becomes less dense than the air in the environment and the buoyancy effect gives motion to the air and drives it in the center of the chimney base. [13] The hot air passes through the turbine that is located a couple meters high of the ground and the kinetic energy from the air is converted to mechanical energy and the turbine starts to rotate. [13] Afterwards, the mechanical energy is converted to electric power with the help of a generator.

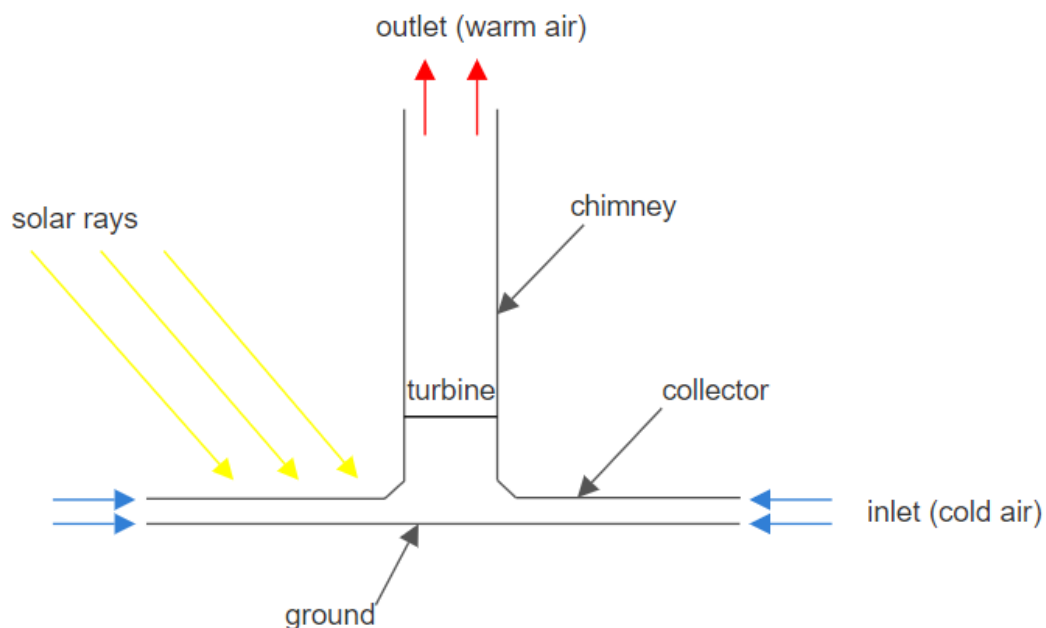


Figure 9: SUT principle

1.4 Comparison between the SUT and other technologies:

Advantages of SUT compared to PV and CSP:

Simplicity and Low Maintenance:

SUTs principle is less complicated. Natural convection is only necessary to drive the wind turbines and generate power. They have lower maintenance than CSP and PV systems because water is not needed for cooling and with no moving parts, there less probability for mechanical failures. Also, there is no need for other materials like molten salt and the heat transfer system is simpler than CSP systems. [14]

No Water Required:

SUTs don't need water for cooling, unlike CSP plants which require large amounts of water.

Multi-functional Potential:

SUTs can also act like hybrid systems and work like large scale greenhouses for agriculture and generate electricity at the same time, unlike CSP and PV which use the land only for electricity production. [14]

Ecological Advantages:

They emit no pollutants and might even clean the air in cities by carrying hot, polluted air to higher altitudes. [14]

Resilience in Harsh Environments:

SUTs can be used in desert areas where PV efficiency is lowered by hot temperatures or where CSP could have problems due to limited water availability.

Drawbacks with SUT compared to PV and CSP:

Inefficient Conversion of Energy:

The efficiency of SUTs is so much lower compared to PV and CSP systems. It is highlighted in the study [14] that chimney efficiency and system-wide energy conversion are becoming major limitations often lower than 1%. This study [14] concludes that due to that low efficiency, SUTs cannot compete with existing mature and modular renewable technologies, especially with photovoltaics, if the aimed use is only for electricity generation. Instead, SUTs can work as hybrid systems by creating electricity while being a large-scale greenhouse area for agricultural applications. [14]

High land area requirements:

By contrast, the energy generation intensity EGI of SUTs is much lower compared to PV and

CSP systems. For instance, PV panels produce much more electricity per unit area, so they can be installed in smaller areas more easily than SUTs. [14]

High Upfront Construction Costs:

SUTs require massive infrastructure, which includes a large chimney and vast solar collection surface area, leading to huge up-front costs. [14]

Structural Obstacles:

The construction and upkeep of tall chimneys that can endure environmental pressures, such as wind forces, present significant engineering difficulties, particularly for unconventional designs which propose very large chimneys. [14]

Seasonal and Geographic Variability:

Those characteristics of climate and geography, therefore, have strong influences on the electrical generation of SUTs that show strong seasonality. For example, Baise in China had much higher variability than that of Muscat in Oman. [14]

Developmental Maturity and Scalability:

SUT technology is still immature, with less practical implementation. On the other hand, PV and CSP systems are already well established, modular, and have proven reliable for big scale projects. [14]

Summary:

Solar Updraft Towers (SUTs) are less competitive with PV and CSP at utility scale and due to its lower efficiency coupled with high land and structural requirements. However, there are special applications for these systems like hybrid designs (e.g. power generation combined with agricultural greenhouses) [14] or in cases where water is limited, and a lot of land is available. [14] Also, SUTs are the most environmentally friendly solutions and have a possibility to develop in the future due to the environmental crisis.

1.5 Manzanares Solar Updraft Tower Plant:

Manzanares Solar Updraft Tower plant was the first every big scale plant and it was constructed in 1982 by German structural engineer J. Schlaich and his team. [13] The project was funded by the German ministry of research and technology [15], while the land distribution was done by Spanish Utility Union Electrical Fenosa. [15]

The plant was composed by a 195 meter tall tower of 10 meters diameter and a circular collector about 244m diameter (45.000m^2) and its peak power was approximately 50kW. [16] The transparent material of the collector was plastic membrane which covered 40000m^2 of the collector area while the rest was made of glass. [16] The height between the ground and the collector cover was 1.85 meters which helped the warmer air navigate through the chimney [16] and across the turbine, which was a 4 bladed, shrouded, pressure staged wind turbine with FXW 151-A blade profile. [17] The average air velocity was 12m/s and the temperature rise was 17 C inside the greenhouse. [16] The experiment was successful and was operating and collecting data from 1982 to 1989 until it was decommissioned. [16] This data validated the SUT concept and caught the attention of a lot of researchers who compose new ideas and hybrid SUT plants. [1]



Figure 10: Manzanares SUT chimney

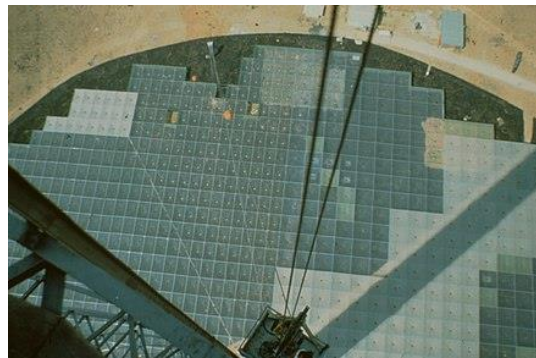


Figure 11: Tower collector from glass and plastic

1.6 Other small-scale plants and proposals:

In 1983, Krisst [18] made a small scaled solar chimney plant in Connecticut, USA which was 10 meters high, had a 6 meters collector radius and could produce 10 watts of electricity. [1] after that, Kulunuk [18] created a small experiment plant in Izmit, Turkey in 1985 which had a potential of producing 0.14W and was only 2 meters high. [1] The first prototype which was made in China was created by Zhou et al. [19] and it was capable of generating 5 Watts with a 8 meter tall chimney and 10 meters collector radius. In 2010, Kasaeian et al [20] build a 12 meters high plant and calculates a maximum air velocity of 3m/s. Maia et al [21] in 2009 developed a bigger prototype with 12.3 meters height and 26 meters radius in Brazil, while in Iran, Kerman city a 60 meter chimney was created by Najmi et al. [1] In the same year, at Mutah university In Jordan, Al-Dabbas [1] demonstrated a small-scale prototype. In Iran, a plant with some special modification of a small tank that could store a huge volume of air, was made by Shahreza and Imani [22] in 2015. This change was proven beneficial and increased the heat flux by 20 percent. Okada et al. [23] estimated

the difference between a circular and a diffuser type plant and found that the diffuser type showed better results. [1] The most recent experiments were performed in India [24], [25], [26], [27], [28], [29] and demonstrated the climatic conditions there.

It is clear that all the projects were created on a laboratory scale, except the Manzanares prototype.

A lot of researchers [1] did small scale experiments as shown in the next table:

Table 1: Experimental research and their values:

Study	Location	Hch (m)	Dcoll or Acoll (m / m ²)	vmax (m/s)
Pasumarthi and Sherif, 1998 [30]	Gainesville, Florida, USA	7.92	9.15 m	-
Zhou et al., 2007 [19]	HUST, Wuhan, China	8	10 m	–
Akbarzadeh et al., 2009 [31]	Bundoora, Australia	8	4.2 m	1 – 1.5
Kasaeian et al., 2011 [20]	University of Zanjan, Iran	12	10 m	2.9
Mehla et al., 2011 [32]	NIT Hamirpur, India	0.8	1.4 m	0.53
M.A. Al Dabbas, 2012 [33]	Mutah University, Jordan	4	36 m ²	7
Raney et al., 2012 [34]	Texas, USA	0.203 – 7.52	11.58 m	0.5 – 2
Maia et al., 2009 [21]	Belo Horizonte, Brazil	12.3	26 m	1.1 – 2.7
Kalash et al., 2013 [35]	Damascus University, Syria	9	12.5 m ²	2.9
Motoyama et al., 2014 [44]	Fukuoka, Japan	2	3 m	1.7 – 1.8
Nasirivatan et al., 2015 [45]	University of Tehran, Iran	2	3 m	0.7 – 1.21
Okada et al, 2015 [23]	Fukuoka, Japan	0.439	0.66 m	1.02
Shahreza and Imani, 2015 [22]	Islamic Azad University, Iran	1.5	0.92 m	5.12
Ghلامchi et al., 2016 [36]	University of Tehran, Iran	2	3 m	1.3
Lal et al., 2016 [37]	Kota, India	8	12 m	6 -12.2
Ahmed and Patel, 2017 [38]	University of the South Pacific, Fiji	4	3.2 m	5.4
Sivaram et al., 2018 [24]	NIT Trichy, India	0.5	0.2 m	0.5 – 0.88
H.H Al-Kayiem et al., 2018 [25]	University Technology PETRONAS, Malaysia	6	6 m	1.76 – 1.81
Mehla et al., 2019 [26]	Panchkula, India	1.78	1.78 m	–

Balijepalli et al., 2020 [27]	NIT Warangal, India	6	3.5 m	2.18 – 2.98
Belkhode et al., 2020 [28]	Nagpur, India	3.6 – 4.8	4.5 m	1.8 – 2
Das and Chandramohan, 2021 [29]	NIT Warangal, India	6	3.5 m	2.66

1.7 CFD case in this thesis:

The goal of this thesis is to analyze the influence of different turbulence models, solar discretization and geometries in the mass flow, pressure, velocities and Power. In chapter 2, the tower design is introduced, including the dimensions, the materials, the turbine, and the location of the plant is determined. Following chapter 3, the equations, performance parameters, boundary conditions and models are determined. In chapter 4, the domain is selected, and the mesh is analyzed to proceed for the running of the simulations. So, in chapter 5, through testing different turbulence models and solar discretization parameters, the RNG k- ϵ turbulence model and the discrete ordinate (DO) solar model were chosen on the ANSYS fluent software for the reference geometry case, which is based on the Manzanares plant. Finally, when the case has been validated, in chapter 6 the simulations of the changes of the collector angle, the connection between the collector and chimney wall and the incline wall of the chimney are shown. The last simulations will test bigger scales of the plant and find what potential a bigger SUT has. Final chapter 7 presents the conclusion of this thesis.

Chapter 2. Tower design:

In this chapter, the design of the case will be represented, including the location and the sun's direction, the scale and dimensions of the geometry, the materials that are used and finally the type of turbine which will be the same with the Manzanares Prototype.

2.1 Location / solar rays' direction:

The position of the sun is calculated using a solar ray tracing model inside ANSYS fluent. [39] The solar ray tracing option is very efficient, and its meaning is the applying solar loads as heat sources in the transport equations. [39] This model includes the solar calculator which is a tool to create the sun's exact location in the sky for a given day, time and location. Also, the solar model is available only in 3D cases. [39] That's the reason that most researchers prefer to use 3D models. The values of the position are identical with the Manzanares Plant location while the time of the day is set at 1 o'clock on 21nd of June, while the time zone is +2GMT. The latitude of this location is 38.99° and the longitude is 3.37° west. [40] [41] In the following figure, the exact location of the Manzanares plant is depicted using google maps.



Figure 5: Exact location of the Manzanares plant in google maps

2.2 Geometry and surfaces:

An up-close size comparison of the plant with a human is shown in the next figure to understand the dimensions of the SUT.

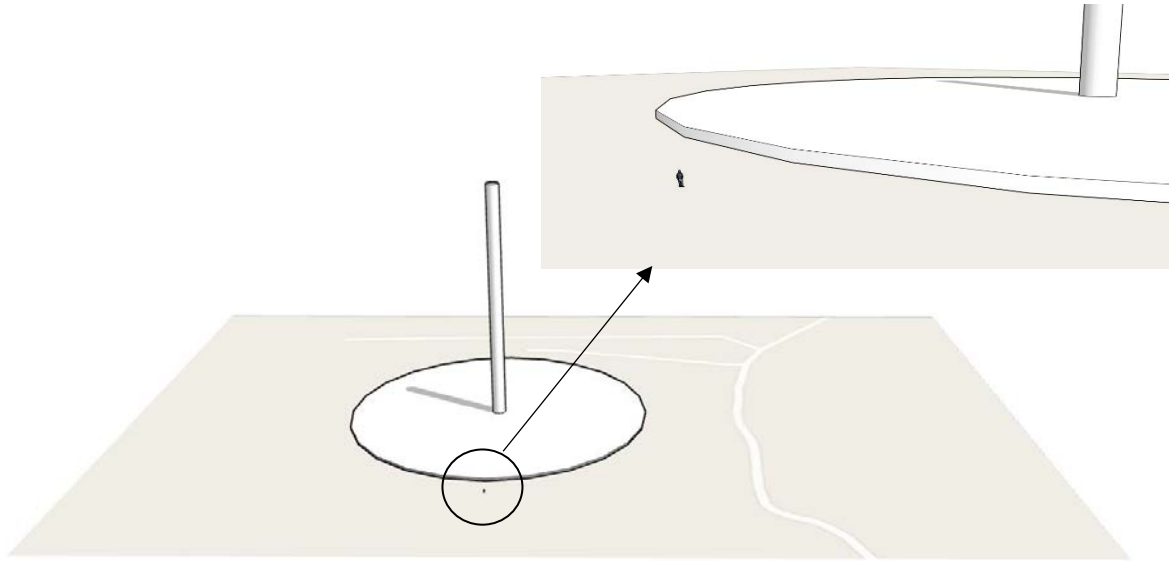


Figure 6 : Representation of size comparison of the plant with a human (3D model using SketchUp)

It is known that thicknesses of ground, collector, and chimney are 0.5 m, 0.004 m, and 0.00125 m, respectively. The Collector has a height of 1.85 meters from the ground and a radius of 122 meters. [40] [41] Also, the chimney is 194.6 meters in height and has a radius of 5.08 meters as shown in the following table:

Table 2: Geometrical Parameters of the Manzanares Prototype

Geometrical Parameters	Value
Collector height	1.85 m
Collector radius	122 m
Chimney height	194.6 m
Chimney radius	5.08 m

In this geometry, a 45-degree chamfer is used near the base of the chimney as shown in the following figure:

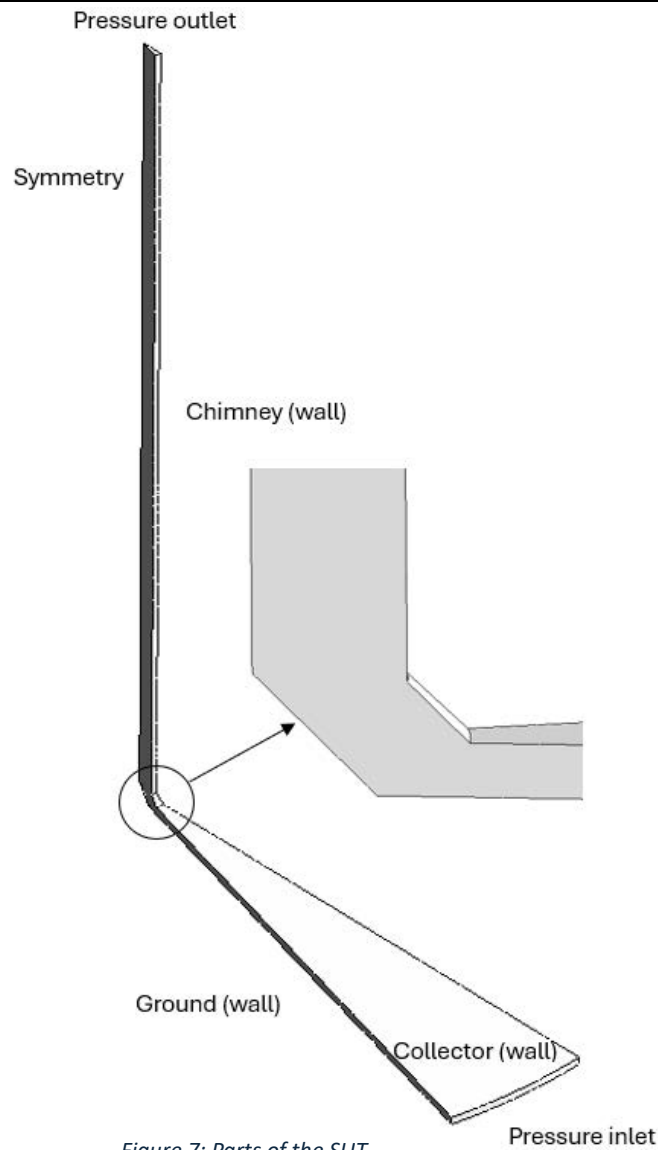


Figure 7: Parts of the SUT

2.3 Materials:

The materials that are used in these simulations are glass for the collector and steel for the tower. [12] Their properties are shown in the following table:

Table 3: Physical properties of the materials used in the simulation

Physical properties (unit)	Glass	Ground	Chimney
Density (kg/m^3)	2500	2160	2719
Thermal conductivity (W/Mk)	1.15	1.83	202.4
Specific heat (J/kgK)	750	710	871
Absorption coefficient	0.03	0.9	0
Transmissivity	0.9	Opaque	Opaque

Emissivity	0.1	0.9	1
Refractive index	1.526	1	1
Thickness (m)	0.004	5	0.00125

The values of the environment parameters are shown in the next table:

Table 4: Environment parameters used in the simulation

Parameter	Value	Units
Solar radiation (G)	1000	W/m ²
Ambient pressure (P_{atm})	101325	Pa
Ambient temperature (T_a)	293.15	K
Gravitational acceleration (g)	9.81	m/s ²
Ambient air density (ρ)	1.2046	kg/m ³
Ideal gas constant (R)	287	J/(kg·K)
Air conductivity (k)	0.0259	W/(m·K)
Air kinematic viscosity (ν)	1.7984×10^{-5}	m ² /s
Air heat capacity (cp)	1006.43	J/(kg·K)
Stefan-Boltzmann constant (σ)	5.667×10^{-8}	W/(m ² ·K ⁴)

2.4 Turbine type:

The Manzanares prototype used a 4 bladed, shrouded, pressure staged wind turbine with FXW 151-A blade profile. [17] It was mounted 9 meters above ground. [42]

Turbines in a solar tower convert mechanical energy in the form of rotational output by harvesting the airflow inside the tower. Unlike conventional wind turbines working on variable velocities, the turbines are designed to work as enclosed pressure staged wind turbo generators. [12] This is similar to hydroelectric plants where static pressure is converted to rotational energy using the principles of enclosed turbines. [12]

The power output per swept area of the rotor in such pressure staged turbines is about one order of magnitude larger than what has been achieved in velocity staged wind turbines. To be sure, the air speed is roughly constant before and after passing through the turbine. The energy output obviously depends on the volume flow rate and on the pressure difference across the turbine. [12]



Figure 9: Pressure staged wind turbine used in Manzanares

The turbine control system changes the pitch of the blades to optimize energy production under varying conditions. When the flat areas of the blades are perpendicular to the airflow, the turbine does not rotate. When the blades are perfectly aligned with the airflow, the air passes through without any drop in pressure and no energy is produced. The optimal blade position is between these two cases. The maximum power is produced when the pressure difference across the turbine is about 80% of the total available pressure difference. [12]

Chapter 3. Modeling:

In Chapter 3, the mathematical equations which are included in ANSYS FLUENT would be represented. Also, the performance parameters of the case will be determined to help calculate the Power Output, while the boundary conditions of the problem are given. In the end of the chapter, angular discretization is introduced, and 3 cases will be performed to select the parameters for the solar load model.

3.1 Equations:

The continuity equation for a 3D space and a steady state problem is as follows:

Equation 1

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

The momentum equations for a 3D space and a steady state problem are as follows:

Equation 2

$$\begin{aligned} \frac{\partial}{\partial x}(\rho u u) + \frac{\partial}{\partial y}(\rho u v) + \frac{\partial}{\partial z}(\rho u w) &= -\frac{\partial P}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \\ \frac{\partial}{\partial x}(\rho v u) + \frac{\partial}{\partial y}(\rho v v) + \frac{\partial}{\partial z}(\rho v w) &= -\frac{\partial P}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \\ \frac{\partial}{\partial x}(\rho w u) + \frac{\partial}{\partial y}(\rho w v) + \frac{\partial}{\partial z}(\rho w w) &= -\frac{\partial P}{\partial z} + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \end{aligned}$$

The energy equation is as follows:

Equation 3

$$\begin{aligned} \frac{\partial}{\partial x}(P c_p u T) + \frac{\partial}{\partial y}(P c_p v T) + \frac{\partial}{\partial z}(P c_p w T) \\ = \lambda \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \end{aligned}$$

RNG $k - \varepsilon$ turbulent model is used to estimate the air flow inside the solar chimney. This model takes account the swirl effect on turbulence to predict the swirling flows. The fluid is an ideal gas.

The kinetic energy and the dissipation rate equations of this model are as follows:

Equation 4

$$\begin{aligned} \frac{\partial}{\partial x_i}(\rho k u_i) + \frac{\partial}{\partial t}(\rho k) &= \frac{\partial}{\partial x_j} \left(\mu_{\text{eff}} \alpha_k \frac{\partial k}{\partial x_j} \right) + G_k \\ &\quad + G_b - \rho \varepsilon - Y_M + S_k \\ \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) + \frac{\partial}{\partial t}(\rho \varepsilon) &= \frac{\partial}{\partial x_j} \left(\mu_{\text{eff}} \alpha_\varepsilon \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (C_{3\varepsilon} G_b + G_k) \\ &\quad - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon. \end{aligned}$$

Discrete ordinate (DO) radiation model is used for solving the radiative heat transfer equation. This is the right model for the semitransparent collector (glass).

Equation 5

$$\begin{aligned} \nabla \cdot (I(\vec{r}, \vec{s}) \vec{s}) + (a + \sigma_s) I(\vec{r}, \vec{s}) \\ = n^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{r}, \vec{s}') \omega(\vec{s} \cdot \vec{s}') d\Omega' \end{aligned}$$

Where I is the solar radiation intensity, $\vec{r}$, $\vec{s}$ and $\vec{s}'$ denote the position, direction and scattering direction vectors. Also a and σ_s are the absorption and scattering coefficients and n , ω and Ω' are the refractive index, phase function and solid angle.

The efficiency of the ray tracing approach has led us to select it for simulating solar loads in the energy equation within this model. To replicate conditions like those in Manzanares, the position of solar radiation at a specific date and time is determined using a solar calculator. The geographic coordinates are set to 38.99° north latitude and 3.37° west longitude.

The flow behavior within the solar chimney system is represented using the Rayleigh number, a dimensionless parameter defined as follows.

Equation 6

$$Ra = \frac{g\beta\Delta TL^3}{\alpha\nu} \quad or \quad Ra = \frac{g\beta\Delta TL^3}{\nu} Pr, \quad \text{where } Pr = \frac{\nu}{\alpha} \text{ is the Prandtl number, } \alpha = \frac{k}{\rho \times c_p}$$

Where g is the gravity acceleration, ΔT is the temperature difference between the collector and the ground, L is the height of the collector, α and ν are the thermal diffusion coefficient and kinematic viscosity and β is the thermal expansion coefficient which is $1/T_a$ for ideal gases, where T_a is the ambient temperature. The material properties are given in table 4.

3.2 Performance parameters:

The equation of the mass flow rate inside the chimney is expressed by the following [40] :

Equation 7

$$\dot{m} = \rho Q,$$

Where Q is the air volume flow rate. Collector efficiency (η_{coll}) is a parameter which depends on solar intensity and is calculated as follows [40]:

Equation 8

$$\eta_{coll} = \frac{\dot{Q}}{A_{coll} G}$$

Where G is the solar intensity, A_{coll} is the area of the collector and $\dot{Q}$ is the input thermal power which is calculated with this expression [40]:

Equation 9

$$\dot{Q} = \dot{m} c_p (T_1 - T_2) = \dot{m} c_p \Delta T$$

Where ΔT is the rising temperature at the collector. The pressure (ΔP) is determined by the pressure contours of the CFD results.

The real Power is with the following expression [40] [41]:

Equation 10

$$P_o = \eta_t \Delta P_t Q_v.$$

Where η_t is the turbine generation efficiency and is takes as 0.8 [40] [41]. Although there are different parameters that affect the performance of SCPPs, the two main factors that influence power are the pressure drop (ΔP_t) in the turbine and the volumetric flow rate Q_v . In this thesis, the pressure drop in the turbine is obtained by the average pressure where the turbine is placed, and it is calculated as follows [40] [41]:

Equation 11

$$\Delta P_t = r_t P_t.$$

Where r_t is the turbine pressure drop ratio, and it is considered as 2/3. [40] [41]

3.3 Boundary conditions:

The boundary conditions of the surfaces in the SCPP are shown in table 5. Air flows into the collector inlet with a temperature of 293.15 K under a pressure inlet boundary condition, while it leaves at the same temperature from the chimney outlet under a pressure outlet condition. Atmospheric pressure is maintained at both the inlet and outlet. On the contrary, an adiabatic wall boundary condition is set with heat flux q set to 0 W/m², is applied to the chimney surface. The ground wall is modelled as an opaque surface, while the collector surface is thermally set by a mixed wall thermal boundary condition with fixed heat transfer coefficient h of 9.5 W/m²K and is treated as semitransparent. The ambient air temperature is set at 293.15 K. For the computational cost, the symmetry boundary condition is assumed for two surfaces as illustrated in the next figure.

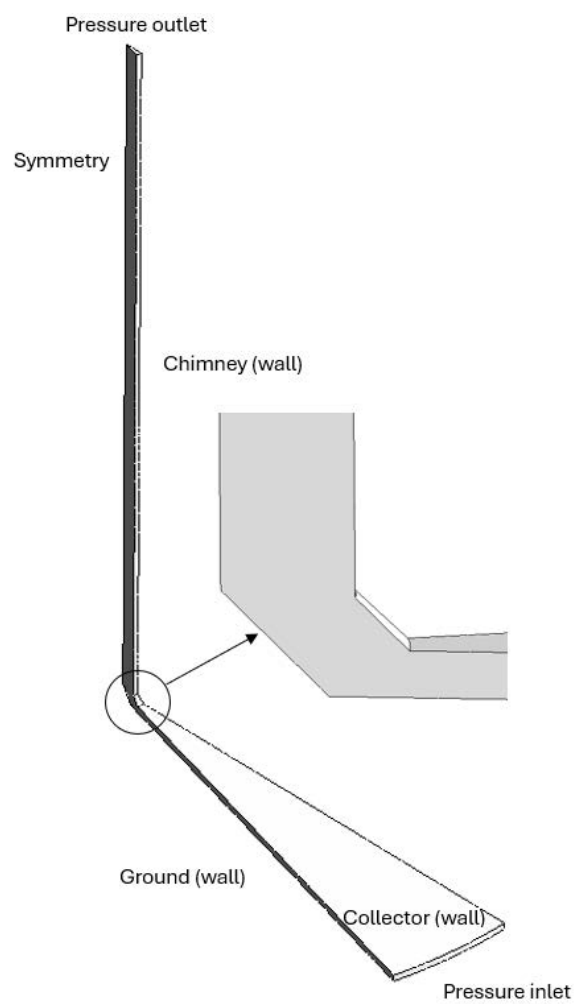


Figure 10: Boundary conditions of the case

Table 5: Boundary conditions in this case

Surface	Boundary Condition Type	Parameter	Value
Collector Inlet	Pressure Inlet	Temperature	293.15 K
		Pressure	Atmospheric
Chimney Outlet	Pressure Outlet	Temperature	293.15 K
		Pressure	Atmospheric
Chimney Surface	Adiabatic Wall	Heat Flux (q)	0 W/m ²
Ground Wall	Opaque Wall	-	-
Collector Surface	Mixed Wall Thermal Boundary Condition	Heat Transfer Coefficient (h)	9.5 W/m ² K
		Treatment	Semitransparent
Ambient Air	-	Temperature	293.15 K
Symmetry Surfaces	Symmetry	-	-

3.4 Solver and radiation model:

The SIMPLE solution method will be employed for the pressure-velocity coupling in ANSYS Fluent. Gradient computation will be performed using Green-Gauss cell-based methods, and pressure spatial discretization techniques have been chosen as PRESTO! Momentum, turbulence, energy, and discrete ordinates were discretized using a second-order upwind scheme.

Theta and Phi pixels and divisions are tools to control the number of solar rays being traced for each surface. [39]

Theta and Phi Divisions:

Theta N_θ and Phi N_ϕ divisions define the number of control angles used to discretize each octant of the angular space. [39] Each octant in the angular space 4π is discretized into $N_\theta \times N_\phi$ solid angles. The angles θ and ϕ are the polar and azimuthal angles and are measured with the use of the cartesian system as shown in the figure below. [43]

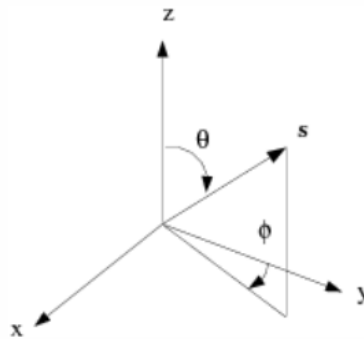


Figure 11: Cartesian system of ϕ and θ divisions
(Picture from ANSYS guide)

For a 2d model, ANSYS will solve 4 octants. So a total of $4 \times N_\theta \times N_\phi$ directions will be solved. For a 3d model 8 octants will be solved, so $8 \times N_\theta \times N_\phi$ directions. For the most practical cases, the default number for phi and theta division is acceptable and its value is 2. Increasing this number, the model accuracy becomes better but there is a computational cost. [44]

Theta and Phi Pixels:

Theta and Phi pixels control the pixelation for any control volume as shown in the next figure. The default pixelation of 1×1 usually is very efficient, but in more complex problems, a 3×3 pixelations is suggested. [44]

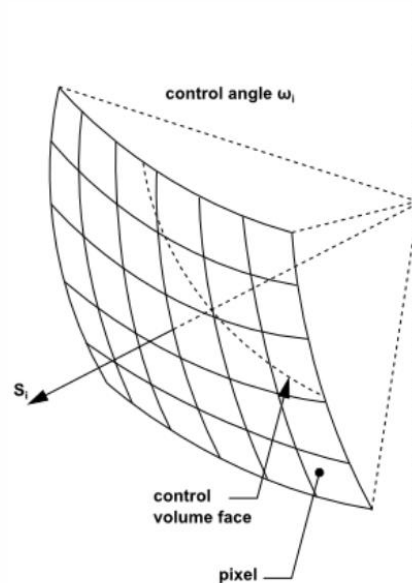


Figure 12: Pixelation of a control volume (Picture from ANSYS guide)

In this thesis, 3 cases of different divisions and pixelations are simulated. The first case has the default numbers of 2×2 for pixelation and 1×1 for division. The second case concludes 3×3 pixelation and 3×3 divisions. Last, the most advanced case 3 has 4×4 pixelation and 4×4 divisions. The simulations will show how these values affect the maximum velocity and if the more complex cases give better results. In the next table, the results are represented:

Table 6: Phi and Theta divisions and pixels CFD analysis

Angular discretization	1st case	2nd case	3rd case
Phi divisions	2	3	4
Phi pixels	1	3	4
Theta divisions	2	3	4
Theta pixels	1	3	4
Max velocity (m/s)	14.3	14.2	14.2

The results show that the phi and theta pixels and divisions do not impact on the results in this particular case. This is happening probably because geometry is not so complex, so the default values are preferable. Also, it was found that the computational cost rises a lot for higher numbers of angular discretization parameters. 1st case running time was about 2 hours while in the 2nd case was double the time close to 4 hours and finally in the 3rd case it rise to 9 to 10 hours. So, in this CFD case there is no need for higher pixels and divisions, and the default values are reliable and more useful.

Table 7: Overall solver settings

Parameter	Setting
Solver	- SIMPLE
	- 3D axisymmetric simulation
	- Steady-state (second-order upwind discretization)
Energy Equation	Activated
Viscous Model	RNG k- ϵ model
Radiation Model	DO (Discrete Ordinates)
Theta divisions	2
Phi divisions	2
Theta pixels	1
Phi pixels	1

Chapter 4. Domain selection and meshing:

The purpose of this chapter is to select, through 3 different cases, the best suited domain size and explain the mesh metrics to analyze the quality of the meshing. Also, a mesh independence test will be done to select the best mesh density in order to avoid extra computational cost.

4.1 Domain size selection:

Comparison will be done with a simulation using the 90-degree angle geometry with identical boundary and solver settings to see just how accurate the geometry at 15 degrees is. Both geometries are fillet and of overall dimensions identical to the Manzanares plant.

This meshes the 90-degree geometry with 491,020 elements and the 15-degree geometry with 83,050 elements. To check that the mesh densities of both cases are indeed comparable, one can see that $491,020/(15/90) = 81,838$. We will now compare the maximum velocity results from both simulations to see if the 15-degree geometry provides good results.

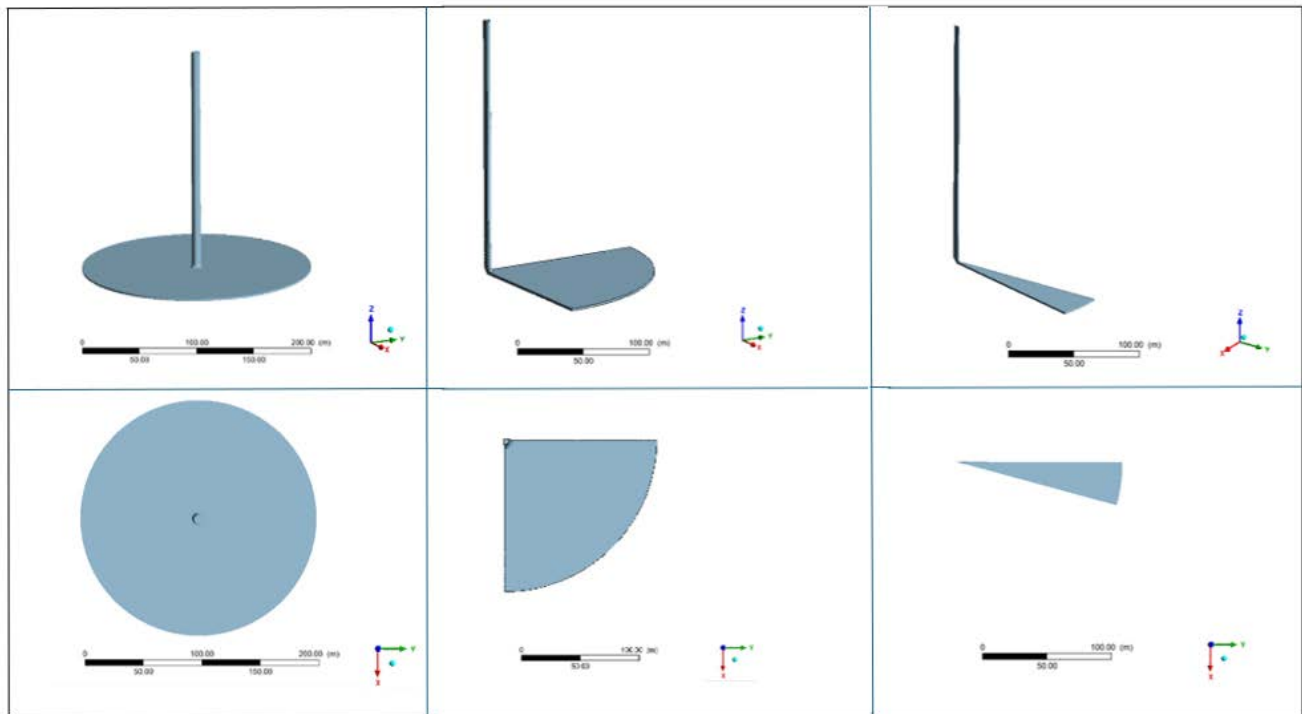


Figure 13: Different geometries for the CFD case

The following table shows the results from the different geometries:

Table 8: Velocity results of the different geometries

Geometry	Max Velocity	% change
360-degree angle	12.27	-
90-degree angle	11.66	4.97
15-degree angle	12.06	3.31

The results show that the 15-degree angle geometry is very accurate, very close to the 90-degree and closer to 360-degree geometry. The maximum velocity is nearly the same. So, the 15-degree angle geometry can be used in simulations to reduce the computational cost by a lot. If the 90-degree angle geometry was used and required to have the same mesh density as the 15-degree mesh that has 380530 elements, then the geometry would have $380530 \times (90/15) = 2283180$ elements. The computational cost, in both memory and time, generally rises with the number of elements roughly linearly to quadratically, depending on the solver complexity, the refinement level desired, and the convergence criteria. Therefore, doubling the number of elements at least doubles the computational cost in terms of time, while for memory and processing demands by nonlinear solvers, it is higher.

4.2 Mesh:

4.2.1 Mesh quality elements explanation:

The ANSYS mesh metrics can indicate quality and performance for the mesh employed during simulations, which are related to accuracy and convergence. [45] The key metrics will be set forth below:

Element quality: The element Quality is the ratio of the volume of the sum of the square of the edge lengths for 2D elements, but in this 3D case is the square root of the cube of the sum of the square of the edge lengths. The best value is 1 and means that the element is a perfect square or cube, and the worst is 0 which means that the element has a 0 or negative volume. [45]

Aspect ratio: The aspect ratio of a triangle compares the “height” and “width” of the triangle. It can take values from 1 to infinite. A triangle with a value of 1 is equilateral. [45] The following figures show the difference in an element with 1 and 20 value of aspect ratio.



Figure 14: Depiction of aspect ratio on a triangular shape



Figure 15: Depiction of aspect ratio on a square

Skewness: Skewness is one of the most important mesh metrics that clarify the mesh quality. It determines how close the ideal (equilateral or inequilateral) a face or cell is. [45] The following images demonstrate the difference between an equilateral and a skewed triangle/Quad:



Figure 16: Depiction of skewness on a Quad

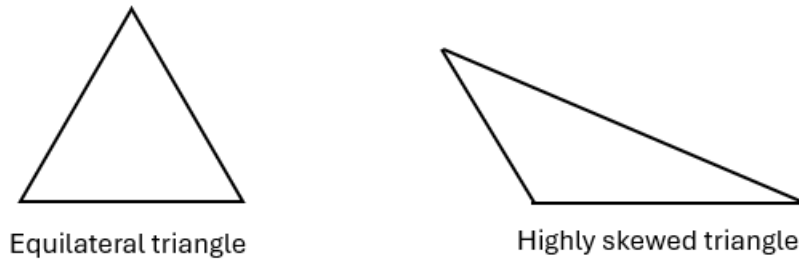


Figure 17: Depiction of skewness on a triangle

The following table shows the preferable skewness values:

Table 9: Skewness metrics

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

Orthogonal quality: Orthogonal quality is dependent on the angle between the vector that connects the two nodes or control volumes and the normal vector for each surface integration point associated with that edge. Value 1 means perfect alignment and thus perfect orthogonality and a value of 0 represents misalignment and poor quality. High orthogonal quality is important to erase numerical errors and make stable and accurate simulations. [4]

Equation 12

$$\min\left(\frac{\vec{A}_i \cdot \vec{f}_i}{|\vec{A}_i| \cdot |\vec{f}_i|}, \frac{\vec{A}_i \cdot \vec{c}_i}{|\vec{A}_i| \cdot |\vec{c}_i|}\right)$$

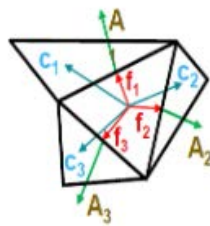


Figure 18: Vectors for Orthogonal quality

Overall metrics [4] as shown in the following table:

Table 10: Overall metrics best and worst values

METRIC	Best  Worst
Element Quality	1 0
Aspect Ratio - Triangle	1 20
Aspect Ratio - Quadrilaterals	1 20
Skewness - Triangular	0 1
Skewness - Quadrilateral	0 1
Orthogonal Quality	1 0

4.2.2 Mesh independence:

To present a more realistic prediction of the airflow inside the system, a mesh independence check was conducted using five grid sizes of 83238 (1), 175457 (2), 282946 (3), 380530 (4), and 536631 elements (5). The computed values of maximum velocity magnitude for the different mesh sizes are presented in the following table. The (1) and (2) meshes showed a poor result not even near the wanted velocity. The maximum velocity differences were 4.2% between (3) and (4), while it is only 0.7% between (4) and (5). In the light of these results and previous studies presented in the literature [40] [41], (4) was selected as a grid-independent solution and was used for further simulations.

Table 11: Results of grid independence test for the CFD model

Cell numbers	Maximum Velocity (m/s)	% change
83238	13.2	-
175457	13.2	0
282946	13.7	3.7
380530	14.3	4.2
536631	14.2	0.7

In the following figures, the mesh display of the different parts of the chimney is represented.

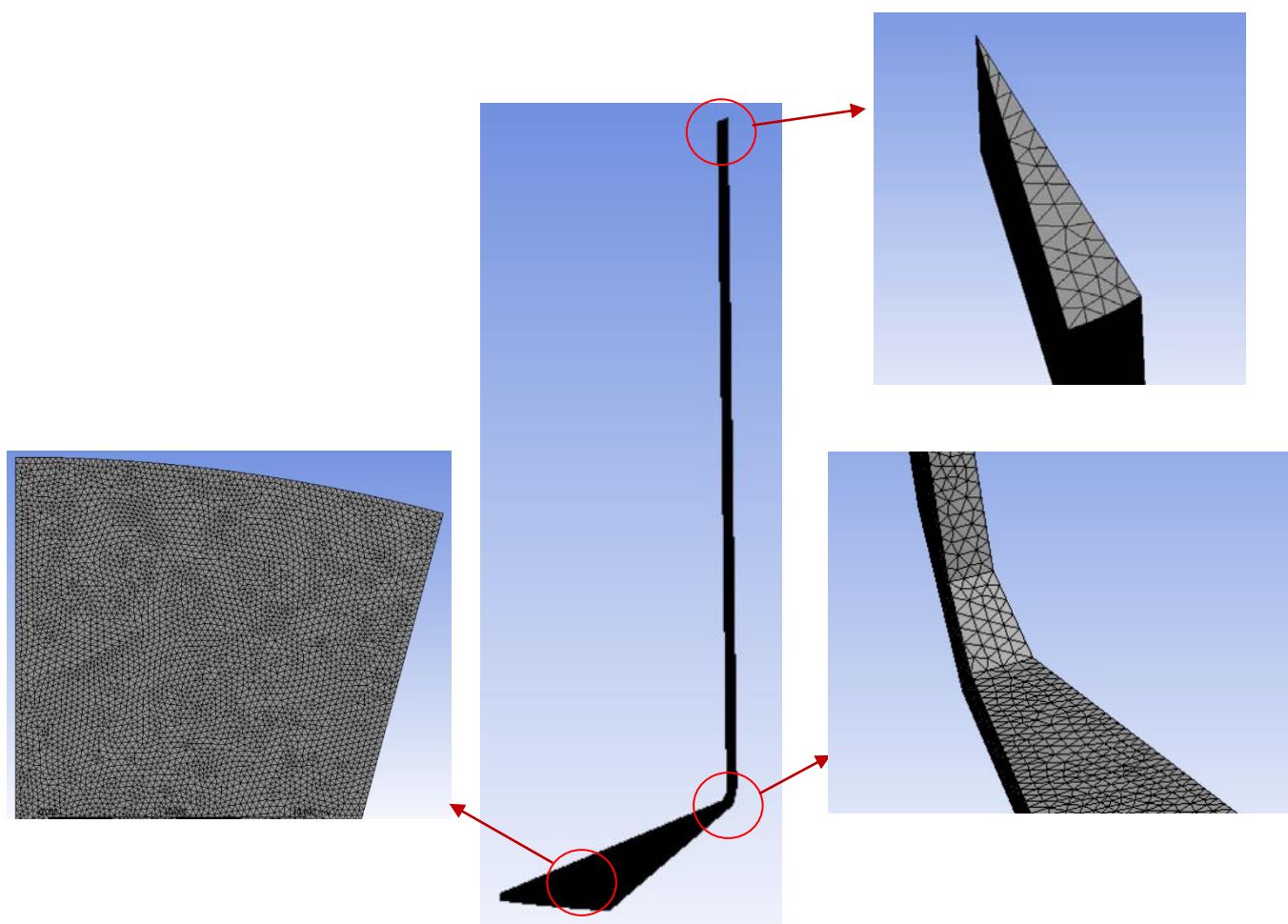


Figure 19: Display of mesh in the chimney parts

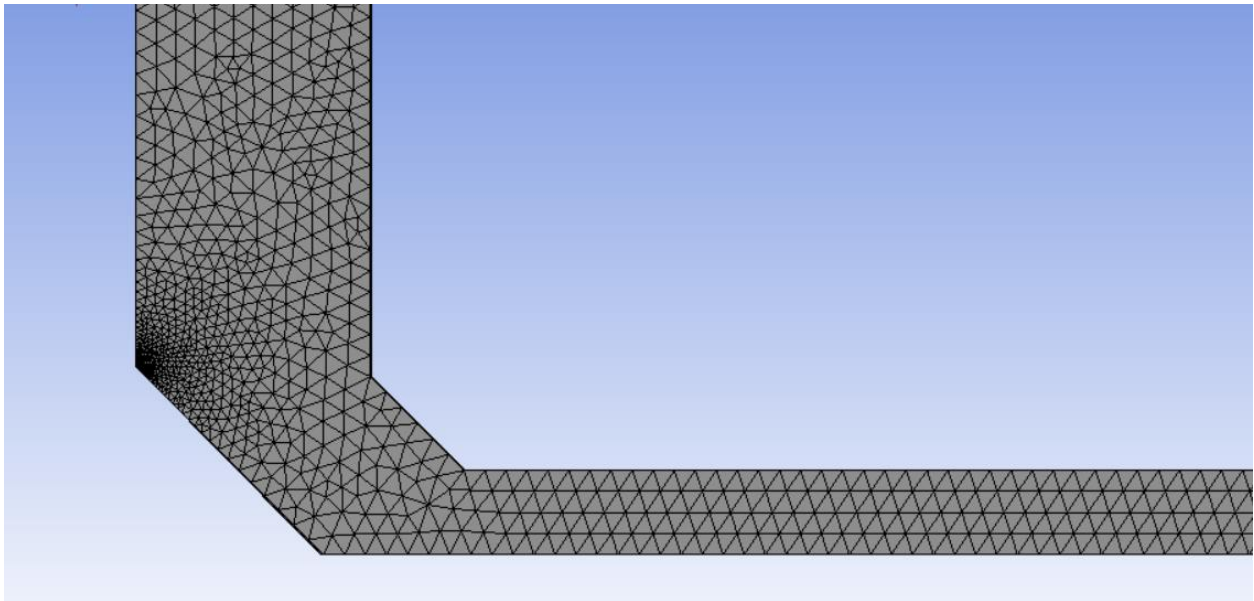


Figure 20: Display of mesh in the side of the chimney

Table 12: Mesh properties

Object Name	Mesh
State	Solved
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	0.45 m
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
Growth Rate	Default (1.2)
Max Size	0.8 m
Mesh Defeaturing	Yes
Defeature Size	Default (2.25e-003 m)
Capture Curvature	Yes

Curvature Min Size	Default (4.5e-003 m)
Curvature Normal Angle	Default (18.0°)
Capture Proximity	No
Bounding Box Diagonal	239.33 m
Average Surface Area	787.17 m ²
Minimum Edge Length	1.0472 m
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.900000)
Smoothing	High
Mesh Metric	Skewness
Min	1.15E-09
Max	0.845
Average	0.24465
Standard Deviation	0.13438
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Straight Sided Elements	
Rigid Body Behavior	Dimensionally Reduced
Triangle Surface Mesher	Program Controlled
Topology Checking	Yes
Pinch Tolerance	Default (4.05e-003 m)
Generate Pinch on Refresh	No
Statistics	
Nodes	82566
Elements	380530

4.2.3 Mesh quality results in this case:

The chosen mesh quality is shown in this final part of the chapter to obtain if the mesh is reliable and accurate. Also, the bad quality elements position will be depicted in the following figures.

The quality of the mesh is calculated according to many signs: - Element quality -Orthogonal quality - Skewness - Aspect ratio.

Element quality: Most of the values are in the range between 0.717 and 0.959, which is a good result as shown in the following figure:

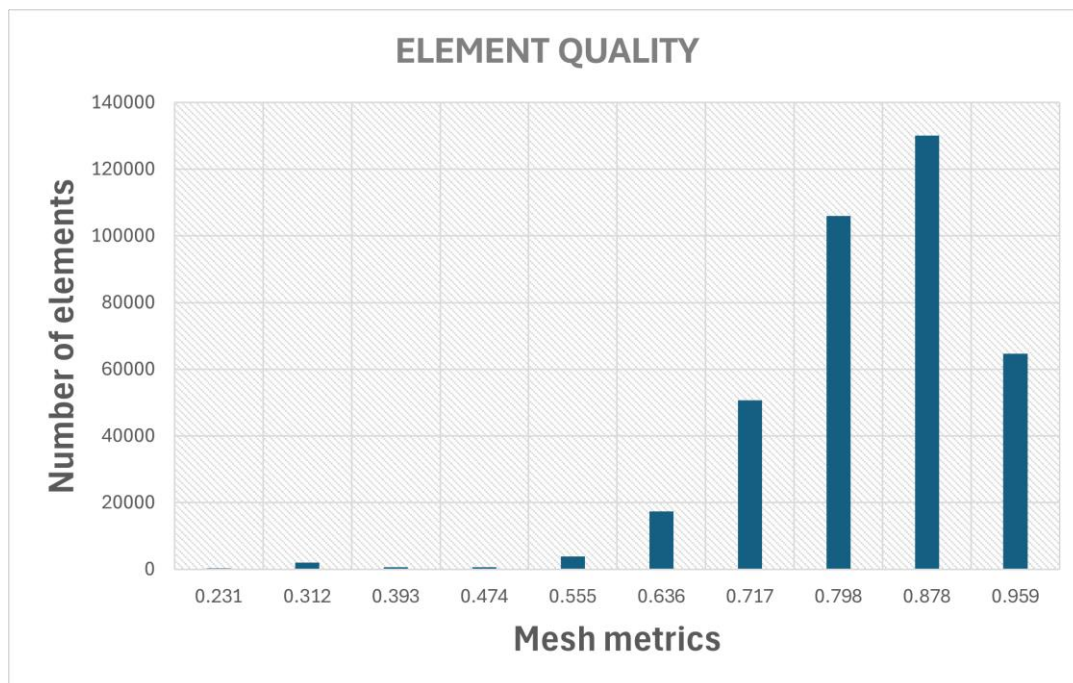


Figure 21: Element quality range of the mesh elements

In the next figure, the position of the elements with the poorest element quality is shown. It is noted that the elements are positioned in the left edge of the geometry where the symmetry is, as expected. These elements aren't in the most critical region (in the middle and base of the chimney), so their existence does not affect the flow.

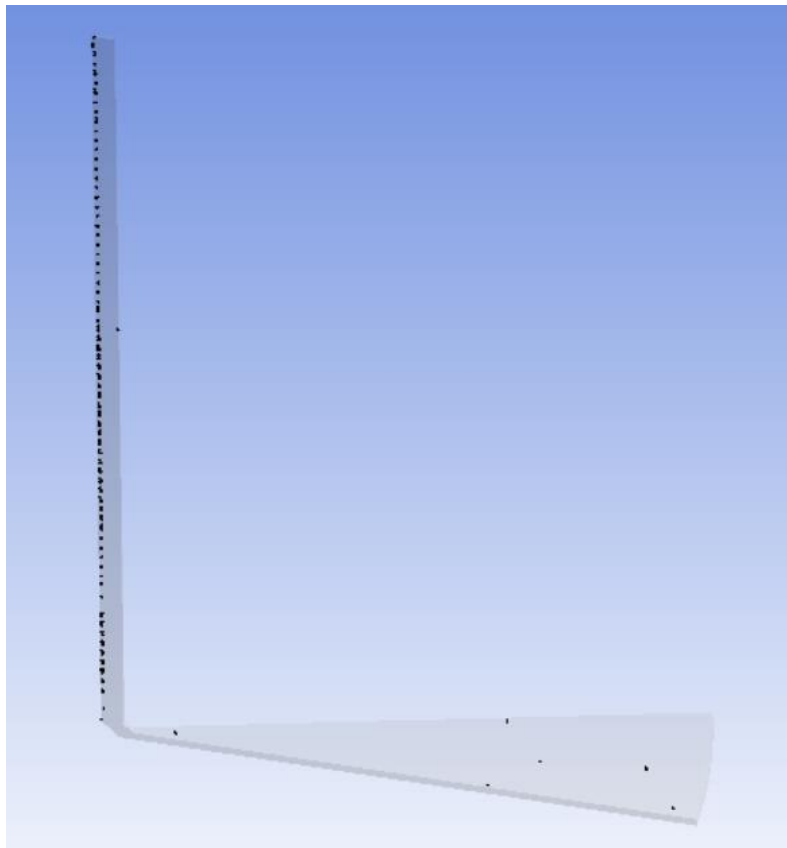
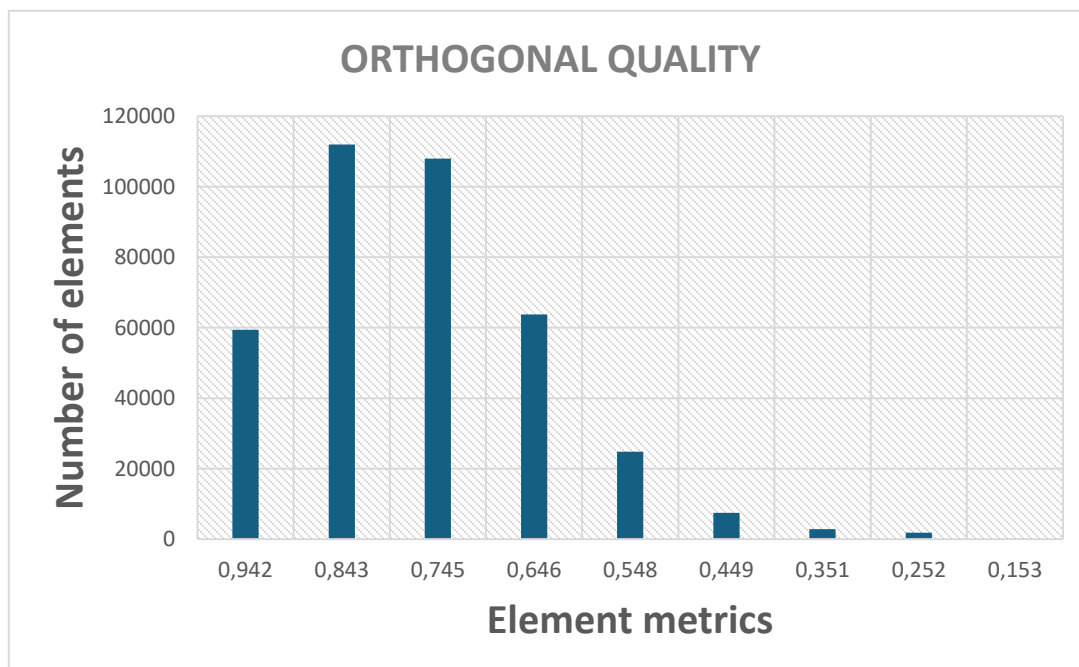


Figure 22: Position of the elements with the poorest element quality metrics

Orthogonal Quality: For the analysis there is a majority with values 0.646-0.942 which is a good value as shown in the following figure:



40 Figure 23: Orthogonal quality range of the mesh elements

Again, the same thing with the element quality metrics happens in the orthogonal quality metrics, where most of the elements are located in the left part of the chimney.

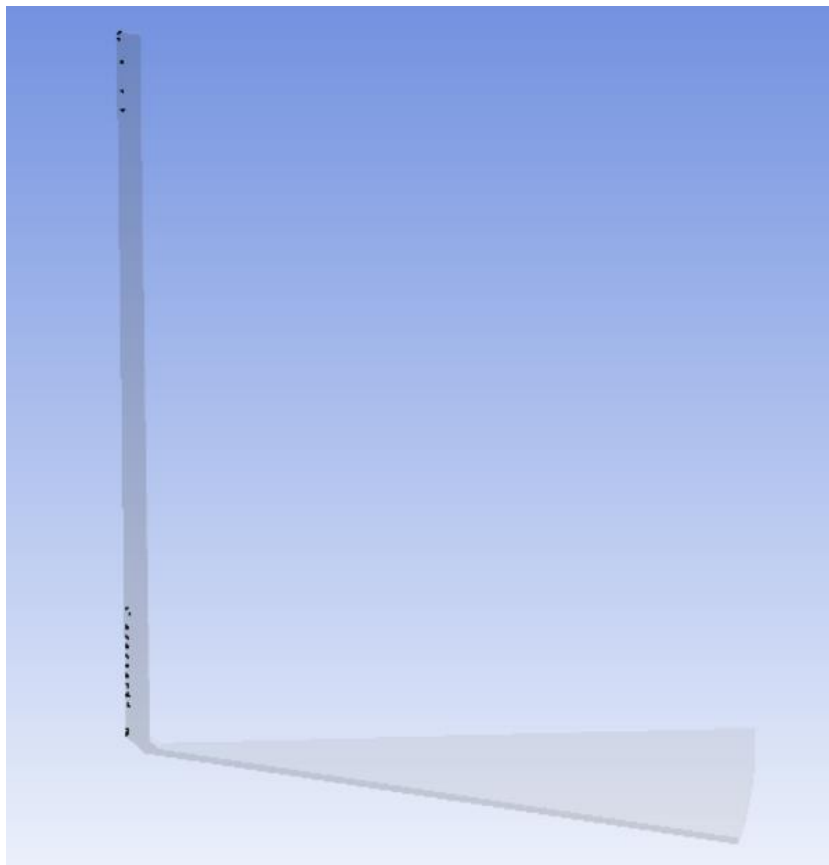


Figure 24: Position of the elements with the poorest orthogonal quality metrics

Skewness: The maximum value for skewness is 0.845 and most of them fall within the very good category between 0-0.447 as shown at the next figure.

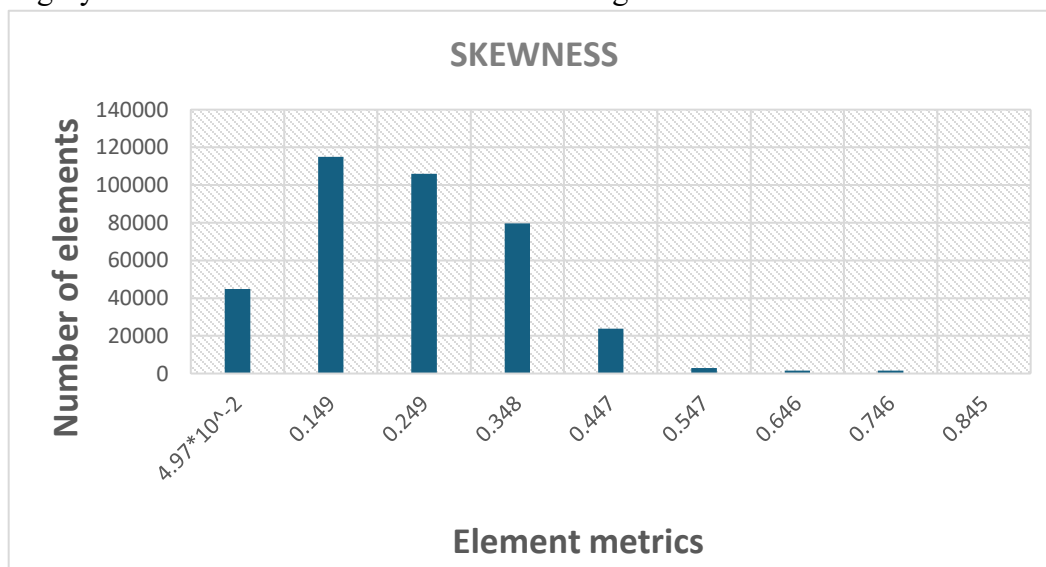


Figure 25: Skewness range of the mesh elements

The same thing with the two previous occasions happens in the skewness metrics also, where most of the elements are located in the left part of the chimney.

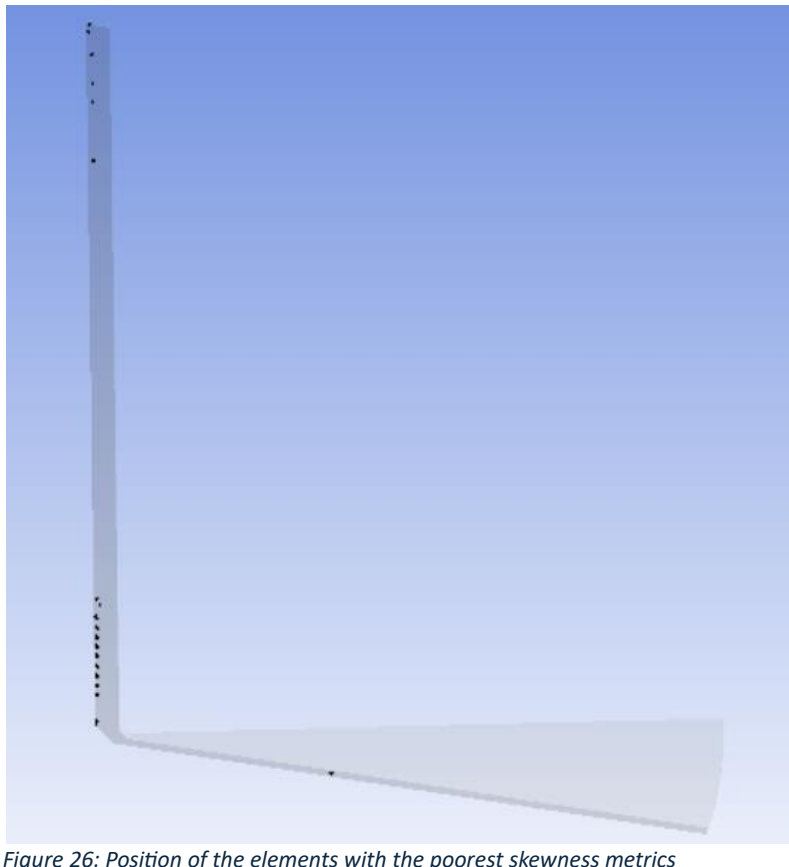


Figure 26: Position of the elements with the poorest skewness metrics

Aspect Ratio: 12 is the highest value that can be allowed and that is good value for such a type of analysis.

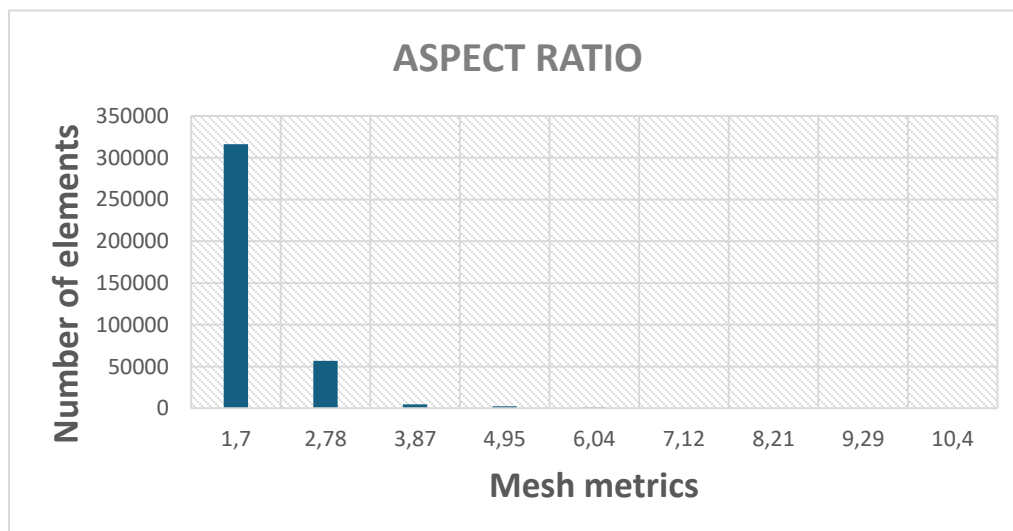


Figure 27: Aspect ratio range of the mesh elements

It is shown in the following figure that the elements with the poorest aspect ratio are very few and located in random positions of the geometry.

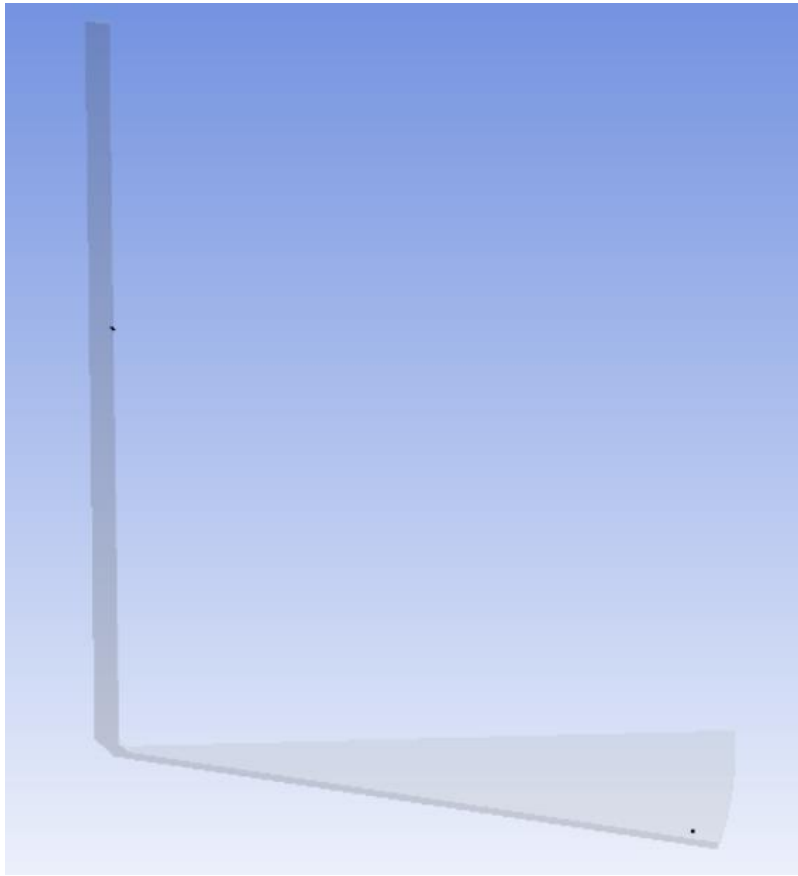


Figure 28: Position of the elements with the poorest aspect ratio metrics

The overall quality of the mesh is clean, and it's acceptable for CFD simulation with enough accuracy.

Chapter 5. Turbulence modeling and validation:

In this chapter, different turbulence models are analyzed and tested according to maximum velocity. The RNG result that the most researchers use is 14.3 m/s. This analysis will show if the other models can reach this value or surpass it. In the end, the case will be validated with the other researchers' results in terms of velocity, temperature and power.

5.1 k- ω and k- ϵ models comparison:

Reynolds-averaged Navier Stokes RANS models compute complex industrial flows. [46] The most common are k- ω and k- ϵ models. Both solve 2 more equations and introduce eddy viscosity (turbulent viscosity) to compute the Reynolds stresses. [46] These models have differences in their applications, advantages, and limitations. So, the choice of the model will depend on considerations such as the physics of the flow, the level of accuracy required, the computational resources and the amount of time available for the simulation. [46]

k- ϵ Turbulence Models:

The k- ϵ model is the fundamental two equation model for practical engineering. It is very popular for industrial flows and heat transfer simulations due to its robustness, economy and solid accuracy. [47]

Limitations:

The drawback of this model is that it is very sensitive to adverse pressure gradients and boundary layer separation. [47] They predict a delayed and reduced separation flow. This can result for false and optimistic evaluations of flows near smooth surfaces. That's the reason why k- ϵ models are not used in external aerodynamic CFD problems. [47] For that type of cases, the best solution is the Realizable k- ϵ model with a combination of enhanced wall treatment or the Menter-Lechner near-wall treatment. [47]

k- ω Turbulence Models:

The ω -equation is superior to ϵ -equation because it can be integrated without additional terms through the viscous sublayer. [48] This makes the formulation of a robust y^+ insensitive treatment relatively straightforward. For that reason, the k- ω models are better for predicting adverse pressure gradient boundary layer flows and flow separation. [48]

Limitations:

The standard k- ω model has a major limitation. It has a strong sensitivity in the solution depending of the freestream values of k and ω especially outside the shear layer. For that reason, the use of the standard model is not recommended. [48]

5.2 k- ω models:

The BSL and SST models have been designed to avoid the freestream sensitivity of the standard k- ϵ model by combining elements of the ϵ and ω equation. The GEKO model use is to offer a single model with flexibility. It has four parameters to adjust, and it is a powerful model for optimization but requires a proper understanding of these parameters. [48]

5.2.1 standard k- ω :

The standard k- ω model, an empirical turbulence model, is based on a model proposed by Wilcox. [49] The downside of this model is its sensitivity of k and ω values outside the shear layer. The newer versions of 2006 also couldn't solve the problem. [49]

The next equation is the transport equation for the standard k- ω model:

Equation 13

$$\begin{aligned}\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) &= \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k + G_b \\ \frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) &= \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega + G_{\omega b}\end{aligned}$$

Where G_k represents the generation of turbulence kinetic energy due to mean velocity gradients, G_ω represents the production of ω , Γ_k and Γ_ω represent the effective diffusivity of k and ω and Y_k and Y_ω represent the dissipation of k and ω due to turbulence. G_b and $G_{\omega b}$ effect the buoyancy and S_k and S_ω are user defined terms. [49]

5.2.2 GEKO k- ω :

The GEKO model is still not published. It is based on k- ω and its purpose is its ability to be tuned and optimized by the users. [50]

These are the 6 parameters that can be adjustable to tune the GEKO model [51] :

CCEP: Main parameter for adjusting separation prediction for boundary layers. It affects all flows and increasing its value reduces the eddy-viscosity leading to more sensitivity to adverse pressure gradients for boundary layers and to lower spreading rates for free shear flows. [51]

Cnw: It affects the inner part of wall boundary layers more and increasing it leads to higher wall shear stress and wall heat transfers in non-equilibrium flows. Also, it has an effect on non-generic flows like vortices, but it's not systematically tested. The most used and preferred value is the default. [51]

C_{MIX}: It affects only free shear flows and increasing its value leads to an increase in spreading rates of free shear flows. Its value is given by the correlation **C_{MIXCOR}** which connects **C_{MIX}** with **C_{CEP}**. [51] The following equation represents the correlation:

Equation 14

$$C_{\text{MixCor}} = 0.35 \text{sign}(C_{\text{Sep}} - 1) \sqrt{(|C_{\text{Sep}} - 1|)}$$

C_{JET}: It is active in a sub-model of **C_{MIX}** and it affects mostly jet flows. [51]

C_{CURV}: A model for curvature correction that can be combined with GEKO. [51]

In the following table, the minimum, maximum and default values of these parameters are shown:

Table 13: Max, Min and default values for the GEKO model parameters

Parameter	Minimum Value (MIN)	Maximum Value (MAX)	Default Value
C_{CEP}	0.7	2.5	1.75
C_{NW}	-2	2	0.5
C_{MIX}	0	1	C_{MIXCOR}
C_{JET}	0	1	0.9
C_{CURV}	0	1.5	1

In this thesis, a tuning of these parameters was done to see if the value of the velocity will increase and make the GEKO results closer with the other models. In the next table the values of the maximum velocity for the maximum and minimum values of these parameters are shown:

Table 14: Results of the different GEKO parameters

Parameters	Velocity (m/s)
DEFAULT	13.9
C_{CEP} = 0.7 (min)	13.7
C_{CEP} = 2.5 (max)	13.9
C_{NW} = -2 (min)	13.7
C_{NW} = +2 (max)	13.9
C_{NW} = 0	13.7
C_{CURV} = 0 (min)	13.7
C_{CURV} = 1.5 (max)	13.9
C_{JET} = 0 (min)	13.7
C_{JET} = 1 (max)	13.9
All max	14.2
All min	13.7

The results show that polyparametric tuning has no significant impact on this case. Only the minimum parameters show a 1.4 percent decrease in the maximum velocity. But when the parameters are all set to max values, the results improve and the maximum velocity becomes 14.2 m/s, very close to the RNG model.

5.2.3 BSL K- ω :

The BSL model was created by Menter and its purpose was to blend the accuracy of k- ω in the near-wall region with the freestream independence of k- ϵ model in the field, improving the standard k- ϵ . [52]

The BSL transport equation is very similar to the standard model and its represented below:

Equation 15

$$\begin{aligned}\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) &= \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k + G_b \\ \frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) &= \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega + G_{\omega b}\end{aligned}$$

The only difference is the term D_ω which is the cross-diffusion term [67] and it is calculated from this equation:

Equation 16

$$D_\omega = 2(1 - F_1)\rho \frac{1}{\omega \sigma_{\omega,2}} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$

5.2.4 SST model:

The SST model is the most accurate of all the k- ω models for a wider range of flows. It includes all the refinements of the BSL model and in addition it accounts for the transport of the turbulence shear stress in the definition of turbulent viscosity. [53]

The advantage of SST compared with the BSL is that it can predict the amount of flow separation from smooth surfaces.

Table 15: $k-\omega$ models comparison

Feature	Standard $k-\omega$	GEKO $k-\omega$	BSL $k-\omega$	SST $k-\omega$
Primary Use	General turbulent flows with simple boundary layers.	Wide range of applications, customizable via adjustable parameters.	General-purpose flows involving adverse pressure gradients.	Aerodynamics, separated flows, and boundary layer separation.
Accuracy	Moderate, sensitive to freestream conditions.	High, tunable for specific flow conditions.	Improved over standard $k-\omega$; reduces freestream sensitivity.	Best for accurate flow separation and adverse pressure gradients.
Flow Types	Adverse pressure gradients and boundary layers.	Complex geometries, separated flows, boundary layer transitions.	Moderate separation, freestream flows, boundary layers.	High-lift airfoils, aerodynamics, and turbomachinery flows.
Turbulent Viscosity Model	Eddy viscosity-based, isotropic.	Parameter-driven, adjustable for non-isotropic behavior.	Blended between $k-\omega$ and $k-\epsilon$ models.	Realizable, accounts for non-isotropic turbulence effects.
Computational Cost	Low	Medium, tuning may require additional effort.	Medium	Medium to High

Results of the simulation using the $k-\omega$ models are shown in the following table.

Table 16: $k-\omega$ models results

Turbulence model	max velocity (m/s)
STANDARD	13.7
GEKO (default)	13.9
GEKO (maximum)	14.2
BSL	13.7
SST	14.2

The SST model, as expected, is the most accurate with the best value close to the velocity that is calculated by the other researchers. [54] [55] [40] [41] Also, the GEKO model is very promising if it is tuned correctly.

5.3 k-ε models:

5.3.1 Standard k-ε:

Standard k-ε model transport equations including the kinetic energy k and dissipation rate ε [56]:

Equation 17

$$\begin{aligned}\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \\ \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon\end{aligned}$$

Eddy viscosity is computed with the following equation [56]:

Equation 18

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

All the default constants are shown below [56]:

$$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$$

These default values are tested and well suited for a wide range of wall-bounded [56] and shear flows and used in this thesis.

5.3.2 RNG k-ε model:

RNG model was made using a statistical technique called renormalization group theory. [57]

This model includes an extra term in its equation which improves the accuracy for rapidly strained flows. Also, the swirl effect on turbulence is included in RNG model and can make very accurate predictions. An additional advantage is that this model can work also for low Reynolds numbers, while the standard model can't work. [57]

That's the reason why the most researchers [54] [55] [40] [41] prefer to use the RNG model.

The RNG model transport equations are very similar to the standard and are represented below [57]:

Equation 19

$$\begin{aligned}\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) &= \frac{\partial}{\partial x_j} \left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \\ \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) &= \frac{\partial}{\partial x_j} \left(\alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon\end{aligned}$$

Where G_k represents the generation of turbulent kinetic energy due to the mean velocity gradients [57], G_b is the generation of turbulent kinetic energy due to buoyancy and Y_M represents the contribution of fluctuating dilatation in compressible turbulence to the overall dissipation rate. [57]

α_ε and α_k are the inversive effective Prantl numbers for k and ε and S_k and S_ε are user defined source terms. [57]

The eddy viscosity for the RNG model is different:

Equation 20

$$\begin{aligned}d \left(\frac{\rho^2 k}{\sqrt{\varepsilon \mu}} \right) &= 1.72 \frac{\hat{v}}{\sqrt{\hat{v}^3 - 1 + C_v}} d\hat{v} \\ \hat{v} &= \frac{\mu_{eff}}{\mu} \\ C_v &\approx 100\end{aligned}$$

5.3.3 Realizable k-ε model:

The modeled transport equations for the realizable k-ε model are shown below:

Equation 21

$$\begin{aligned}\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \\ \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \\ &+ \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon\end{aligned}$$

where:

Equation 22

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \eta = S \frac{k}{\varepsilon}, S = \sqrt{2S_{ij}S_{ij}}$$

$G_k, G_b, Y_M, S_k, S_\varepsilon$ represent the same values like the RNG and standard models, while C_2 and $C_{1\varepsilon}$ are constants and σ_k and σ_ε are the turbulent Prantl numbers for k and ε . [58]

Just like standard model, the eddy viscosity is computed from:

Equation 23

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

With the difference that C_μ is calculated with the following equation:

Equation 24

$$U^* \equiv \sqrt{S_{ij}S_{ij} + \tilde{\Omega}_{ij}\tilde{\Omega}_{ij}}$$

And

Equation 25

$$\begin{aligned} \tilde{\Omega}_{ij} &= \Omega_{ij} - 2\varepsilon_{ijk}\omega_k \\ \Omega_{ij} &= \overline{\Omega_{ij}} - \varepsilon_{ijk}\omega_k \end{aligned}$$

Where $\overline{\Omega_{ij}}$ is the mean of rate-of-rotation tensor viewed in a moving reference frame with the angular velocity ω_k . [58] A_0 and A_s are given by:

$$A_0 = 4.04, A_s = \sqrt{6}\cos \varphi$$

Where

Equation 26

$$\varphi = \frac{1}{3} \cos^{-1}(\sqrt{6}W), W = \frac{S_{ij}S_{jk}S_{ki}}{\tilde{S}^3}, \tilde{S} = \sqrt{S_{ij}S_{ij}}, S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$$

There are three turbulence models used in the literature of the analysis of the updraft solar tower. The $k-\epsilon$ is used by some researchers [54] [55] [40] [41] because it is accurate in many flows and does not depend on geometry or flow regime. Also, the $k-\epsilon$ realizable is used also depending on geometric characteristics. In most of the studies [54] [55] [40] [41], the RNG turbulence model is the most used model which is better with heavy turbulence flows with strong swirl and vortex effects. The difference between the $k-\epsilon$ models is shown in the next table:

Table 17: $k-\epsilon$ models' comparison

Feature	Standard $k-\epsilon$	RNG $k-\epsilon$	Realizable $k-\epsilon$
Primary Use	Simple turbulent flows	High strain rates, swirling	Complex flows with separation, boundary layer reattachment
Accuracy	Moderate	Improved for high strain rates	Best for complex geometries
Flow Types	Internal flows, jets	Swirling flows, mixing layers	Swirling flows, separated flows, boundary layers
Complexity	Basic, less computational cost	Higher due to additional terms	Higher due to advanced formulations
Turbulent Viscosity Model	Constant, isotropic	Adjusts based on flow features	Realizable (non-isotropic)
Computational Cost	Low	Medium	Medium to High

In this thesis, all the three models are tested with the same mesh elements and the same boundary conditions and solver.

Table 18: $k-\epsilon$ models results

Turbulence model	max velocity (m/s)
$k-\epsilon$ standard	13.3
$k-\epsilon$ RNG	14.3
$k-\epsilon$ Realizable	14.1

As expected, the result of the $k-\epsilon$ standard is not accurate. The result of the $k-\epsilon$ RNG has the best velocity value closest to the experimental result [59] and the other researchers [54] [55] [40] [41] [60] in the Manzanares plant. So, in this study, the $k-\epsilon$ RNG model is chosen to estimate the results of the air flow.

5.4 Other turbulence models:

5.4.1 Spalart-Allmaras Model:

Spalart-Allmaras model is very popular in turbomachinery applications, and it was designed for aerospace applications involving wall bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. This model is effective for lower Reynold number cases. It is not suggested to use it in industrial flows, and it can make errors for some free shear flows. [61]

This is the transport equation for the modified turbulent viscosity $\tilde{\nu}$:

Equation 27

$$\frac{\partial}{\partial t}(\rho \tilde{\nu}) + \frac{\partial}{\partial x_i}(\rho \tilde{\nu} u_i) = G_\nu + \frac{1}{\sigma_{\tilde{\nu}}} \left[\frac{\partial}{\partial x_j} \left\{ (\mu + \rho \tilde{\nu}) \frac{\partial \tilde{\nu}}{\partial x_j} \right\} + C_{b2} \rho \left(\frac{\partial \tilde{\nu}}{\partial x_j} \right)^2 \right] - Y_\nu + S_{\tilde{\nu}}$$

Where G_ν is the production of turbulent viscosity, Y_ν is the destruction of turbulent viscosity that occurs in the near wall region due to wall blocking and viscous damping, $\sigma_{\tilde{\nu}}$ and C_{b2} are constants and $S_{\tilde{\nu}}$ is a user defined term. [61]

5.4.2 Transition SST Model:

Transition SST model is based on a coupling of SST k- ω model with two extra transport equations, one for the intermittency and one for the transition onset criteria, in terms of momentum-thickness Reynolds number. [62]

The results of these two models are shown in the following table:

Table 19: Transition SST and Spalart Allmaras results

Turbulence model	max velocity (m/s)
Transition SST	14.1
Spalart Allmaras	14.3

The results are very close with those from the RNG model that is most common in this case, especially these from the Spalart Allmaras model.

5.5 Validation:

The model has been validated by comparing the predicted results with the measured data of the Spanish solar chimney power plant from HAAF et al. [59] and with numerical results published in previous works [54] [55] [40] [41] [60] as presented below.

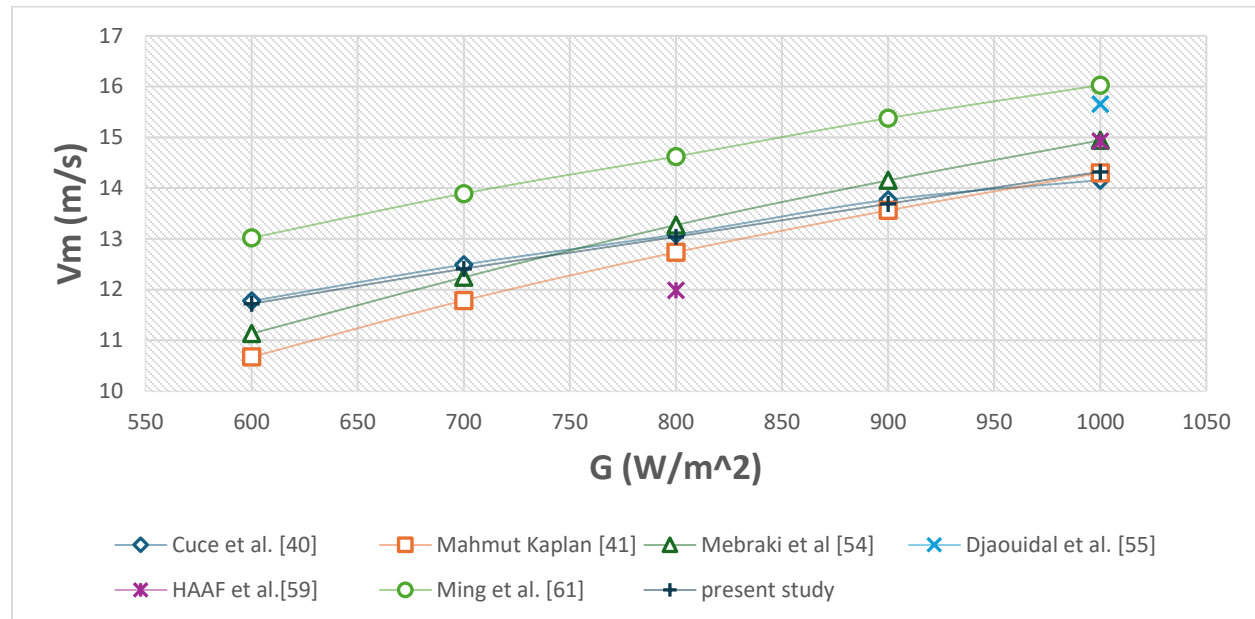


Figure 29: Velocity validation

In comparison to previous studies, the calculated maximum velocity (V_m) values align closely with experimental results at 800 and at 1000 W/m², as illustrated in Figure 26. The largest discrepancy between the experimental and predicted values is 8.07 % at 800 W/m². Additionally, the current numerical simulation shows strong agreement with the computational results reported by Cuce et al. [40] for the 600–1000 W/m² range.

One researcher [40] has the computational result of the temperature rise T_C in the collector at 600, 800 and 1000 W, using the same geometry of the Manzanares power plant. Compared with this study, the value of the temperature is very close to the Cuce et al research [40], especially at 600 W as shown in the next figure.

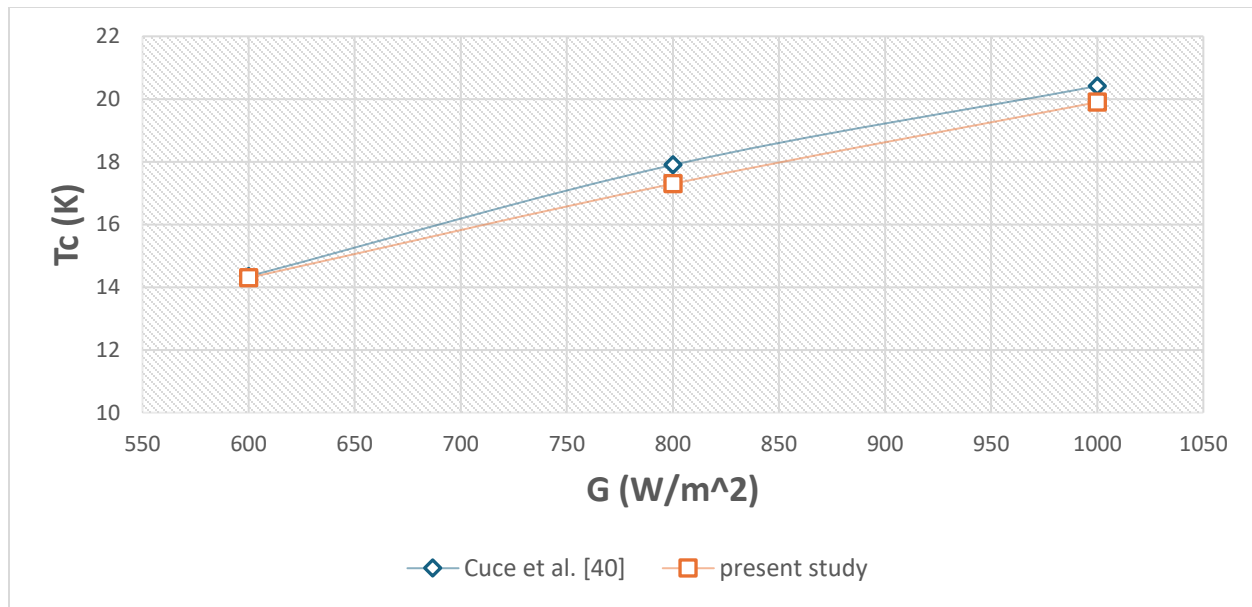


Figure 30: Temperature validation

Another important value to compare with other researchers is Power output (P) at the different irradiation (G) values. The comparison will be with the Cuce at al. [40] and the experimental results [59]. Again, geometry is the same as the Manzanares plant and the boundary conditions remain the exact same with the works of these researchers. The power output is calculated with equation 11, using the mass flow, the pressure and the density of the air at 9 meters high at the chimney (turbine). The results are shown in the following figure:

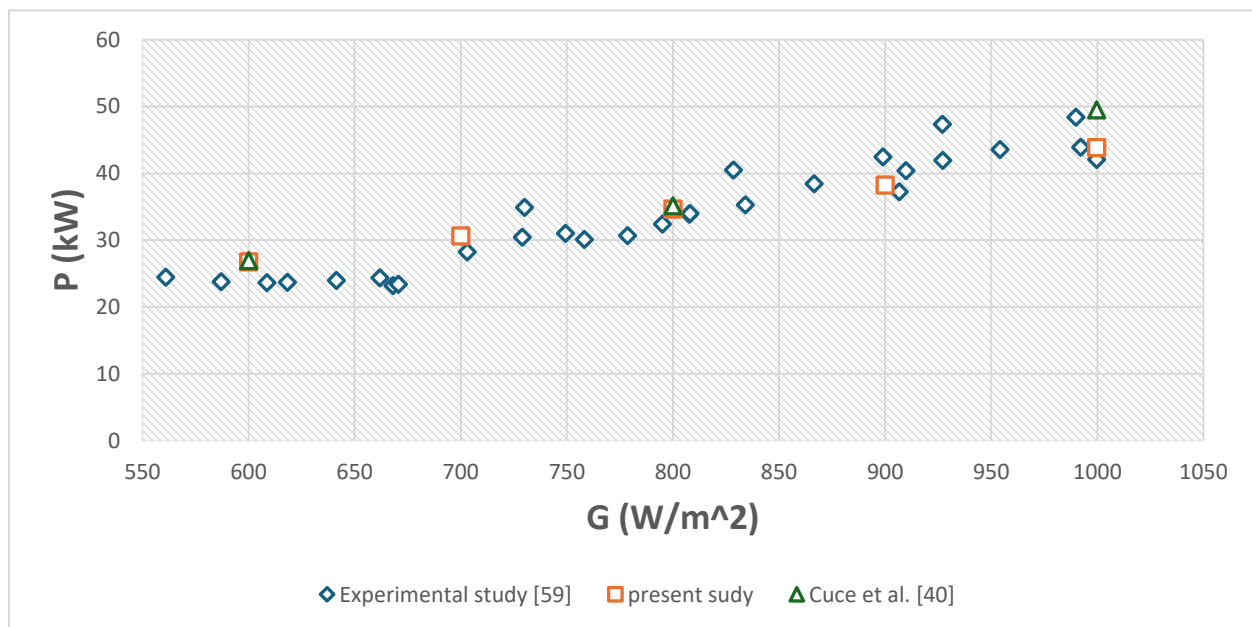


Figure 31: Power validation

Output value align very closely with the experimental results and is the exact same with the Cuce

et al. [40] at 600 and 800 watts. The biggest difference is 16.3% with the numerical results and very close to the experimental from 600 to 1000 Watts.

In conclusion, using the RNG for turbulence model and finding the optimal solar discretization parameters, the case in this thesis is validated and the results strongly agree with these of the other researchers. So, it is confirmed that the analysis is accurate and reliable.

Chapter 6. Results and discussion:

While having the case validated and having chosen the right models and parameters, in this chapter, the results for the reference geometry and variations of the case will be shown. Different geometries were made and will be tested to study if the power output, velocity and pressure results can improve.

6.1 Reference geometry results:

Results of the pressure contour across the chimney are represented in the following figure:

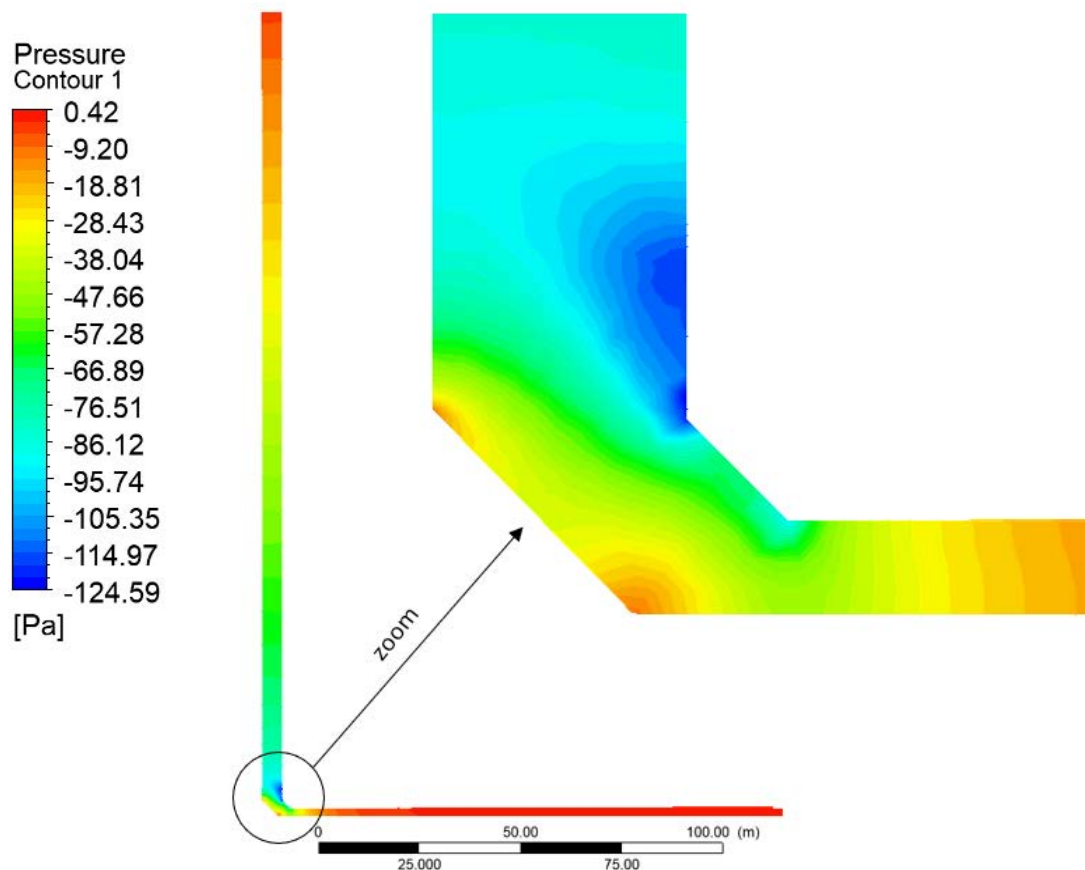


Figure 32: Pressure contour for the standard geometry

It is noted that the maximum static pressure is located in the corners of the chamfer as expected. Higher up the chimney, the static pressure stabilizes and becomes the same along the chimney's width. This location is important for the turbine placement which is 9 meters above ground.

Results of the velocity contour across the chimney are represented in the following figure:

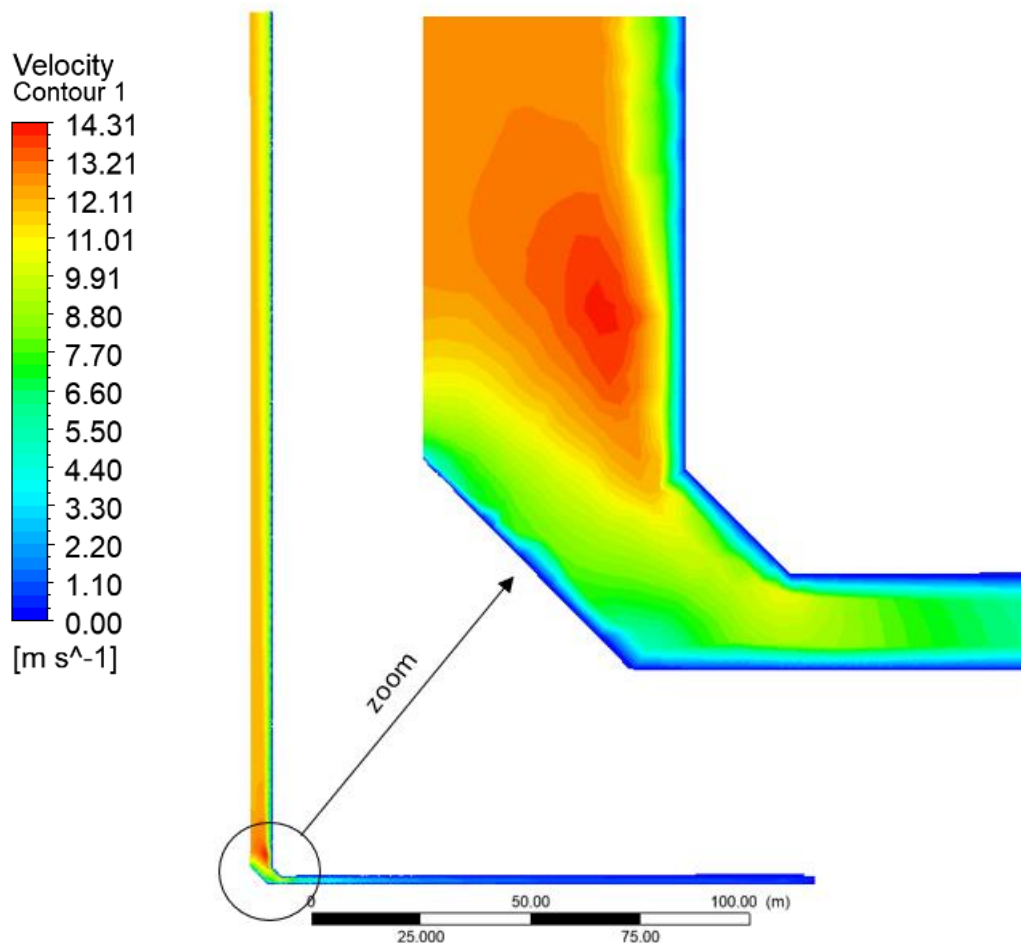


Figure 33: Velocity contour for the standard geometry

6.2 Geometrical variations effect:

For this thesis, modifications of the standard geometry will be done to find if the power generation can be improved. These modifications will be a different connection between the collector and the chimney (chamfer, curve, 90-degree corner, alternative curved geometry), an inclined collector (1,2,3,4 degrees) and finally an inclined chimney wall (1,2,3 degrees).

6.2.1 Focus on the collector and chimney connection:

Three different CFD cases are proposed to analyze geometry variations' impact at the interface between the collector and the chimney wall. Case 1: This includes a sharp 90-degree angle between the collector and the chimney wall. Case 2: introduces a chamfer to help the transition on the flow and probably increase the mass flow. Case 3: Between the collector and the chimney, a

curved incline with a radius of 2 meters is used, which shall help provide a smoother transition of flow into the chimney.

The fillets are replaced by two arcs. The radius of the small arc R1 is 2 meters and the radius of the bigger arc, R2, at the bottom of the chimney is 4 meters.

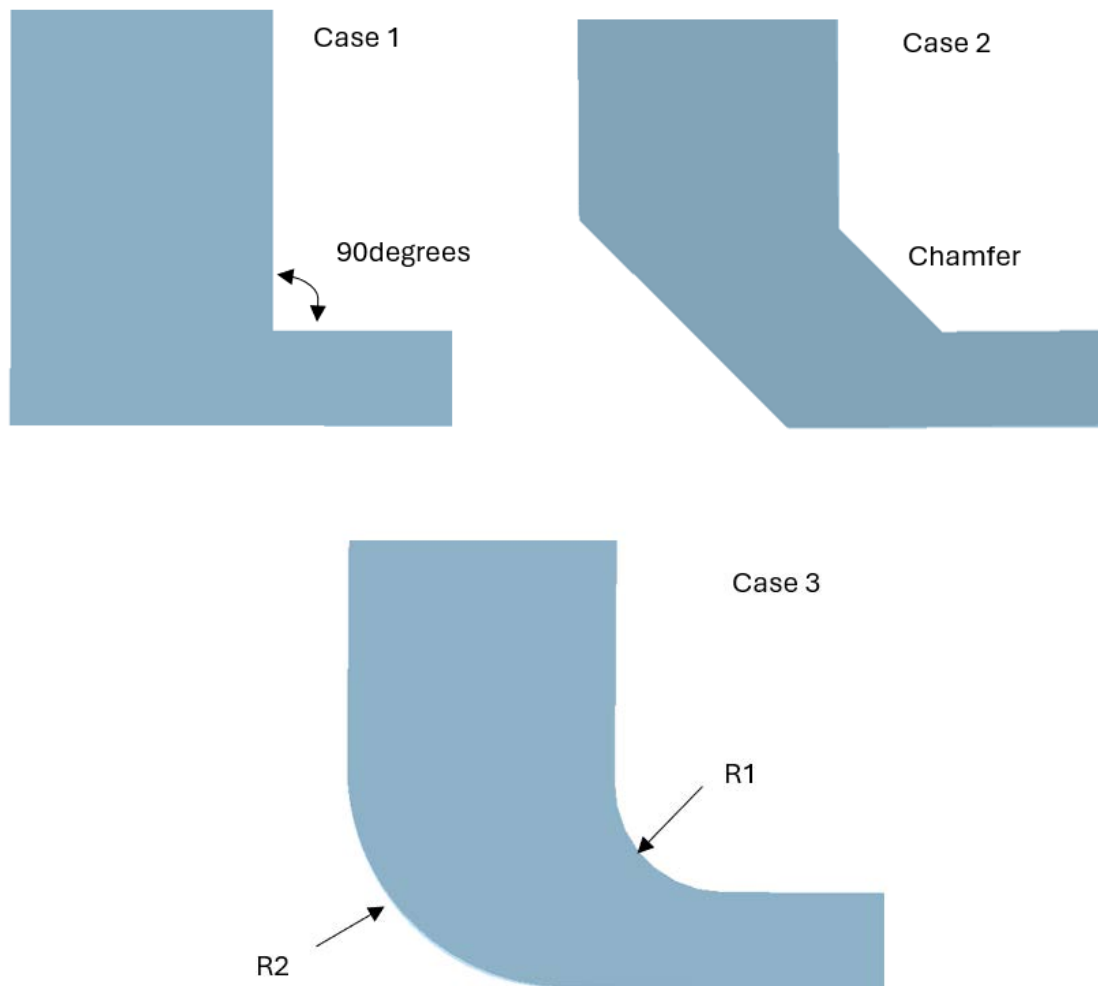


Figure 34: Different connections between the collector and the chimney wall

In the following table, the results of these 3 cases are represented:

Table 20: Results for the different connections

Geometry	Mass flow (kg/s)	Density (kg/m ³)	Pressure (Pa)	Power (W)
90degree (case 1)	25.46	1.07	137	41725.85
Chamfer (case 2, reference)	36.08	1.09	95	43873.28
small curve (case 3)	39.2	1.09	68	31302.46

The results show that case 2 is the best modification while the 90degree angle (case 1) is very close to case 2 and the small curve (case 3) doesn't show good values of Power. The explanation for this is that the turbine that is used is a pressure staged turbine, similar to hydroelectric type enclosed turbines which need high static pressure values to operate. [12] This is also shown in equation 11. Case 3 has better mass flow values m (15-degree geometry) as shown in the following diagram, while it eliminated the turbulence near the entrance as shown in the figure below, but the static pressure is reduced a lot, resulting in huge power loss.

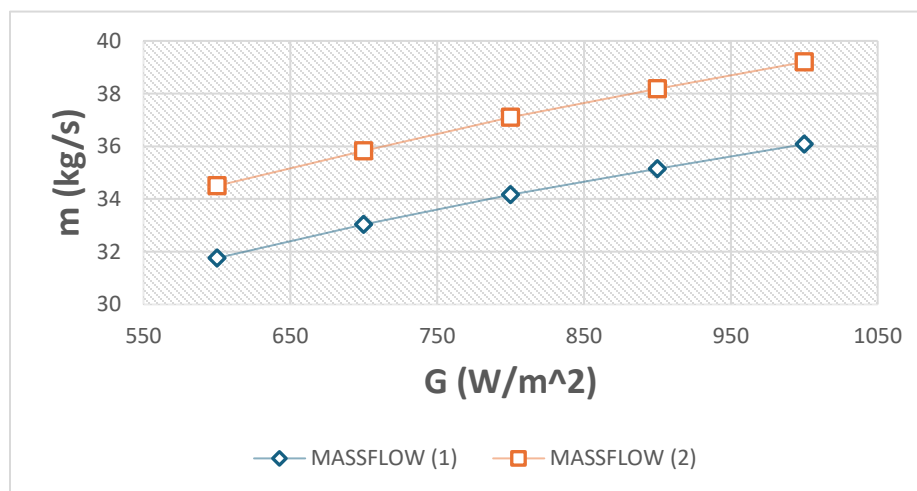


Figure 35: Mass flow at case 2 and 3 for the 15-degree geometry

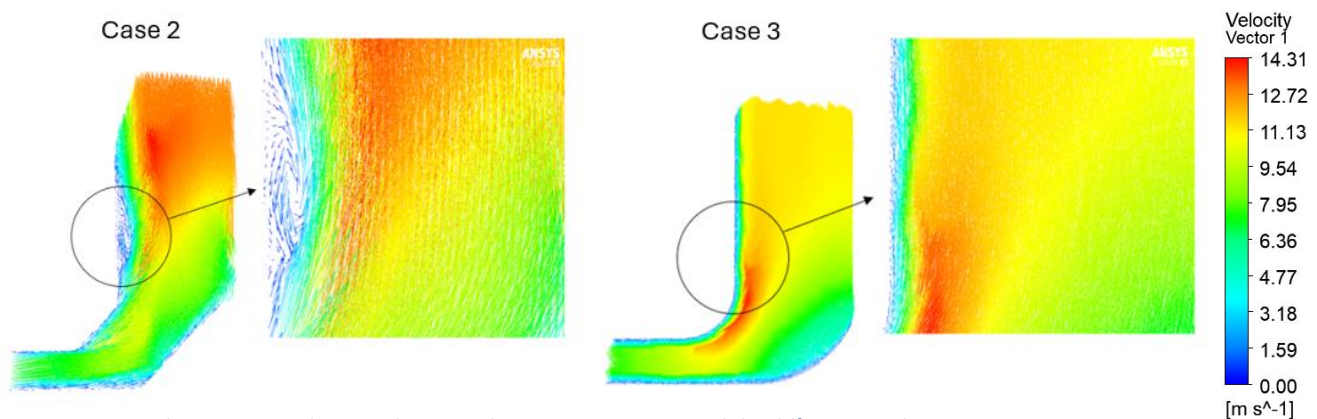


Figure 36: Velocity Vectors showing the recirculation area in case 2 and the difference with case 3

One more case is proposed to try to maintain the higher mass flow while having a decent static pressure value. This will be case 4, which is a geometry similar to case 3 but with a bigger outer curve that can decrease and then increase the area of the chimney to create the pressure difference that is needed. The idea of this geometry design came by analyzing figure 36, where the turbulence is in the outer part of the chimney wall. To eliminate the recirculation area, a curve was designed to follow the flow. The recirculation area needs to be eliminated because it wastes energy. So, with the help of the curved geometry, the mass flow will remain high while this reduction of the area will increase the static pressure difference, unlike case 3 where the pressure difference is not distributed well due to the curve. Case 4 is shown in the next figure:

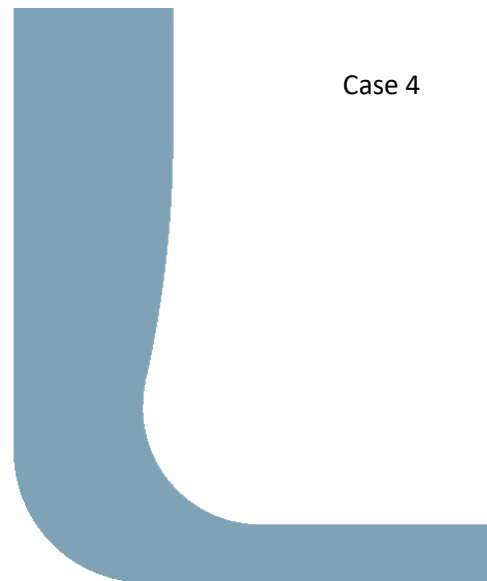


Figure 37: Connection in case 4

Geometry	Mass flow (kg/s)	Density (kg/m ³)	Pressure Difference (Pa)	Power (W)
Chamfer (case 2, reference)	36.08	1.09	95	43873.28
Case 4	39.49	1.09	108	50083.46

The results of the modification on the curved geometry (case 4) are better and maintain a high mass flow rate while the pressure difference, which is shown in the next figure, is even higher than the reference geometry. In conclusion, case 4 is the most preferable with the highest power output.

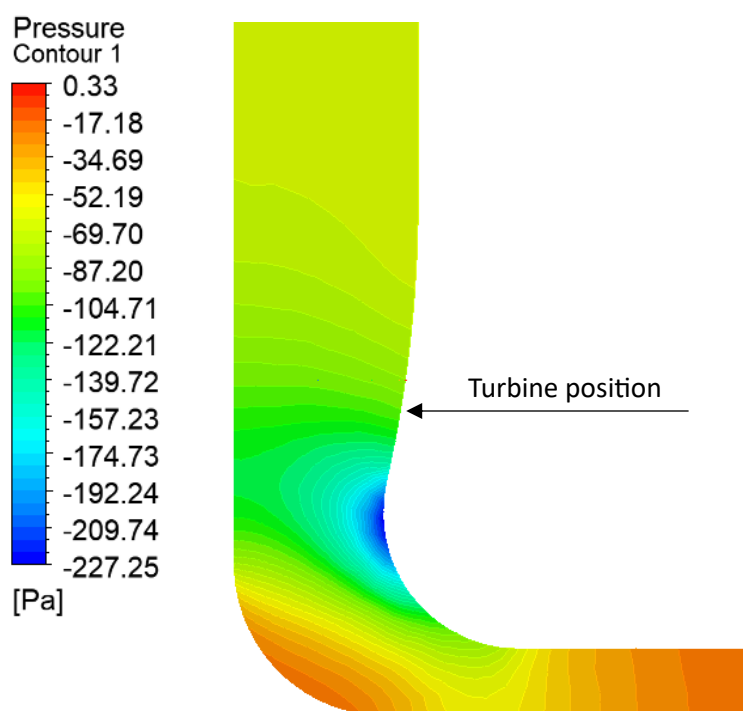


Figure 38:Gauge Pressure contour for case 4

6.2.2 Collector slope:

Another geometry change is the incline of the collector. It is expected that an inclined collector will increase the mass flow while maintaining a high-pressure difference. 4 cases will be tested. The first case (5) is a 1-degree inclined collector. Case 6 is 2 degrees while case 7 and 8 are 3 and 4 degrees respectively. These cases are shown in the following figure:

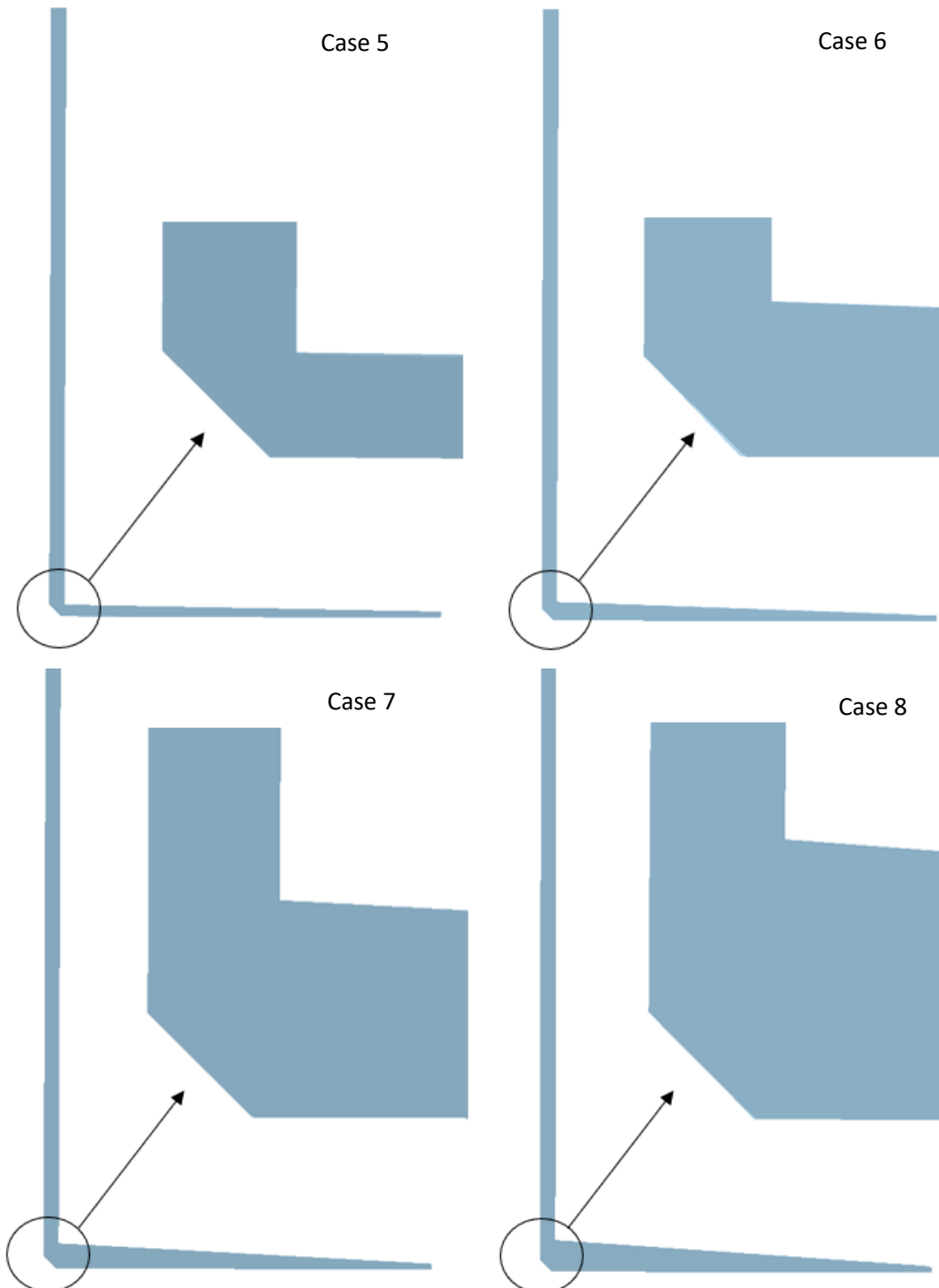


Table 21: Results of the different inclined collectors

Degrees	mass flow (kg/s)	Static pressure (Pa)	density (kg/m ³)	power (W)
0	25.46	137	1.07	41725.8
1 (Case 5)	31.34	133	1.07	49862.8
2 (Case 6)	33.17	127	1.07	50393.6
3 (Case 7)	33.48	116	1.07	46458.9
4 (Case 8)	32.69	101	1.07	39496.8

The results from table 19 show that case 6 has the highest power output at 50kW while as the degrees rise, static pressure and mass flow are reducing a lot.

6.2.3 Chimney walls angle:

One very important modification that can increase the output power a lot is the incline of the outer chimney wall. 3 cases will be simulated. Case 9 has a 1-degree inclined chimney wall while Case 10 and 11 are 2 and 3 degrees inclined respectively. It is expected that static pressure will be much higher with this geometry modification, resulting in higher Power output. The 3 geometries of these cases are represented in the following figures:

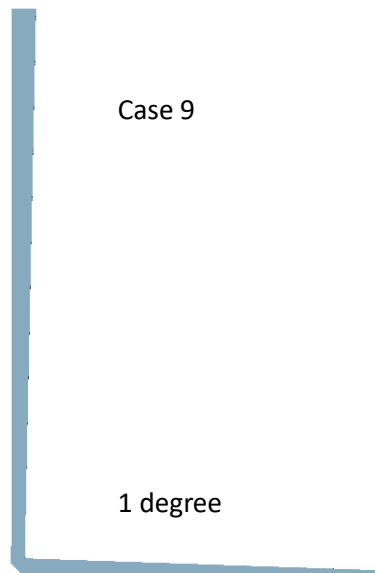


Figure 41: 1 degree chimney wall angle

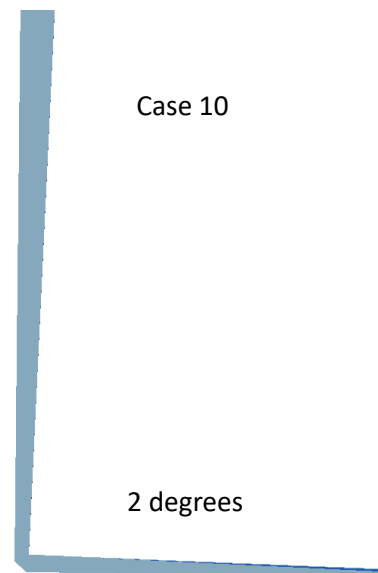


Figure 40: 2 degrees chimney wall angle

A



Figure 42: 3 degrees chimney wall angle

Table 22: Results of the different inclined chimney walls

Degrees	mass flow (kg/s)	Density (kg/m ³)	Pressure (Pa)	Power (W)	Turbine position (m)
0	25.46	1.07	137	41725.8	9m
1 (Case 9)	41.85	1.09	203	99764.3	13m
2 (Case 10)	39.08	1.09	196	89948.5	13m
3 (Case 11)	36.60	1.09	141	60601.1	13m

As expected, all 3 cases show better results with the 0-degree geometry. Case 1, with a 1-degree inclined chimney wall, gives the best results for output power, close to 100kw. It is also seen from table 22 that the turbine positions change while the degrees of the inclined walls rise. This is expected because with the inclined walls, static pressure stabilizes higher up the chimney.

In the following figures, the velocity and pressure contours are shown.

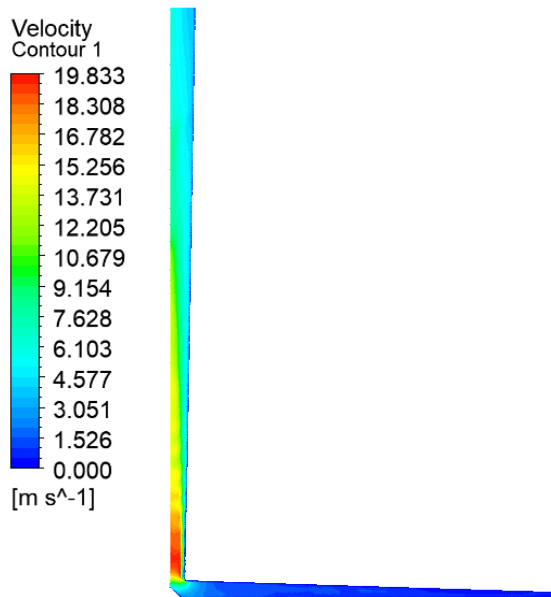


Figure 43: Velocity contour for Case 9

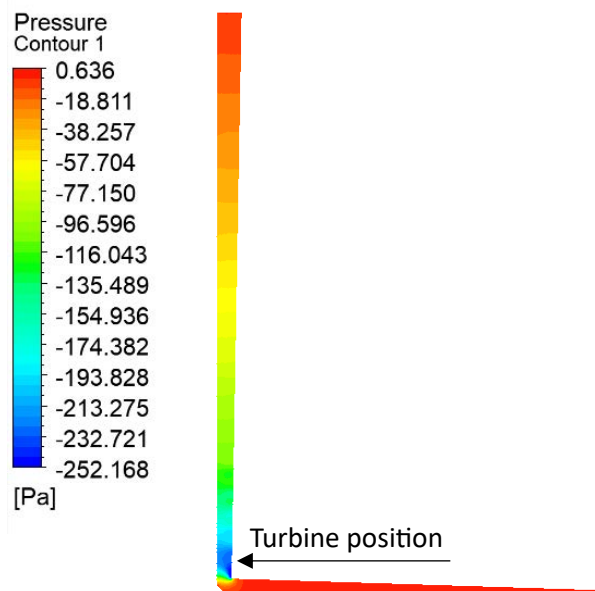


Figure 44: Pressure contour for Case 9

6.2.4 Optimized Geometry:

A simulation with all the geometry modifications merged in one case was performed (Case 12). In the following figures the contours of pressure and velocity are shown.

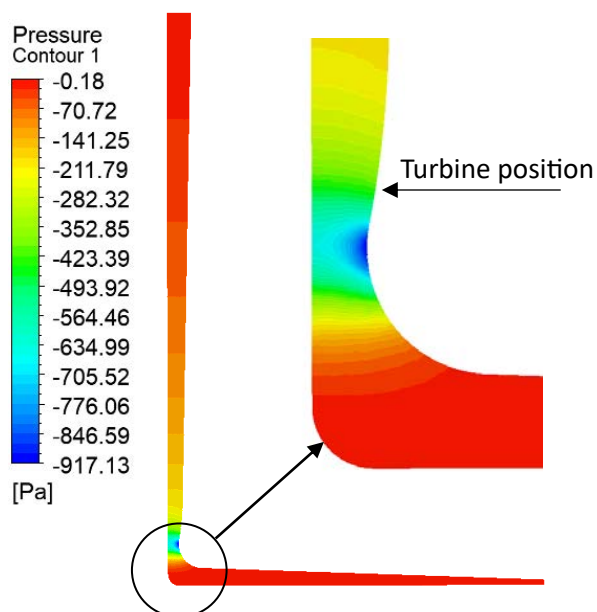


Figure 45: Pressure contour for the optimized geometry

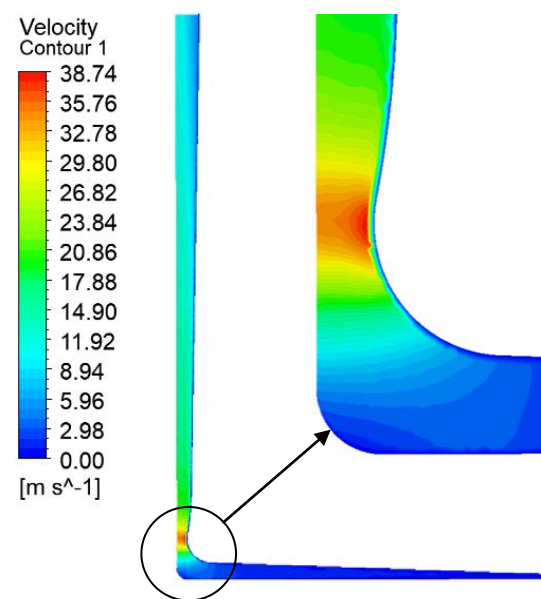


Figure 46: Velocity contour for the optimized geometry

The results are very good. The output power value is very high near 360 kW, while the mass flow increased to 70.6 kg/s and the static pressure difference to 445 Pa. This is probably due to the high acceleration the air has near the base of the chimney as shown in the contours.

Table 23: Results of the optimized geometry

Geometry	mass flow (kg/s)	Density (kg/m ³)	Static pressure (Pa)	Power (W)	turbine position (m)
Merged (Case 12)	70.6	1.12	445	359051.4	17

This geometry proved to be the best, and these combined modifications improved the power output a lot.

6.2.5 Bigger scale plant:

The final geometry modification is the increase of the tower height, the collector radius and the chimney diameter while maintaining the same analogies between those. For this, the constant factors A_1 and A_2 will be used to calculate the collector and chimney radius while increasing the chimney height 20 meters in each case. These factors are shown as follows:

Equation 28

$$A_1 = \frac{\text{Chimney height}}{\text{Collector diameter}} = \frac{194.65}{122} = 1.595$$

Equation 29

$$A_2 = \frac{\text{Chimney height}}{\text{Chimney diameter}} = \frac{194.65}{5.08} = 38.307$$

To ensure that the flow remained turbulent while the geometry is getting bigger, Rayleigh number was calculated. Rayleigh number can be thought of as the ratio of buoyancy forces and the product of thermal diffusivity and momentum diffusivity. [63]

If $1708 < Ra$, then there is no flow, and the heat transfer is created only from heat conduction.

If $1708 < Ra < 3 \times 10^5$, then the buoyancy overcomes the fluid resistance and causes natural convection currents which have the shape of hexagonal cells called Benard cells. [63]

If $Ra > 3 \times 10^5$, then Benard cells collapse, and the flow becomes turbulent. [63]

In this case the lowest Rayleigh number is at the start of the collector where the flow isn't fully developed, and the temperatures of the collector and the ground have low values. The important factor that changes the Ra number is the temperature difference of the ground with the collector as shown in equation 6.

The following table shows the change in the Rayleigh number as the Plant get bigger.

Table 24: Rayleigh number values at the start of the collector

Chimney height (m)	DT (K)	Ra	criterion	flow type
194.65	6.6	1037884372	$>3 \times 10^5$	turbulent
214.65	7.6	1195139580	$>3 \times 10^5$	turbulent
234.65	9.1	1431022392	$>3 \times 10^5$	turbulent
254.65	10.7	1682630725	$>3 \times 10^5$	turbulent

As expected, Rayleigh number increases and the flow remains turbulent along the collector.

The following table shows the results of the bigger geometries, while the figure is a diagram showing the % growth of the power:

Table 25: Results for bigger scale plant

Chimney height (m)	mass flow (kg/s)	density (kg/m ³)	Static Pressure (Pa)	Power (W)	Turbine position (m)
194.65	41.9	1.09	203	99764	13
214.65	55.7	1.08	240	158435	13
234.65	61.7	1.08	270	197440	13
254.65	68.7	1.07	330	271204	14
274.65	75.2	1.07	384	345442	14

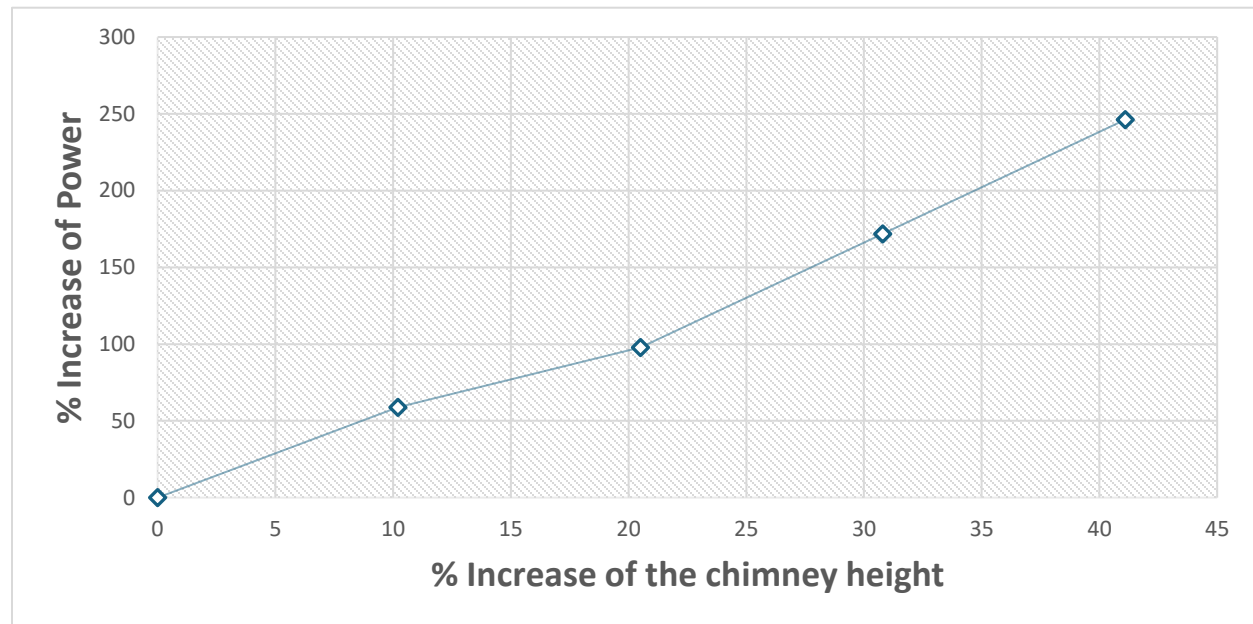


Figure 47: Increase of Power for the bigger geometries

In conclusion, the results show that bigger scaled plants have potential. The geometry with a 274.65 meters chimney height produces 345 kw of electricity while its mass flow and its static pressure have much higher values from the reference geometry. These results show that SUT technology has the capability to increase its practicality and power output.

6.2.6 Bigger scale plant with optimized geometry:

The same steps would be applied in the optimized geometry, with the only difference that plants with smaller heights will be tested to see how much power can produce. This is important because when lowering the chimney height, then the upfront costs of the structure will be less. In the following table the results are shown.

Table 26: Different results for different scales of the optimized geometry

Chimney height (m)	mass flow (kg/s)	density (kg/m ³)	Static Pressure (Pa)	Power (W)	Turbine position (m)
174.65	63.7	1.12	397	289016	17
184.65	66.7	1.12	424	323209	17
194.65	70.6	1.12	445	359051	17
204.65	78.4	1.12	455	407680	17
214.65	87.6	1.12	461	461526	17

It is noted that, in the optimized geometry, the mass flow has a very high increase when the chimney height reaches 214.65 meters. This mass flow change creates a greater power output close to 460 kW. Also, with this modification, a 20 meters smaller chimney can produce much more power output than the reference geometry where the power output was 43 kW. The % change of power generation due to height increase is shown below:

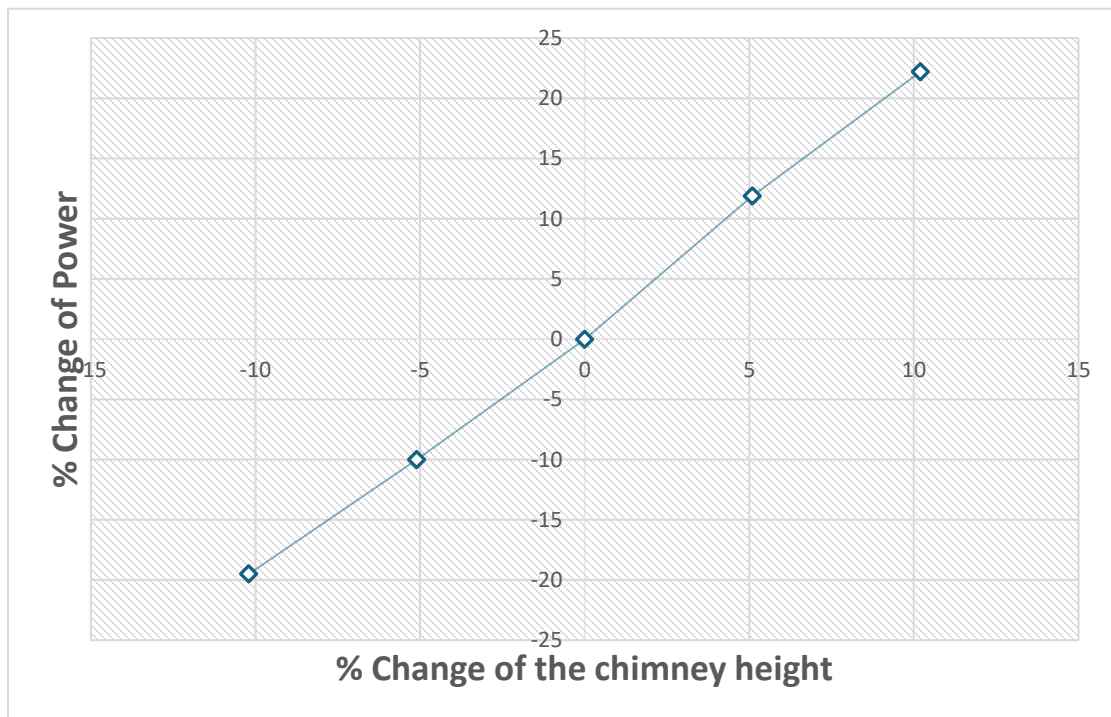


Figure 48: % change of Power in optimized bigger geometry

Chapter 7. Conclusions:

A step-to-step CFD study was performed in this thesis, while different results occurred from the changes in the reference geometry, the different turbulent models and solar discretization parameters. To summarize, to solve a CFD case, especially for a SUT plant type geometry, it's important to design the geometry exactly like the reference study that it will validate it. The next step is to create an acceptable mesh with reliable mesh metrics, which would have enough cells to avoid extra computational cost and have acceptable results, close to other numerical and experimental studies. Also, very important is the understanding and implying of the boundary conditions on the surfaces of the geometry and the environment and fluid parameters. Before running the simulation, the correct models of turbulence must be used. This study proved the best models are the RNG and the Spalart Allmaras models, which showed a maximum velocity of 14.3 m/s. Also, the tuned GEKO model and SST were promising. In addition, for the solar load model, the preferable discretization parameters were valued 2 for theta and phi divisions and 1 for theta and phi pixels because the increase of these parameters didn't show better results but worsened the case by increasing the computational cost and thus, simulation time. The better geometry modifications were the alternative curve connection between the collector and the chimney (case 4) which helped to produce near to 50kW of power, the 2-degrees inclined collector which also produced 50 kW of power (case 2) and at last the 1-degree inclined chimney wall (case 1) which increased the power to 100 kW. The best overall results were shown from the optimized geometry which included all the above and it is in Case 12 where 360 kW of power could be potentially produced. Finally, the bigger scaled geometry (expanding case 9) showed a power output near 270 kW and the bigger scaled optimized geometry with a chimney height at 214 meters showed a power output at 460 kW. These results showed that SUT technology has potential to increase its power but even then, it cannot be compared with the PV and CSP systems because of their higher energy generation intensity (EGI) and better efficiency. For example, a PV system that can produce 360kW of power needs approximately 1700 m² of land, while the area of the Manzanares plant is 46000 m². The best way SUT technology can be useful is when it is combined with agricultural uses (assuming that the plants must not be big and interrupt the flow), taking advantage of the greenhouse effect the semi-transparent collector creates, or in harsh environments like deserts, when the water is limited thus, systems like PV's and CSP plants which use it as a cooling fluid cannot operate. Another way to make SUT technology practical is if it is combined with PVs or other technologies to support the power generation of those. These concepts can be studied in the future.

References

- [1] P. Das, "A review on solar updraft tower plant technology: Thermodynamic analysis, worldwide status, recent advances, major challenges and opportunities," *Sustainable Energy Technologies and Assessments*, vol. 52, 2022.
- [2] S. R. Department, "Share of fossil fuels in primary energy consumption worldwide from 1965 to 2023," Statista Research Department, 2024.
- [3] M. J. Saleh, "Enhancement methods of the performance of a solar chimney power plant: Review," *Results in Engineering*, 2023.
- [4] Z. N. Ndalloka, "Solar photovoltaic recycling strategies," *Solar Energy*, 2024.
- [5] D. Cao, "The Working Principle of Solar Panels," SolarCtrl, 6 March 2024. [Online]. Available: <https://www.solarctrl.com/blog/the-working-principle-of-solar-panels/>.
- [6] G. Masson, "Snapshot of Global PV Markets 2024," IEA Photovoltaic Power System Programme, 2024.
- [7] E. Graham, "Solar power continues to surge in 2024," Ember, 2024.
- [8] S. Suman, "Performance enhancement of solar collectors—A review," *Renewable and Sustainable Energy Reviews*, 2015.
- [9] E. Bellos, "Progress in beam-down solar concentrating systems," *Progress in Energy and Combustion Science*, 2023.
- [10] A. H. Alami, "Concentrating solar power (CSP) technologies: Status and analysis," *International Journal of Thermofluids*, 2023.
- [11] J. Lappalainen, "Modelling a molten salt thermal energy system – A validation study," *Applied Energy*, 2019.
- [12] J. Schlaich, "Design of Commercial Solar Updraft Tower Systems—Utilization of Solar Induced Convective Flows for Power Generation," *Journal of Solar Energy Engineering*, 2005.
- [13] R. Balijepalli, "Optimized design and performance parameters for wind turbine blades of a solar updraft tower (SUT) plant using theories of Schmitz and aerodynamics forces," *Sustainable Energy Technologies and Assessments*, vol. Volume 30, 2018.
- [14] O. A. Marzouk, "Energy Generation Intensity (EGI) of Solar Updraft Tower (SUT) Power Plants Relative to CSP Plants and PV Power Plants Using the NewEnergy Simulator “Aladdin”," *Energies*, 2024.

- [15] s. b. partner, "Solar Updraft Tower Pilot Plant Manzanares," sbpsonne, 2024.
- [16] H. H. Al-Kayiem, "Historic and recent progress in solar chimney power plant enhancing technologies," *Renewable and Sustainable Energy Reviews*, vol. Volume 58, 2016.
- [17] H. Rahimi, "2D Numerical Investigation of the Laminar and Turbulent Flow Over Different Airfoils Using OpenFOAM 2D Numerical Investigation of the Laminar and Turbulent Flow Over Different Airfoils Using OpenFOAM," *Journal of Physics Conference Series*, vol. 555, 2014.
- [18] A. Kasaeian, "A review on solar chimney systems," *Renewable and Sustainable Energy Reviews*, vol. 67, pp. 954-987, 2017.
- [19] X. Zhou, "Experimental study of temperature field in a solar chimney power setup," *Applied Thermal Engineering*, vol. Volume 27, no. 11–12, pp. 2044-2050, 2007.
- [20] A. Kasaeian, "Experimental investigation of climatic effects on the efficiency of a solar chimney pilot power plant," *Renewable and Sustainable Energy Reviews*, vol. 15, no. 9, pp. 5202-5206, 2011.
- [21] C. B. Maia, "Analysis of the Airflow in a Prototype of a Solar Chimney Dryer," *Heat Transfer Engineering*, vol. 30, no. 2009, pp. 393-399, 2010.
- [22] A. R. Shahreza, "Experimental and numerical investigation on an innovative solar chimney," *Energy Conversion and Management*, vol. 95, pp. 446-452, 2015.
- [23] S. Okada, "Improvement in Solar Chimney Power Generation by Using a Diffuser Tower," *J. Sol. Energy Eng.*, vol. 137, no. 3, 2015.
- [24] S. P.M., "Performance analysis of solar chimney using mathematical and experimental approaches," *International Journal of Energy Research*, 2018.
- [25] H. H. Al-Kayiem, "Experimental and numerical analysis of the influence of inlet configuration on the performance of a roof top solar chimney," *Energy and Buildings*, vol. 159, pp. 89-98, 2018.
- [26] N. Mehla, "Thermal analysis of solar updraft tower by using different absorbers with convergent chimney," *Environment, Development and Sustainability*, vol. 21, p. 1251–1269, 2018.
- [27] R. Balijepalli, "Development of a small scale plant for a solar chimney power plant (SCPP): A detailed fabrication procedure, experiments and performance parameters evaluation," *Renewable Energy*, vol. 148, pp. 247-260, 2020.
- [28] P. Belkhode, "Evaluation of the experimental data to determine the performance of a solar chimney power plant," *Materials Today: Proceeding*, vol. 27, pp. 102-106, 2020.
- [29] P. Das, "Experimental studies of a laboratory scale inclined collector solar updraft tower plant with thermal energy storage system," *Journal of Building Engineering*, vol. 41, 2021.

- [30] N. Pasumarthi, "Experimental and theoretical performance of a demonstration solar chimney model—Part I: mathematical model development," *International Journal of Energy Research*, 1998.
- [31] A. Akbarzadeh, "Examining potential benefits of combining a chimney with a salinity gradient solar pond for production of power in salt affected areas," *Solar Energy*, vol. 83, no. 8, pp. 1345-1359, 2009.
- [32] N. Mehla, "EXPERIMENTAL ANALYSIS OF A VELOCITY FIELD USING VARIABLE CHIMNEY DIAMETER FOR SOLAR UPDRAFT TOWER," *International Journal of Engineering Science and Technology*, vol. 3, pp. 3167-3171, 2011.
- [33] M. A. Al-Dabbas, "The first pilot demonstration: solar updraft tower power plant in Jordan," *International Journal of Sustainable Energy*, vol. 31, no. 6, pp. 399-410, 2012.
- [34] S. M. Raney, "Experimental Validation of Solar Chimney Performance Models and Operational Characteristics for Small Scale Remote Applications," in *International Conference on Energy Sustainability*, San Diego, California, USA, 2012.
- [35] S. Kalash, "Experimental investigation of the solar collector temperature field of a sloped solar updraft power plant prototype," *Solar Energy*, vol. 98, pp. 70-77, 2013.
- [36] M. Ghalamchi, "An experimental study on the thermal performance of a solar chimney with different dimensional parameters," *Renewable Energy*, vol. 91, pp. 477-483, 2016.
- [37] S. Lal, "Experimental investigation and CFD simulation studies of a laboratory scale solar chimney for power generation," *Sustainable Energy Technologies and Assessments*, vol. 13, pp. 13-22, 2016.
- [38] M. R. Ahmed, "Computational and experimental studies on solar chimney power plants for power generation in Pacific Island countries," *Energy Conversion and Management*, vol. 149, pp. 61-78, 2017.
- [39] Ansys®, "16.3.11. Solar Load Model," in *Fluent User's Guide*, ANSYS, 2024.
- [40] E. Cuce, "A thorough performance assessment of solar chimney power plants: Case," *Cleaner Engineering and Technology*, vol. 1, 2020.
- [41] M. Kaplan, "Influence of Inclination Angle at the Chimney Inlet on the Power Generation in Solar Chimney Power Plants through 3D CFD Model," *International Journal of Photoenergy*, 2023.
- [42] E. Gholamalizadeh, "Analysis of fluid flow and heat transfer on a solar updraft tower power plant coupled with a wind turbine using computational fluid dynamics," *Applied Thermal Engineering*, vol. 126, pp. 548-558, 2017.
- [43] Ansys®, "5.3.6. Discrete Ordinates (DO) Radiation Model Theory," in *Fluent Theory Guide*, ANSYS, 2024.
- [44] Ansys®, "16.3.5.1. Angular Discretization," in *Fluent User's Guide*, ANSYS, 2024.

- [45] Ozgün, "mechead.com," 2023. [Online]. Available: <https://www.mechead.com/mesh-quality-checking-ansys-workbench/>.
- [46] Ansys®, "15.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models," in *Fluent User's Guide*, ANSYS, 2024.
- [47] Ansys®, "15.2.1.2. k- ϵ Models," in *Fluent User's Guide*, ANSYS, 2024.
- [48] Ansys®, "15.2.1.3. k- ω Models," in *Fluent User's Guide*, ANSYS, 2024.
- [49] Ansys®, "4.4.1. Standard k- ω Model," in *Fluent Theory Guide*, ANSYS, 2024.
- [50] Ansys®, "4.5. Generalized k- ω (GEKO) Model," in *Fluent Theory Guide*, ANSYS, 2024.
- [51] F. Menter, Best Practice: Generalized k- ω (GEKO) Two-Equation Turbulence Modeling in Ansys CFD, ANSYS, 2019.
- [52] Ansys®, "4.4.2. Baseline (BSL) k- ω Model," in *Fluent Theory Guide*, ANSYS, 2024.
- [53] Ansys®, "4.4.3. Shear-Stress Transport (SST) k- ω Model," in *Fluent Theory Guide*, ANSYS, 2024.
- [54] A. Mebarki, "CFD analysis of solar chimney power plant: Finding a relationship between model minimization and its performance for use in urban areas," *Energy Reports*, vol. 8, pp. 500-513, 2022.
- [55] B. Djaouida, "Controlling power output of solar chimney power plant according to demand," *INTERNATIONAL JOURNAL OF AMBIENT ENERGY*, vol. 41, p. 1467–1481, 2020.
- [56] Ansys®, "4.3.1. Standard k- ϵ Model," in *Fluent Theory Guide*, ANSYS, 2024.
- [57] Ansys®, "4.3.2. RNG k- ϵ Model," in *Fluent Theory Guide*, ANSYS, 2024.
- [58] Ansys®, "4.3.3. Realizable k- ϵ Model," in *Fluent Theory Guide*, ANSYS, 2024.
- [59] W. HAAF, "Solar Chimneys Part II: Preliminary Test Results from the Manzanares Pilot Plant," *International Journal of Sustainable Energy*, vol. 2, pp. 141-161, 1984.
- [60] M. Tingzhen, "Analytical and numerical investigation of the solar chimney power plant systems," *INTERNATIONAL JOURNAL OF ENERGY RESEARCH*, vol. 30, p. 861–873, 2006.
- [61] Ansys®, "4.2. Spalart-Allmaras Model," in *Fluent Theory Guide*, ANSYS, 2024.
- [62] Ansys®, "4.7. Transition SST Model," in *Fluent Theory Guide*, ANSYS., 2024.
- [63] Y. A. ÇENGEL, "9–5 . NATURAL CONVECTION INSIDE ENCLOSURES," in *HEAT AND MASS TRANSFER FIFTH EDITION*, New York, McGraw-Hill Education, 2015.

- [64] H. Rahimi, "2D Numerical Investigation of the Laminar and Turbulent Flow Over Different Airfoils Using OpenFOAM 2D Numerical Investigation of the Laminar and Turbulent Flow Over Different Airfoils Using OpenFOAM," in *Journal of Physics Conference Series* 555(1), 2014.