



RESEARCH ARTICLE

CFD MODELING OF A RADIATOR AXIAL FAN FOR AIR FLOW DISTRIBUTION BY COMPARING THE FOUR DIFFERENT DESIGNS FOR THE FAN

Mohamed A. Elhaj^a, Osama D.H Abdulrazig^b, Monaem Elmnifi^{*c}

^a Department of Mechanical Engineering Faculty of Engineering-Misurata University, Libya.

^b Department of Mechanical Engineering Faculty of Engineering-Bright Star University, Misurata University, Libya.

^c Department of Mechanical Engineering Technology, Belgorod Technological University named after. V.G. Shukhov, Belgorod, Russia, Department of Mechanical Engineering Faculty of Engineering-Bright Star University, Libya.

*Corresponding Author E-mail: monm.hamad@yahoo.co.uk

This is an open access journal distributed under the Creative Commons Attribution License CC BY 4.0, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

ARTICLE DETAILS

Article History:

Received 23 May 2024
Revised 15 June 2024
Accepted 30 July 2024
Available online 01 August 2024

ABSTRACT

The fluid mechanics principle is used extensively in designing axial flow fans and their associated equipment. This report presents a computational fluid dynamics (CFD) modelling of airflow distribution from a radiator axial flow fan CFD Modelling of a Radiator Axial Fan for Air Flow Distribution by Comparing between the four different designs for fan. The work will be considered in the field of flow across the axial fan, especially in the front part where they will be change the shape and design of the fan and the extension of the axis of the fan as well as to explain how the reaction of the flow field will be, when the shape of the front fan and along the axis will be extended by using the four different designs of the fan, which is the same number of blades and the same diameter, It also will change the speed RPM also change the temperature radiator three times for each of the four fans in order to determine whether there was any impact on the regularity of the field of flow, velocity, temperature and pressure. 3D models were created to investigate the use of solid and software work the Phoenix software.

KEYWORDS

axial flow, (CFD) modeling, temperature radiator.

1. INTRODUCTION

Computational Fluid Dynamics is a great tool to calculate the whole flow field in a mixing vessel. In the past, it would take highly skilled Design Engineers with both expert knowledge and experience many man-hours to understand the complexities of fluid flow behavior surrounding a design. Recently, a technology is available designed purely for the engineering, aimed at helping engineers to perform these tasks in a '3 dimensional virtual environment, saving many hours of physical testing in facilities such as costly wind tunnels (Duvenhage et al., 1996; Yu and Chan, 2007). Impeller fans are designed in such a way to improve their efficiency but this inherently introduces some aerodynamics challenges due to backward facing of their blades; the backward inclination of the blades relative to the direction of rotation creates a negative angle of attack between the leading edge of the aero foil and the impinging air thereby reducing the aerodynamic efficiency of the impeller. Secondly, the natural aerodynamic forces cause the airstream across the inward facing or lower pressure side of the blade to separate from the blade prior to reaching the trailing edge, thereby causing turbulence and undesirable fan noise (Twizell and Bright, 1981).

THE axial flow fan is extensively used in many engineering applications. Its adaptability has resulted in implementation into large scale systems, from industrial dryers and air conditioning units to automotive engine cooling and in-cabin air recirculation systems. The benefit of using axial flow fans for the purpose of augmenting heat transfer is particularly evident in the automobile industry because of the need for relatively

compact designs. (Twizell and Bright, 1981; Bredell et al., 2005; Stafford et al., 2010). This course work will be tailored to addressing some of the impeller design parameters for automobile radiator fan like hub shapes, length, velocity, temperature and pressure and impeller operating speed that may affect impeller aerodynamic performance. From findings, air is generally known to turbulence at low velocity; with proper hub design by Comparing between the four different designs for fan.

The work will be considered in the field of flow across the axial fan, especially in the front part where they will be change the shape and design of the fan and the extension of the axis of the fan as well as to explain how the reaction of the flow field will be, when the shape of the front fan and along the axis will be extended by using the four different designs of the fan, which is the same number of blades and the same diameter, It also will change the speed RPM also change the temperature radiator three times (50°C, 70°C, 90°C) for each of the four fans in order to determine whether there was any impact on the regularity of the field of flow, velocity, temperature and pressure and Extraction with pictures of these cases and draw curves and diagrams that are given by (Phoenix software) Which will be extension at the end of this study (Stafford et al., 2010; Ho et al., 2009; Stokse, 2004; Kreith, 1999). The adverse effects can be reduced through proper design of impeller parameters and specification of optimum operating rotational speed; the need to reduce or eliminate these adverse effects has informed research work along this area in recent time (Shojaeeford et al., 2011; Wang, 2004; Elmnifi et al., 2023; Mousa Al-Musawi et al., 2024).

Quick Response Code



Access this article online

Website:
www.actamechanicamalaysia.com

DOI:
10.26480/amm.02.2024.109.138

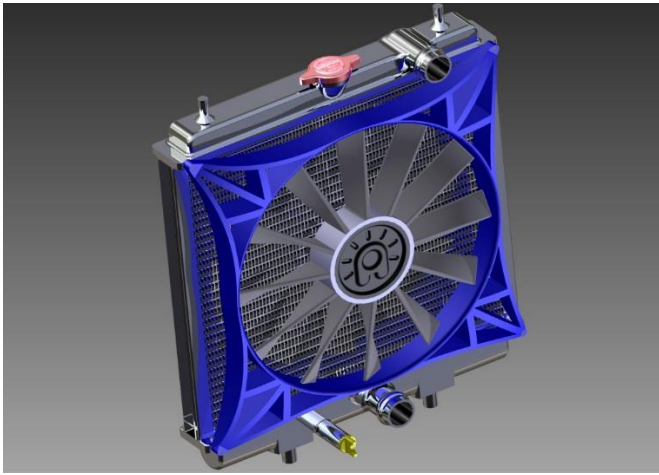


Figure 1: Fan and radiator of an automobile

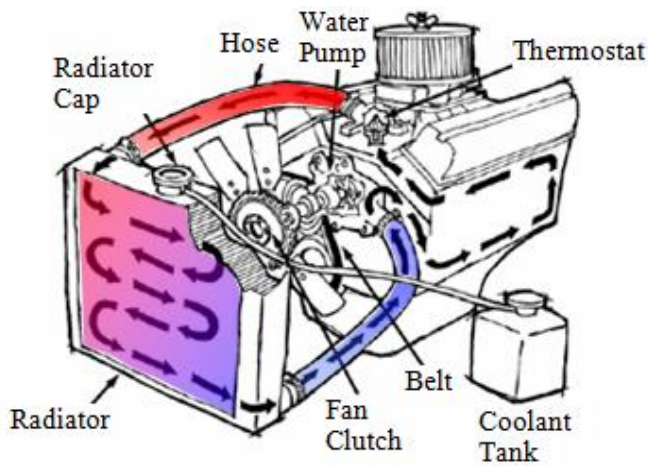


Figure 2: A schematic of a cooling system

2. MODELLING OF PHYSICAL PHENOMENA

The fan provides airflow through the radiator. The orientation of the blade also plays an important role in understanding the flow of air across the radiator and fan. Figures 3 and 4 show a fan and its associated orientation configuration. For a right oriented blade, the direction of fan rotation is clockwise; for a left oriented blade, the direction of fan rotation is counter clockwise (Ho et al., 2009; Stokse, 2004; Elmnifi et al., 2024; Majdi et al., 2022).

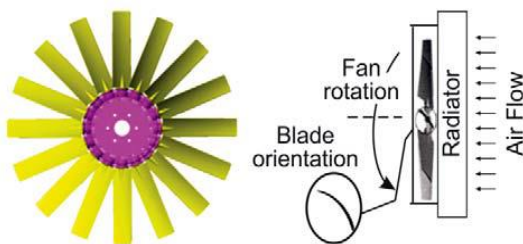


Figure 3: Right oriented blade

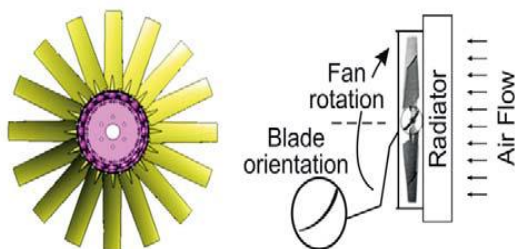


Figure 4: Left oriented blade

3. NUMERICAL CFD MODEL AND PROCEDURE

The first step is to identify a typical radiator axial flow fan that can be reproduced as a 3-D CAD Solid works® software engineering drawing package (Figure 5) (Elmnifi et al., 2023; Majdi et al., 2023; Jenkins et al., 2020; Shams et al., 2023).

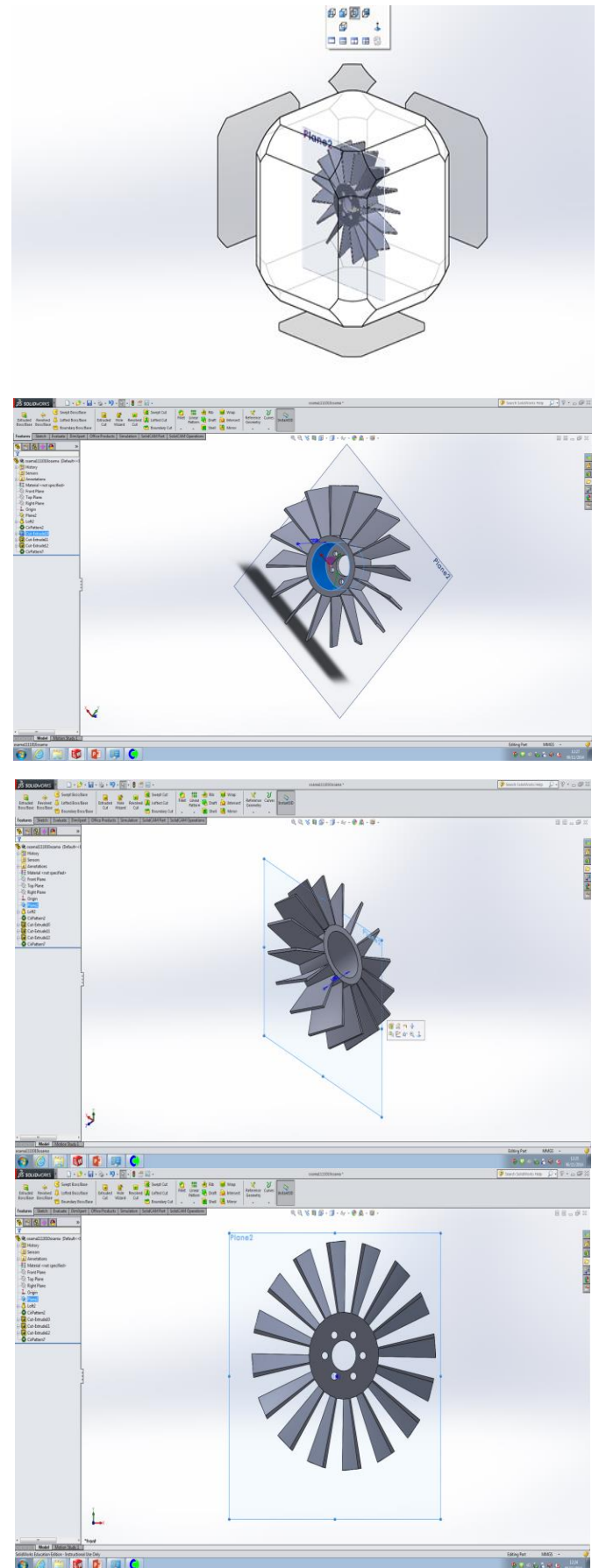


Figure 5: Computational domain of a fan (Solid works® software)

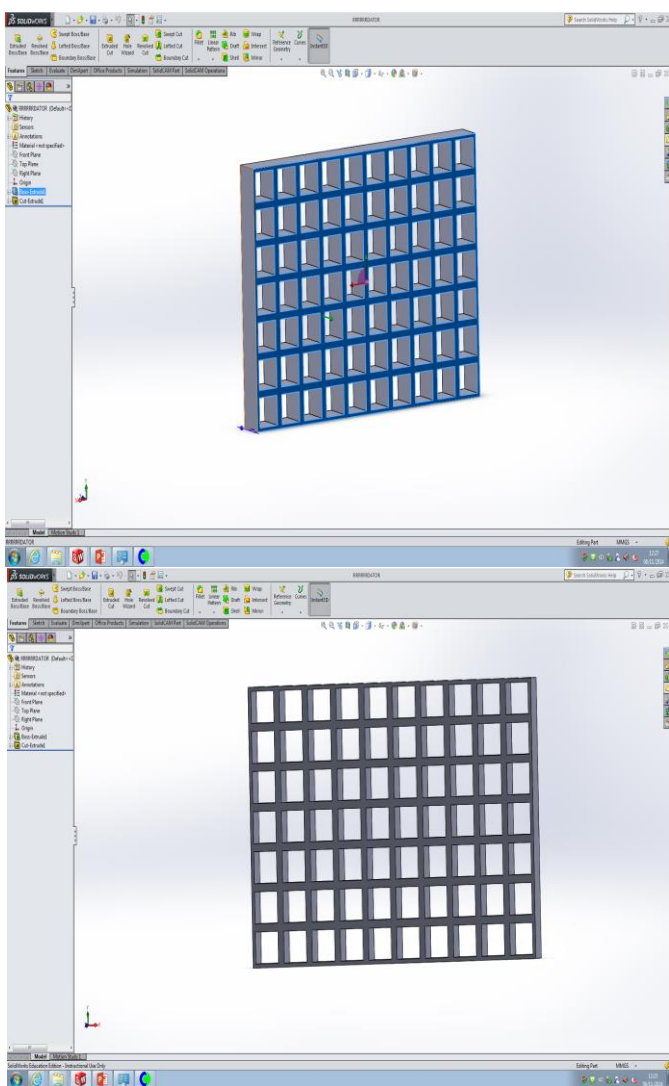


Figure 6: Computational domain of a radiator (Solid works® software)

The 3-D models are then imported into the CFD software, remodeled into different sections, and refined to generate a finite volume meshing (Figure 8 and 9). This is a crucial step, where details of the geometrical shape need to be defined precisely.

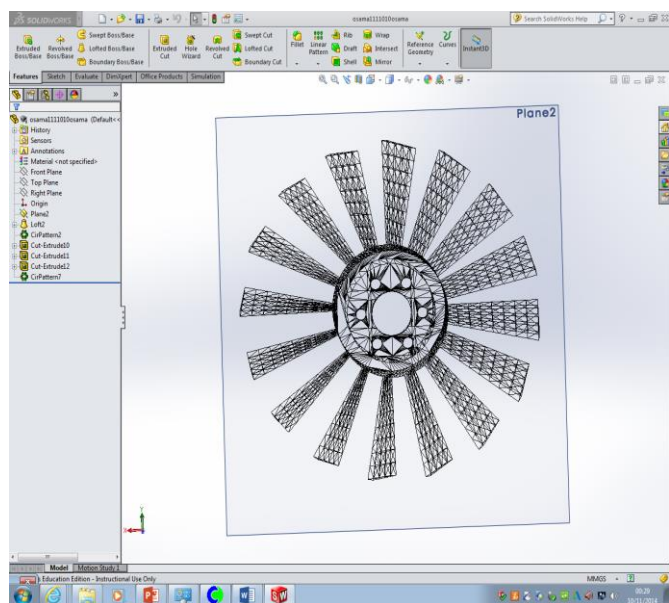


Figure 7: Computational domain of a fan (CTL)

The flow domain is also created, and the final meshing of all components needs to be accurate. Any errors in the drawings and flow area need to be corrected before continuing.

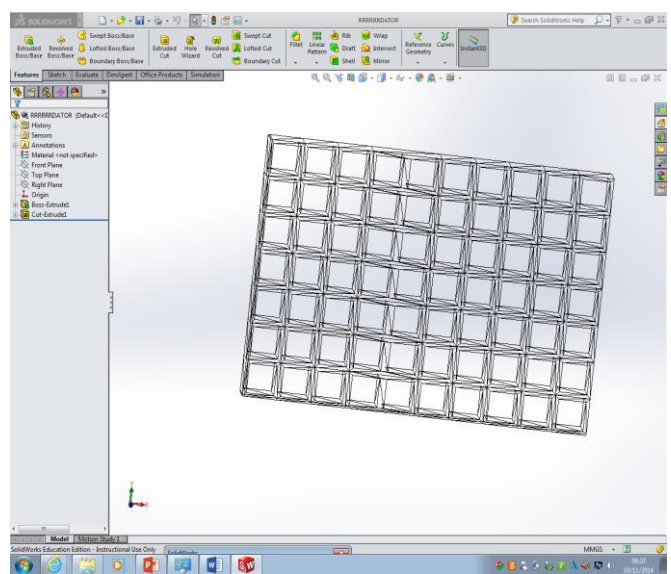


Figure 8: Computational domain of a radiator (CTL)

The flow domain is also created, and the final meshing of all components needs to be accurate. Any errors in the drawings and flow area need to be corrected before continuing.

Harvest vibration or motion energy from humans walking, machinery vibrating, or cars moving on a roadway is an area of great interest, because this energy is otherwise untapped, and since movement is everywhere, the ability to capture this energy cheaply would be a significant advancement toward greater efficiency and cleaner energy production. Consequently, a speed bump as a system generating and harvesting the energy has been designed in this project by using Piezoelectric Smart materials, as well as many materials experimented, load states and formats, then testing feasibility of this application. For exploiting the wasted mechanical energy, for instance; in lighting.

3. METHODOLOGY

Estimating External Force: In American Classification of Trucks according to "GVWR" (Twizell and Bright, 1981). In addition, assuming that the road is designated for the transit the Light and Medium duty trucks, to calculate the Arithmetic Mean of the mass:

$$\text{Mean of mass: } \frac{6000+10000+14000+16000+19500+26000}{6} = 15250 \text{ Pounds.}$$

External force: $67789.379 = 70$ Kilo Newton approximately. This is affecting a speed bump, specifically on the Moving substrate, above springs.

The flow will be treated as steady, incompressible and turbulence in three dimensional. The dimension will be expressed in Cartesian co-ordinate (x, y and z) and the axis of rotation will be the X-axis. The experiment was carried out using PHOENICS, domain was chosen and number of cells manipulated until visibility of flow pattern around the impeller was established (Shojaeeford et al., 2011; Wang, 2004). The experiment is set up in such a way that the flow pattern and certain flow parameters of interest like velocity and pressure changes can be studied. Hence, a control experiment will be carried out to ascertain the flow pattern, velocity and pressure profile of the original impeller and later compare with the profiles obtain when impeller of different hub shapes and shaft dimensions are used.

The parameters of interest like velocity and pressure are then taken along the domain of flow in the x, y, and z-axes and then compare to see if there will be any changes to the flow pattern in term of flow, velocity and pressure profile (Twizell and Bright, 1981; Bredell et al., 2005). Throughout this experimental study, the same dimension and impeller (with the same physical properties-like radius, thickness, material, and surface features) will be used; the only variation will be to the shape and length of shaft for the hub (See Figure 9, 10, 11, and 12). The location of the impeller fan in the domain geometry will also be maintained constant throughout the experiment to eliminate error that might aroused from the effect of change in proximity of the impeller to source of mass from the inlet, though this will be negligible since only one inlet and outlet are specified for the domain and air which is the medium use in this experiment is a continuum (Stafford et al., 2010; Ho et al., 2009).

The domain is setup to replicate scenario of flow across the impeller fan of a radiator of an automobile.

The other parameters that will remain constant throughout this study are:

- Domain Size;
- Inlet and Outlet Size and location;
- Mesh Setting and Relaxation Parameters;
- Number of iterative steps, that is, number of sweep.

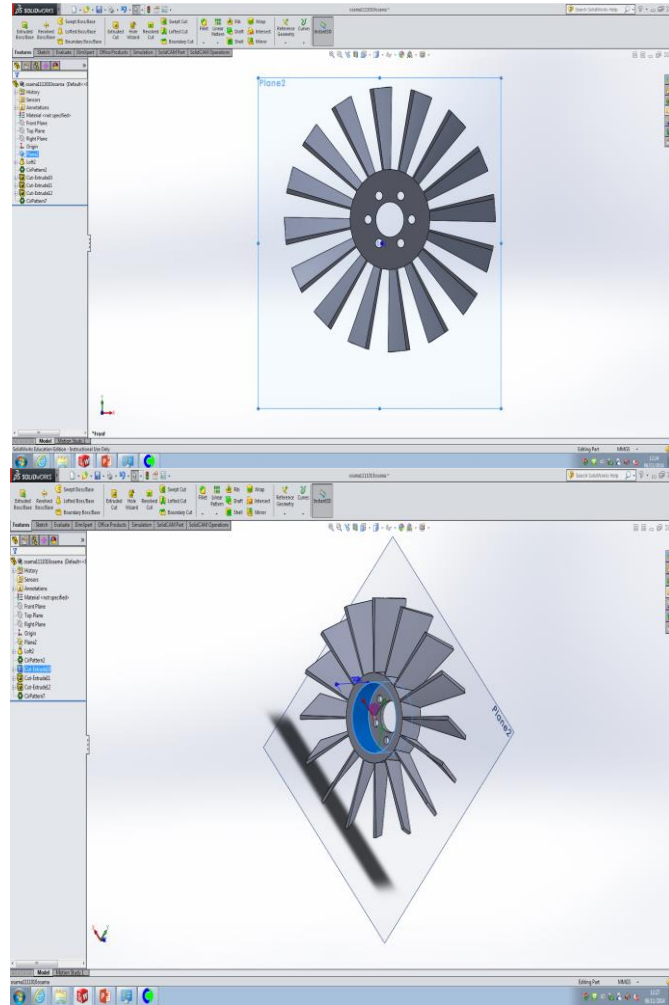


Figure 9: The first design fan under the name (00-00)

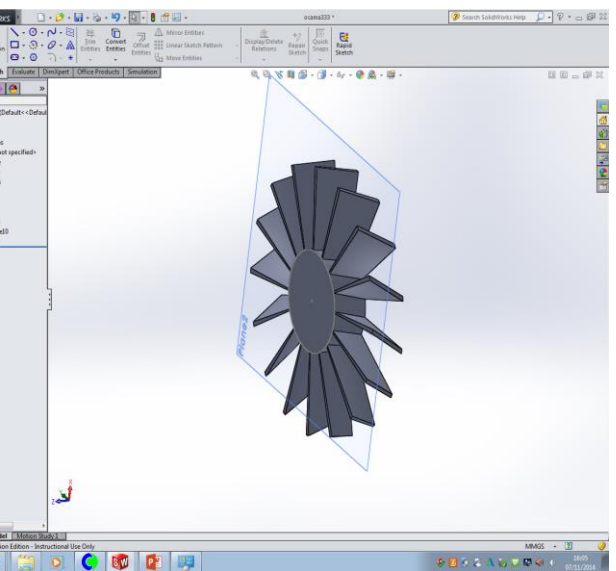
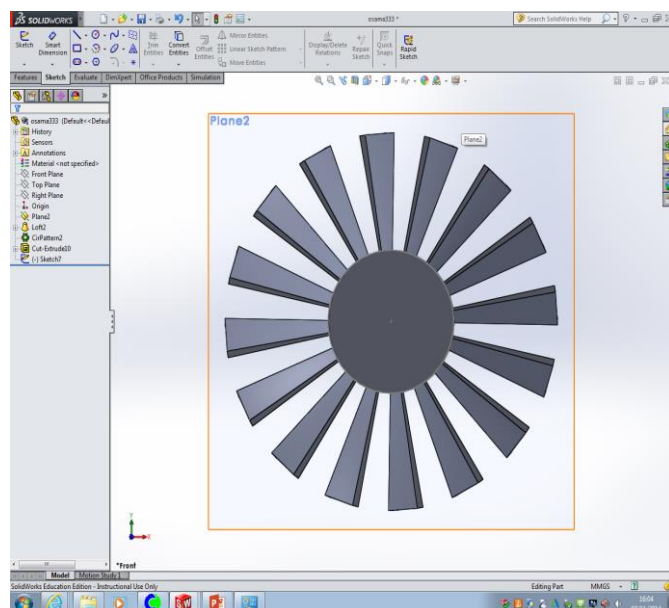


Figure 10: The second design fan under the name (11-11)

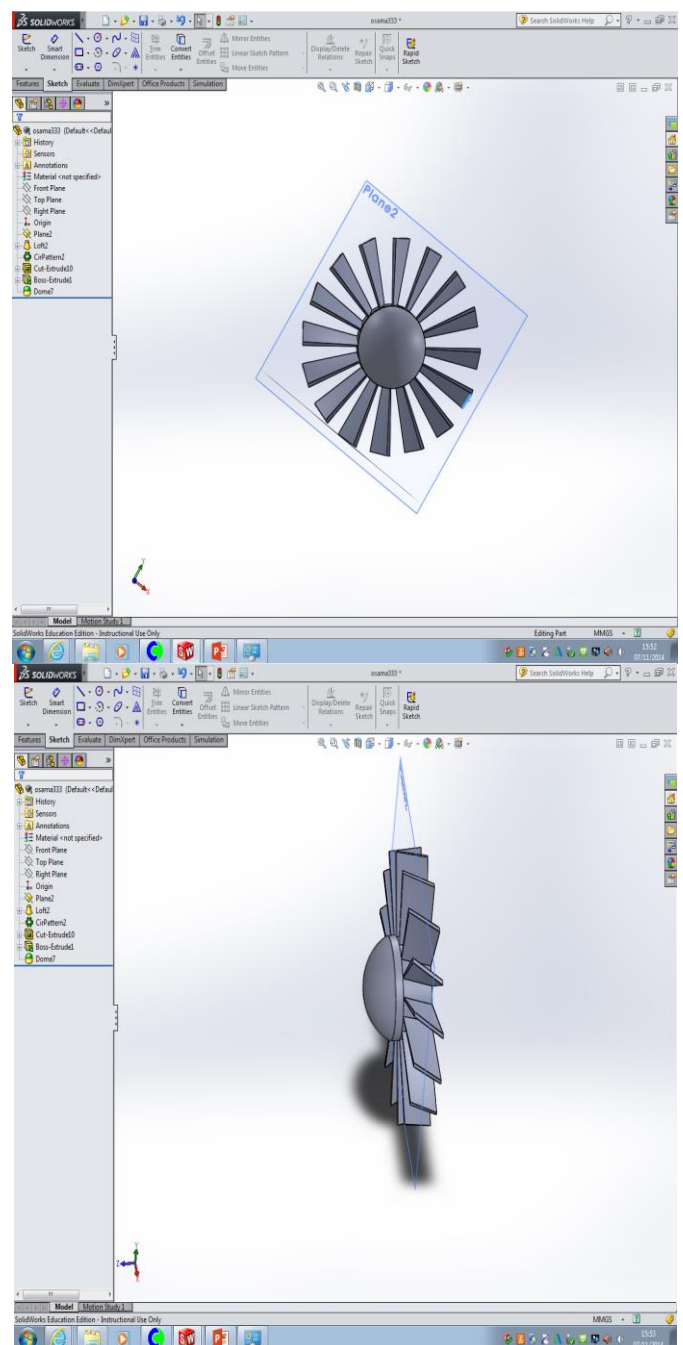


Figure 11: The third design fan under the name (22-22)

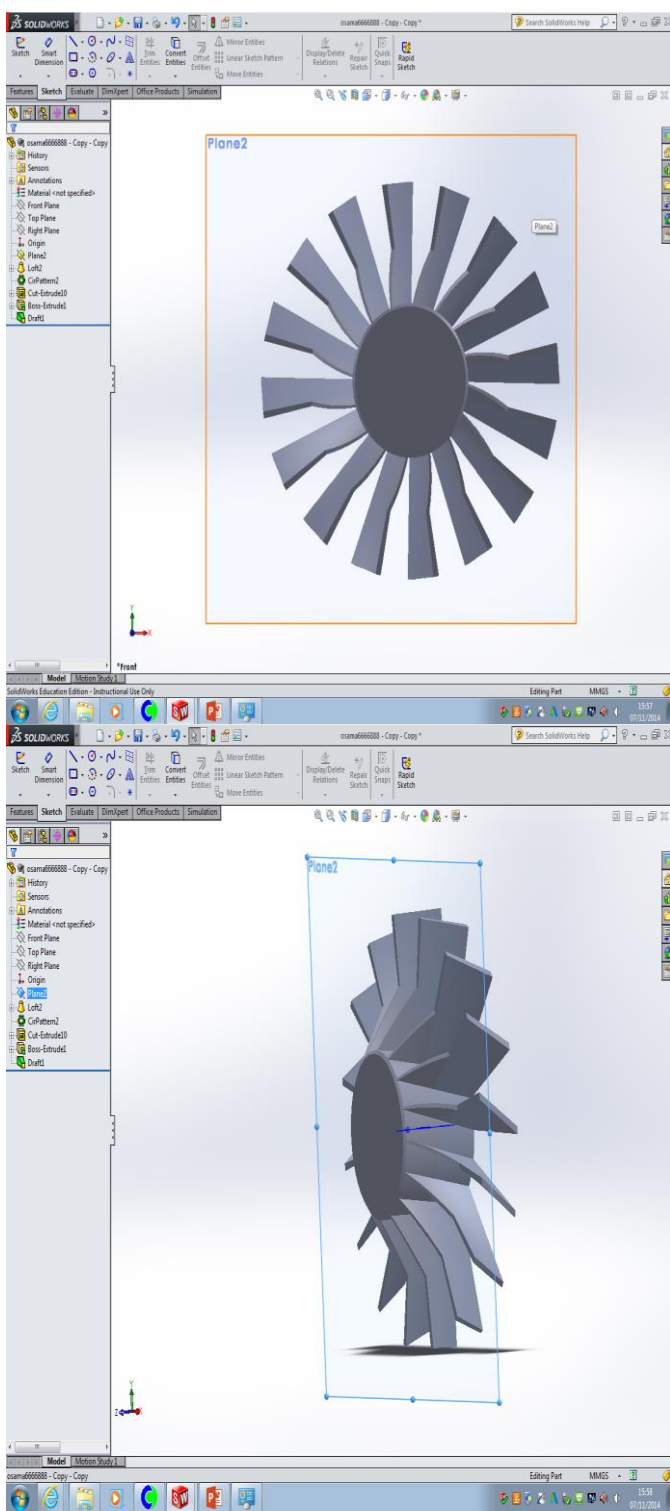


Figure 2: The fourth design fan under the name (33-33)

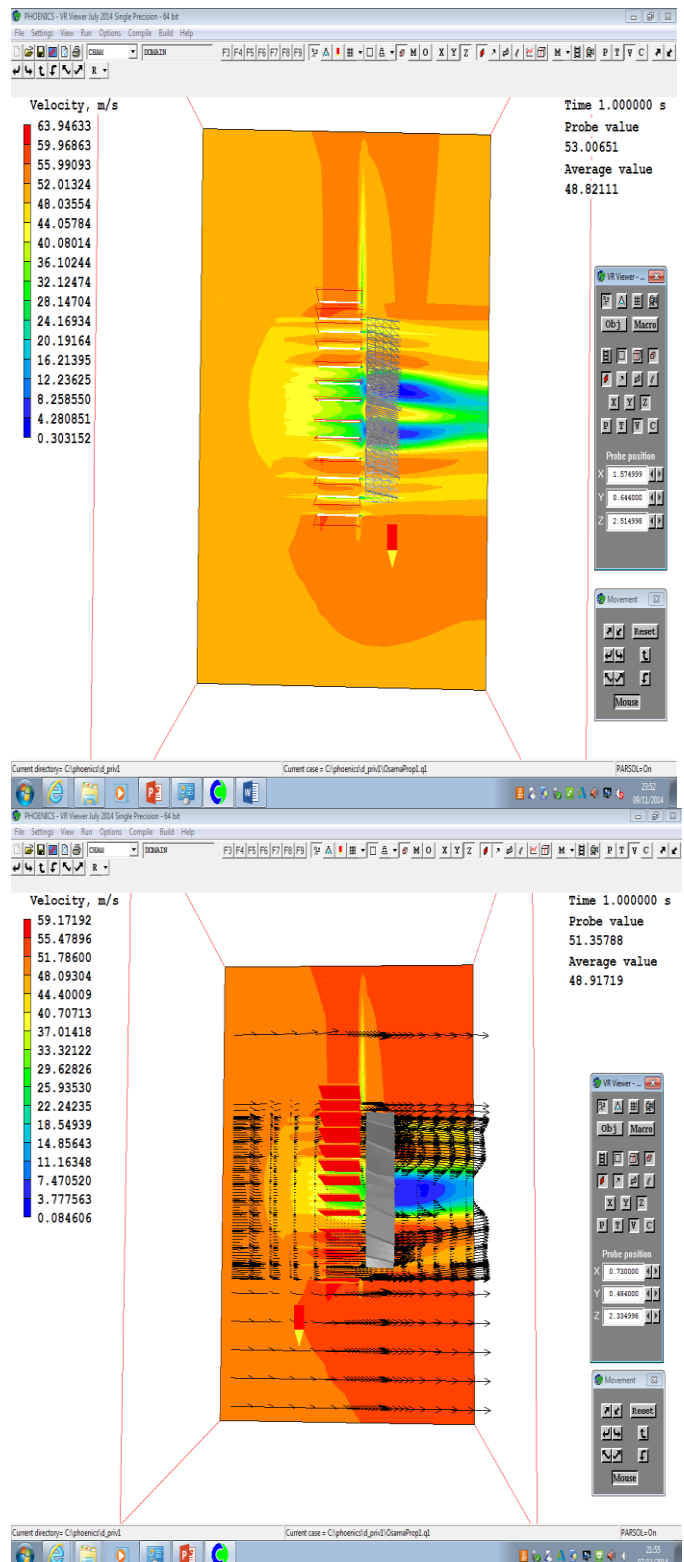
4. RESULT

The results of the experiment are presented below in table and graphical format; followed by drawing inference from findings; non-dimensional data will be used in the analyses to eliminate/isolate the effects of initial boundary conditions of velocity at the inlet which could influence the result of the experiment. The result and analyses are presented in the following subsequent subsections:

It was observed above that the velocity vary along the X-axis as the probe moves progressively from the inlet to outlet, an indication that the momentum from the rotating impeller was being transferred to the flowing air in the domain further confirmed this trend with increment and variation more pronounced when RPM was 3000 rev/min. For all RPMs and more pronounce at 1500 and 3000 rev/min, the velocity dropped suddenly after the impeller at the downstream region of the flow; an indication of energy drop and hence formation of vortices and turbulence initiation (Stafford et al., 2010; Stokse, 2004).

For all RPMs, the pressure was relative stable with minute variations; however, the value of pressure at 60 rev per min was the greatest, follow by pressure for 1500 rev per min and the lowest pressure was obtained at 3000 rev per min. This is an important finding that suggest that increasing RPM does not necessarily translate into increasing pressure, and hence pressure build up is determine by nature of systems; either the system is open channel, closed of partially closed. The more closed a system is, the more the effect of RPM will have on pressure build up in such system (Stokse, 2004; Kreith, 1999; Elmnifi et al., 2021).

It was observed that the velocity upstream of the hub for all specimen actually increased approaching the domain of the hub and more pronounced for the original impeller, followed by the impeller with dome and hybrid hub(cylindrical with spherical end) and velocity was relatively stable upstream for the impeller with cylindrical hub. The order of increasing influence of impeller shape hub on increment of velocity upstream of impeller.



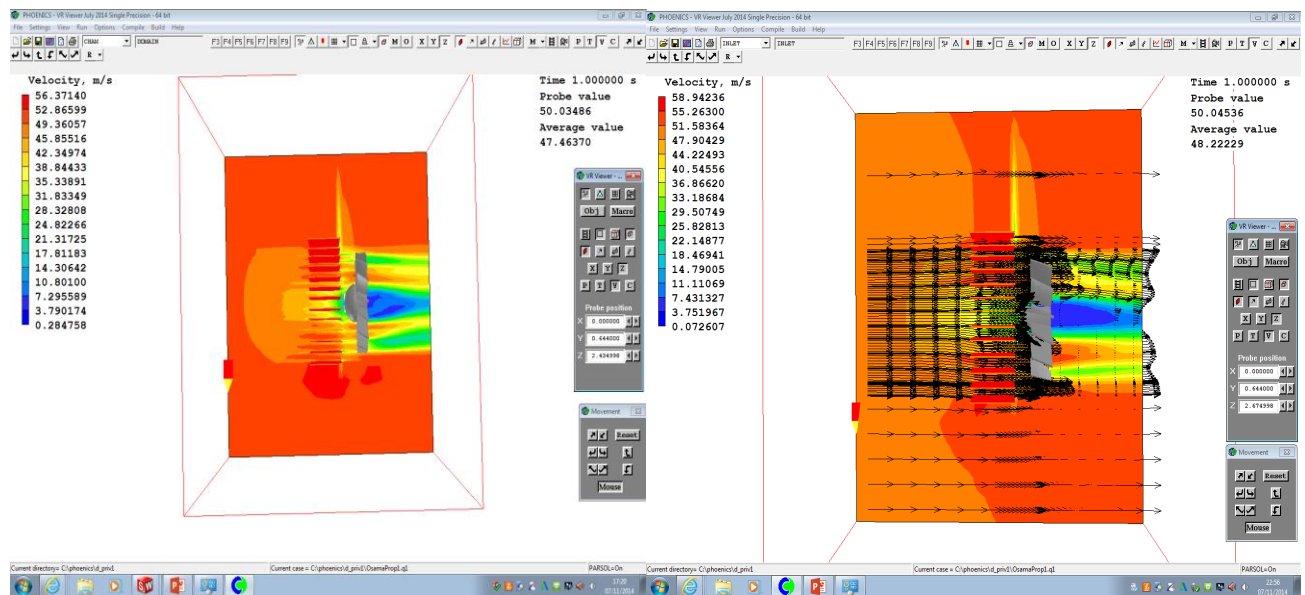


Figure 13: Comparing between the values of velocity four different designs.

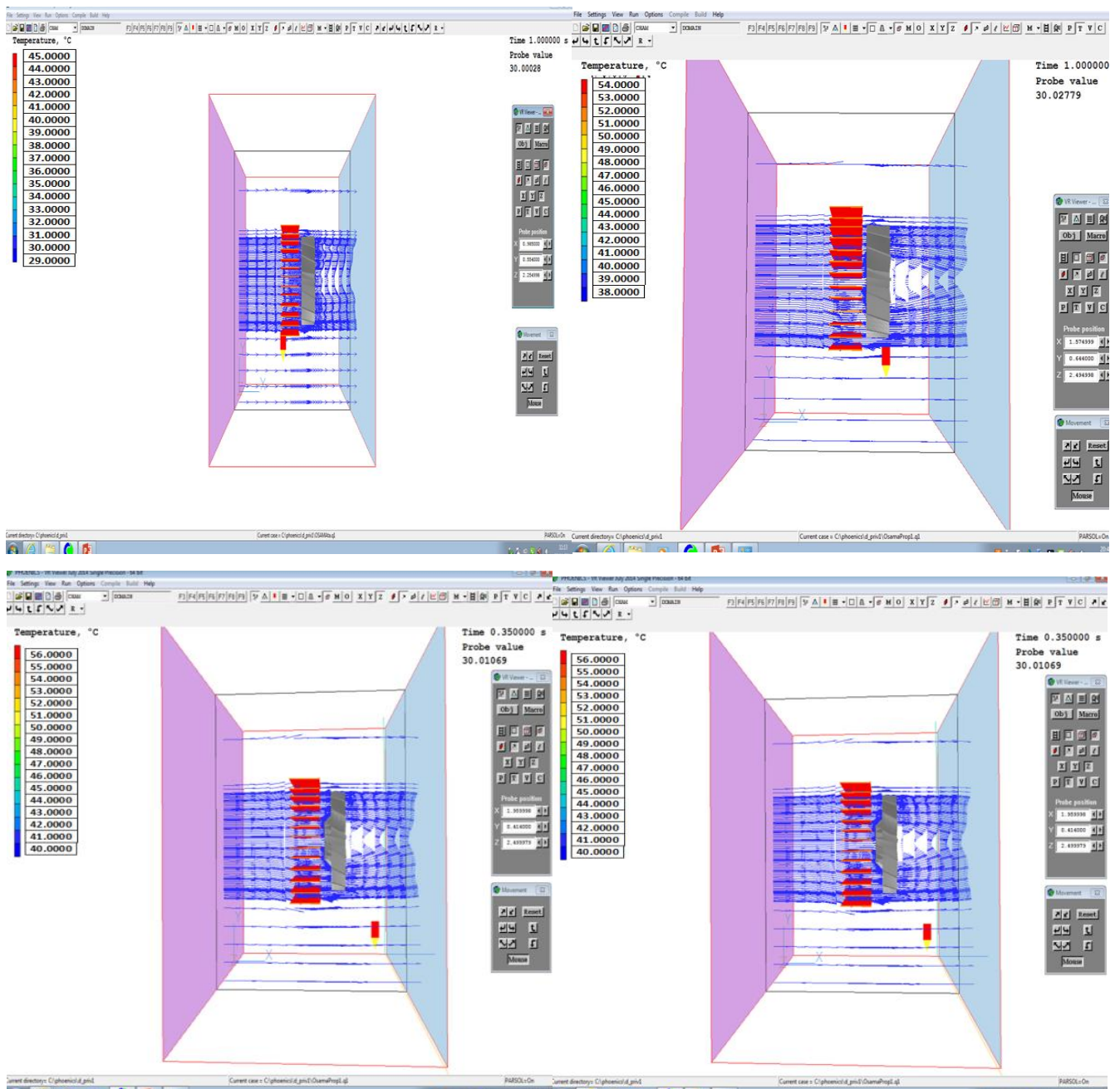
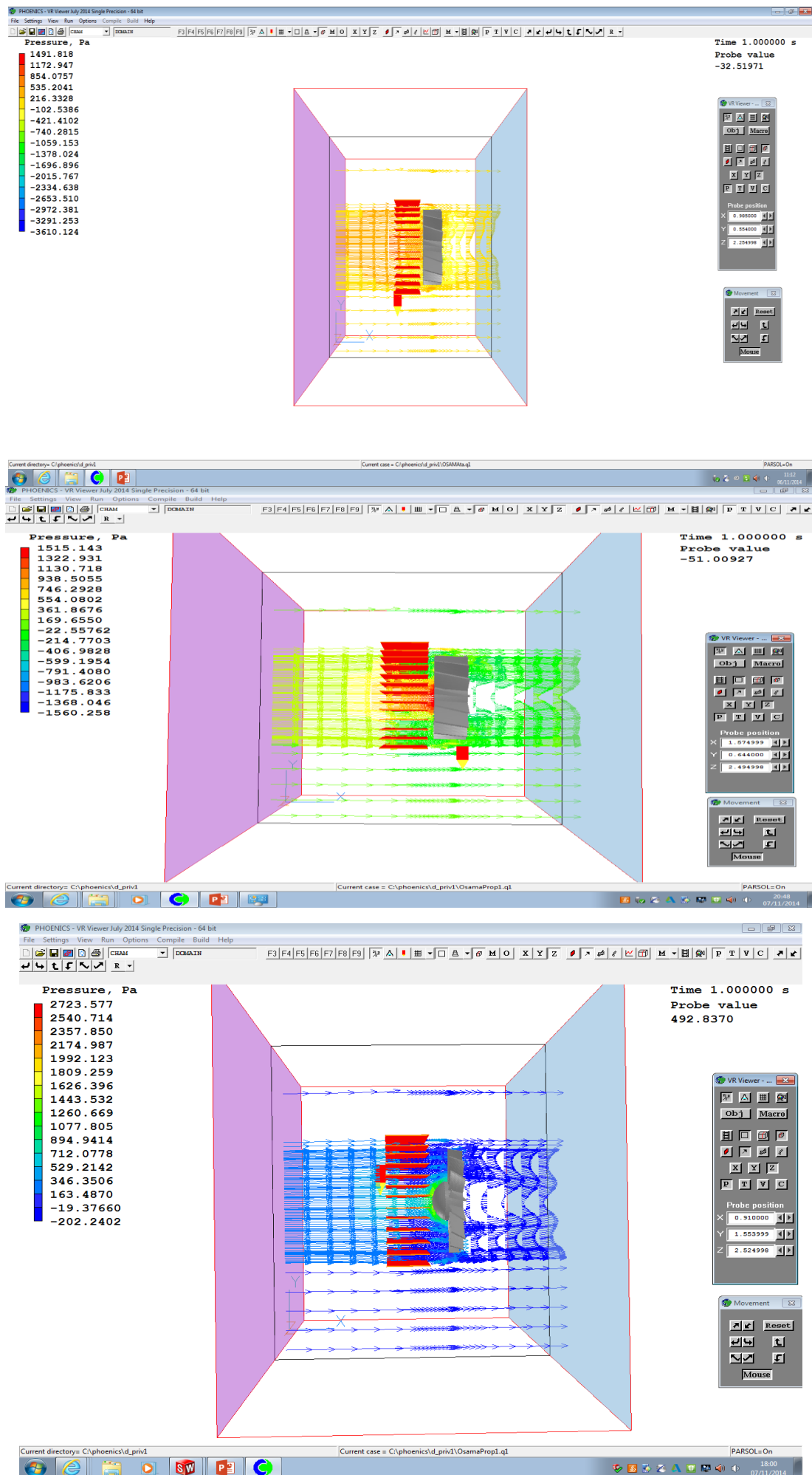


Figure 14: Comparing between the values of temperature four different designs of fans.



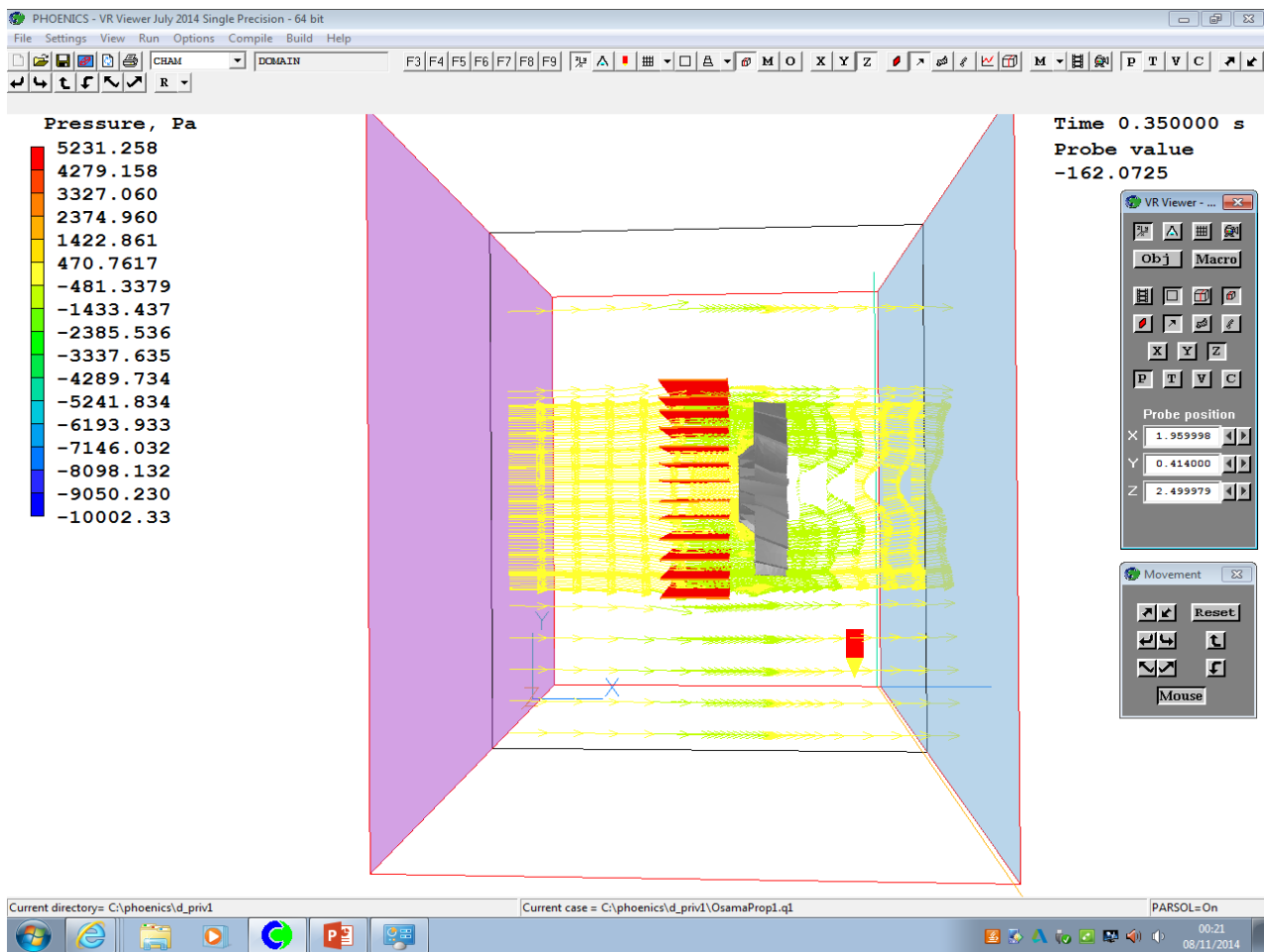
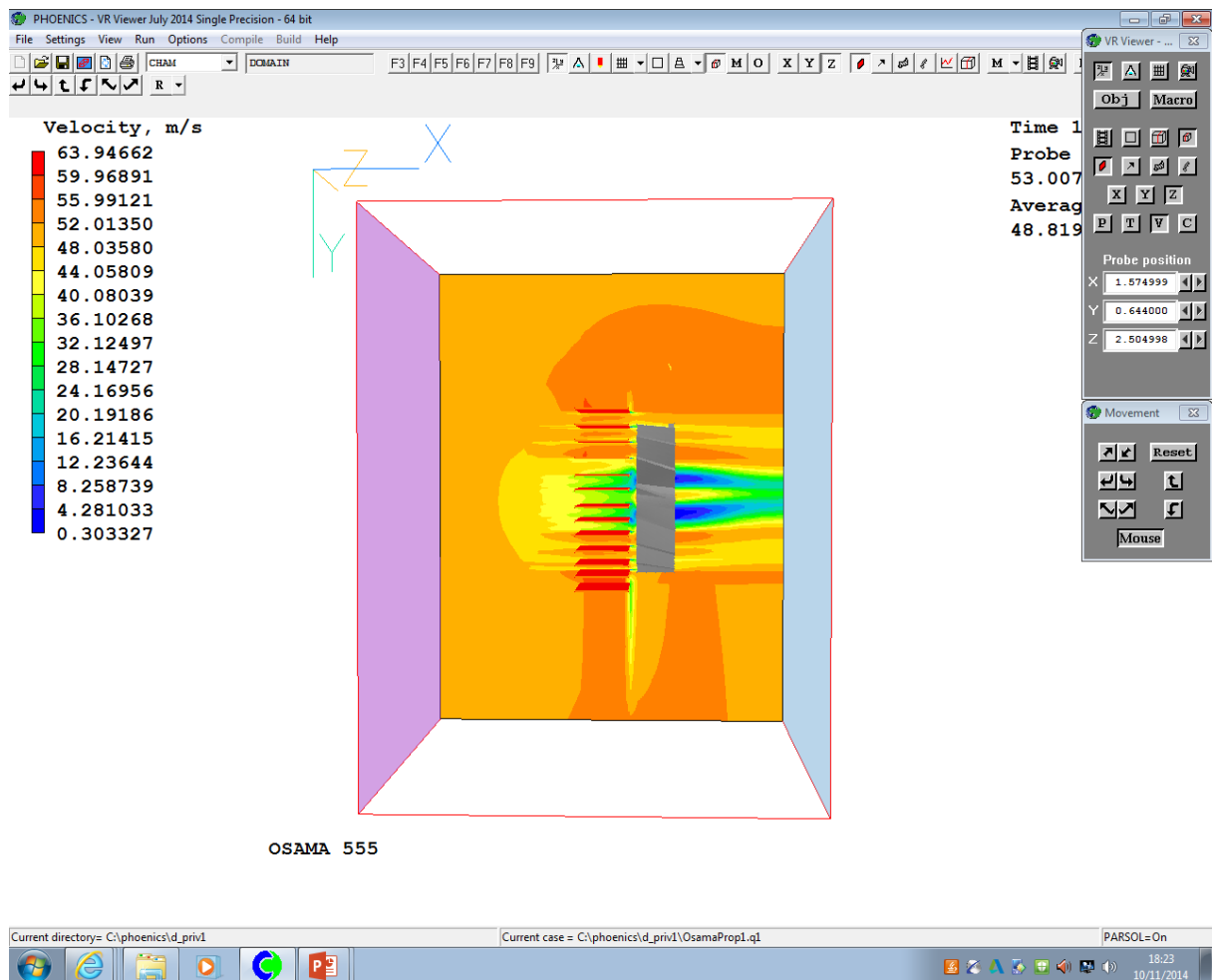


Figure 15: Comparing between the values of pressure four different designs of fans.



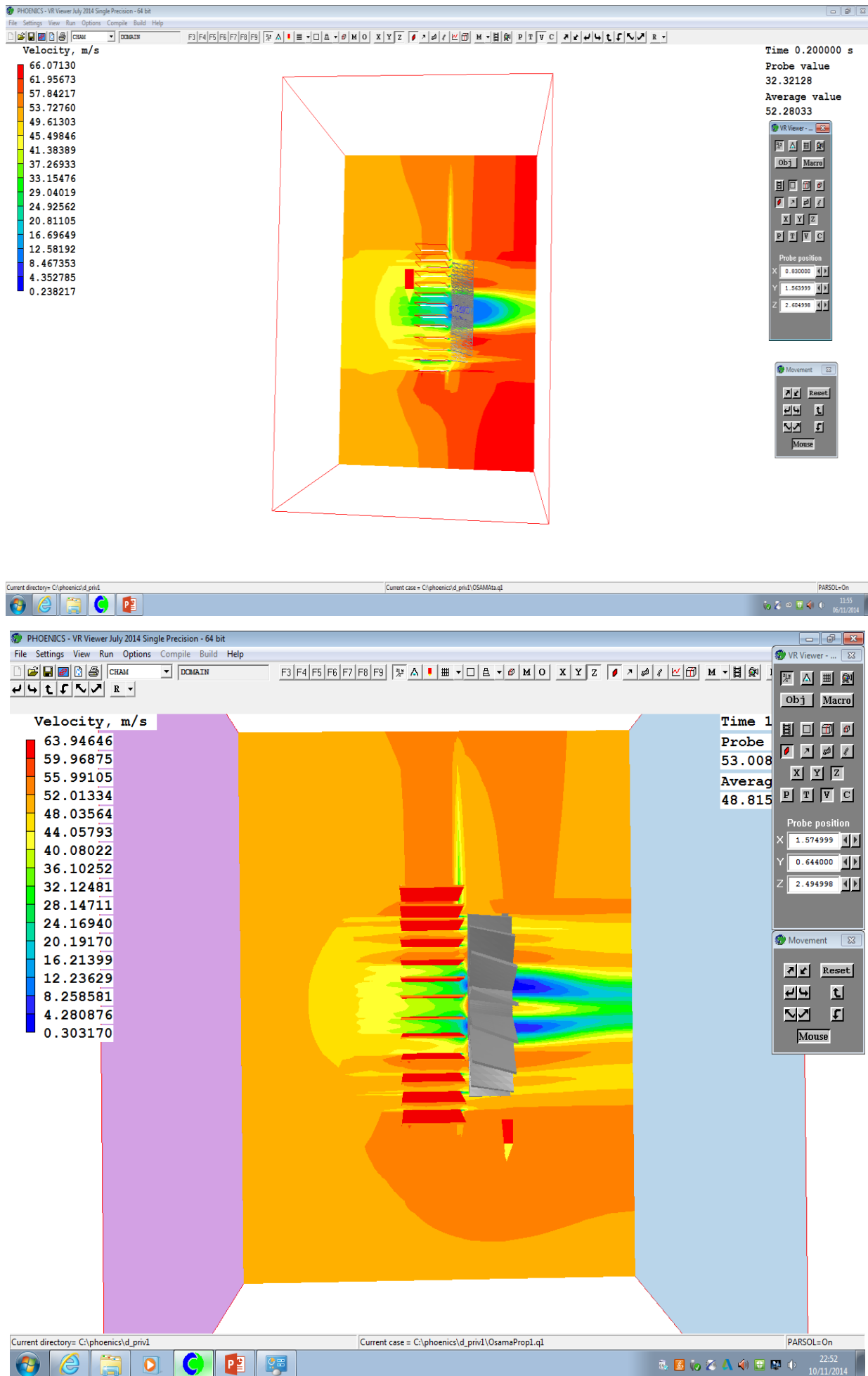
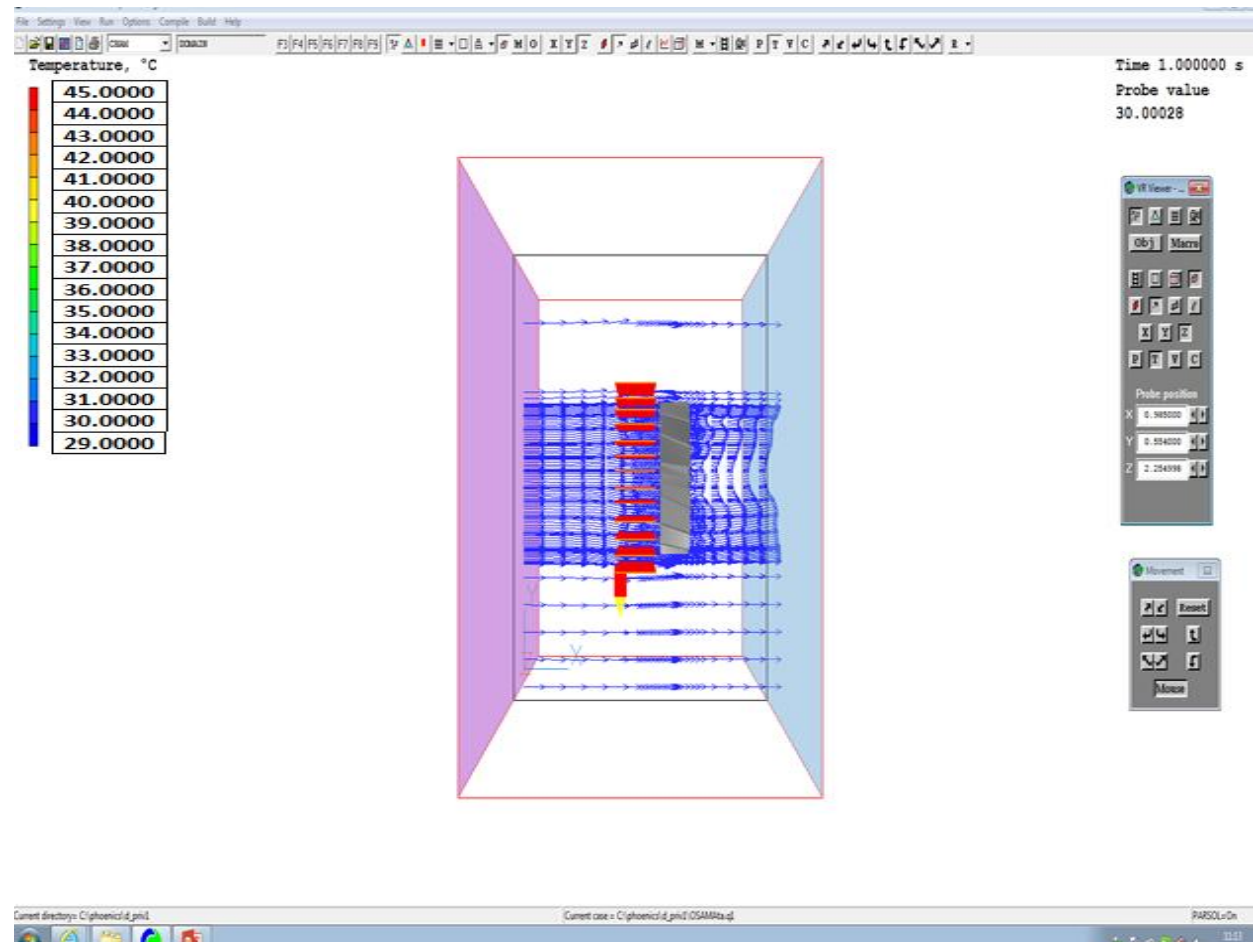
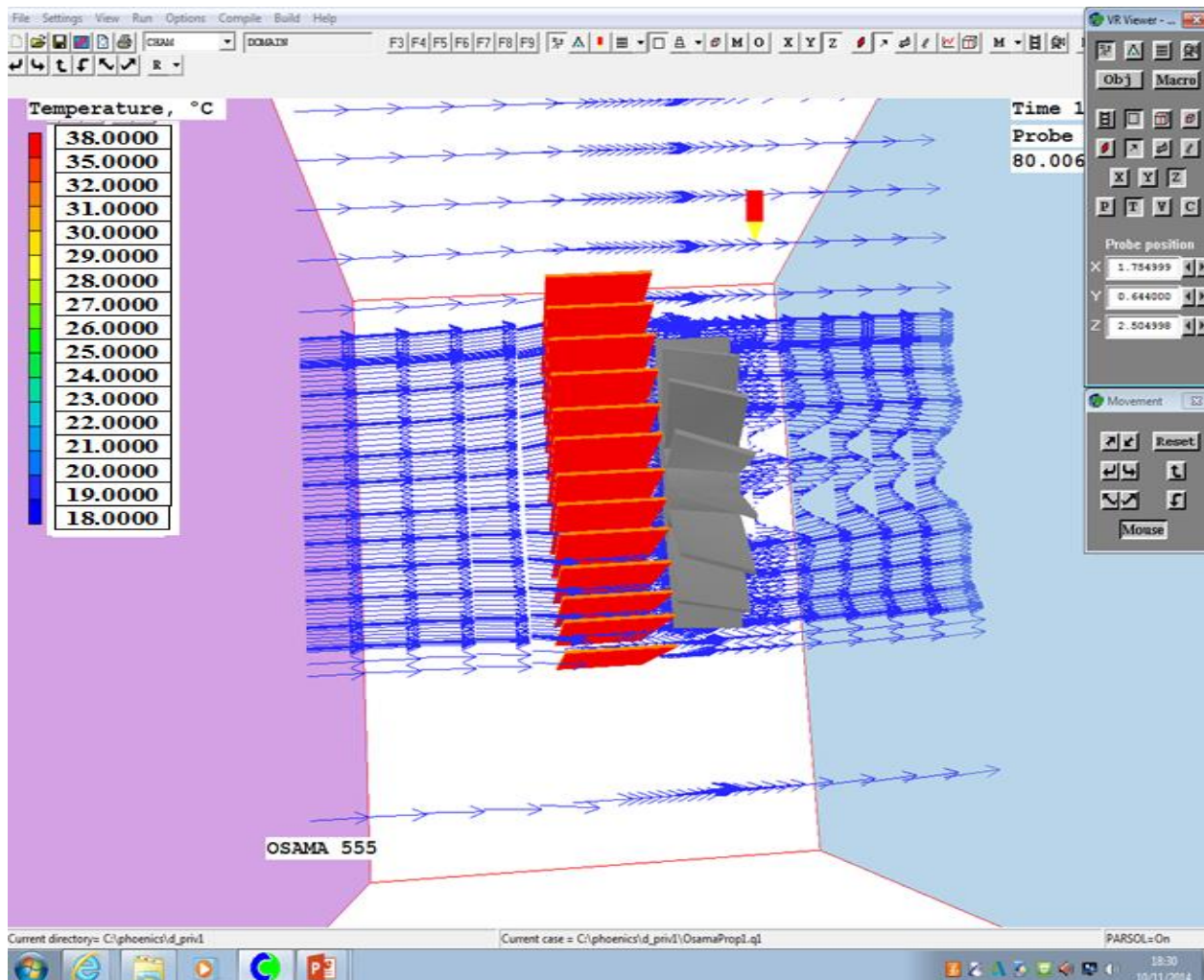


Figure 16: Comparing between the values of velocity for three different temperature (50°C, 70°C, 90°C) fan (00-00).



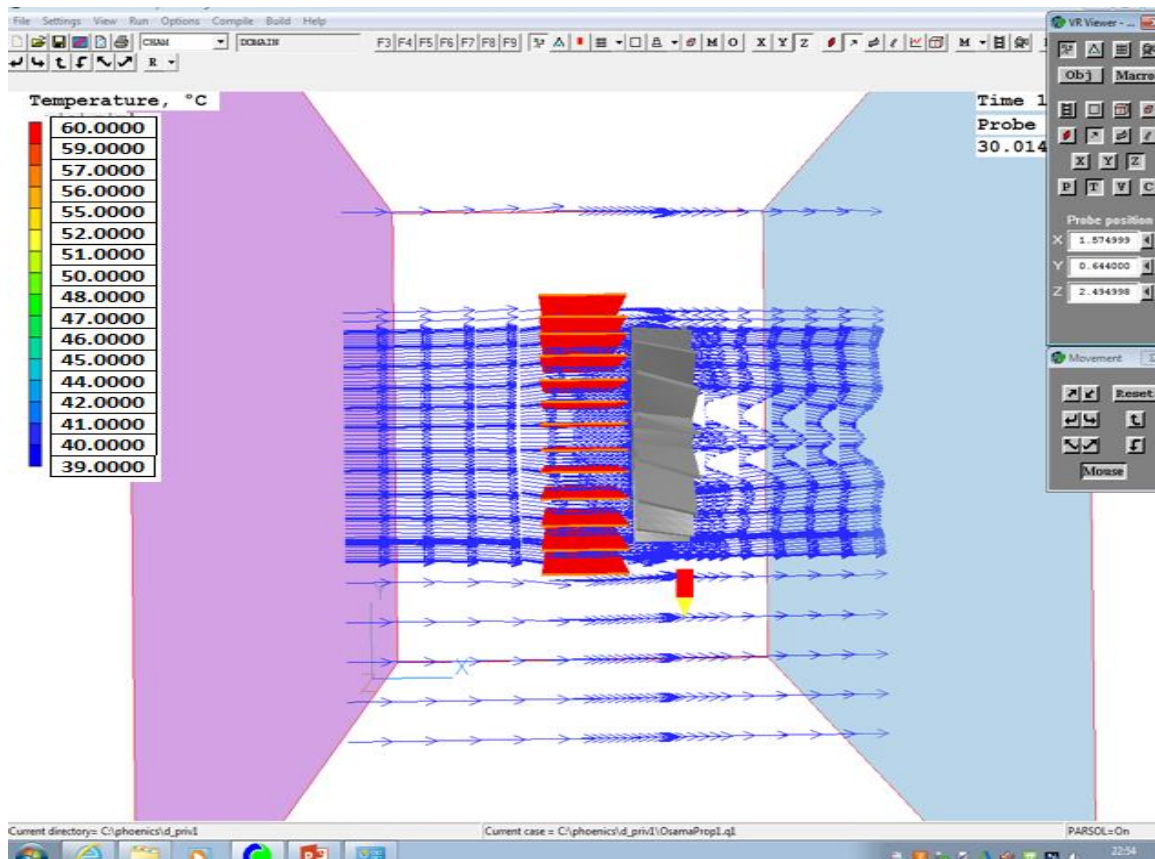


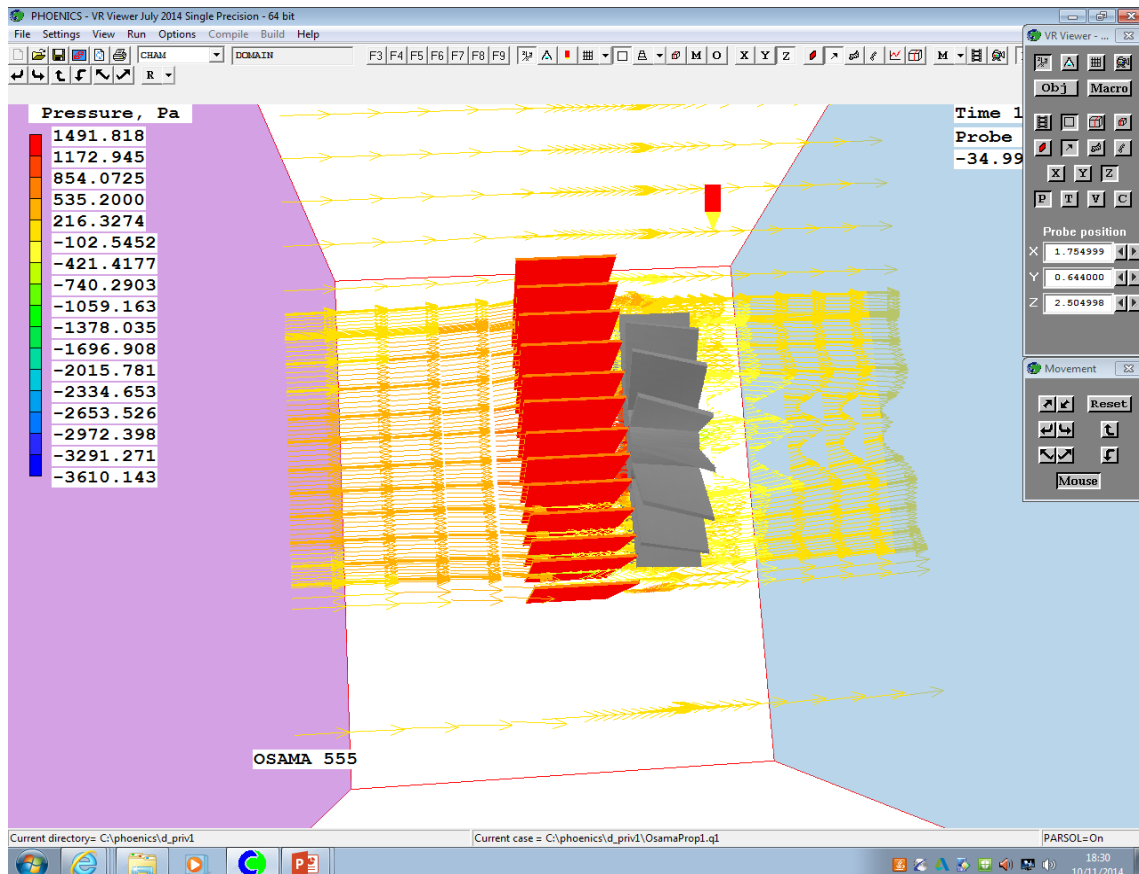
Figure 17: Comparing between the values of temperature for three different temperature (50°C, 70°C, 90°C) fan (00-00)

(zero state) and its symbol (00 - 00) which is the original fan recommended by car manufacturers and which was used for comparison in this project to the other three fans designed for the subject of the study. We note the following:

1. When the coolant temperature is 50°C, the temperature ranges between 18°C - 38°C.

2. When the coolant temperature is 70°C, the temperature ranges between 29°C - 38°C

3. When the coolant temperature is 90°C, the temperature ranges between 39°C - 60°C.



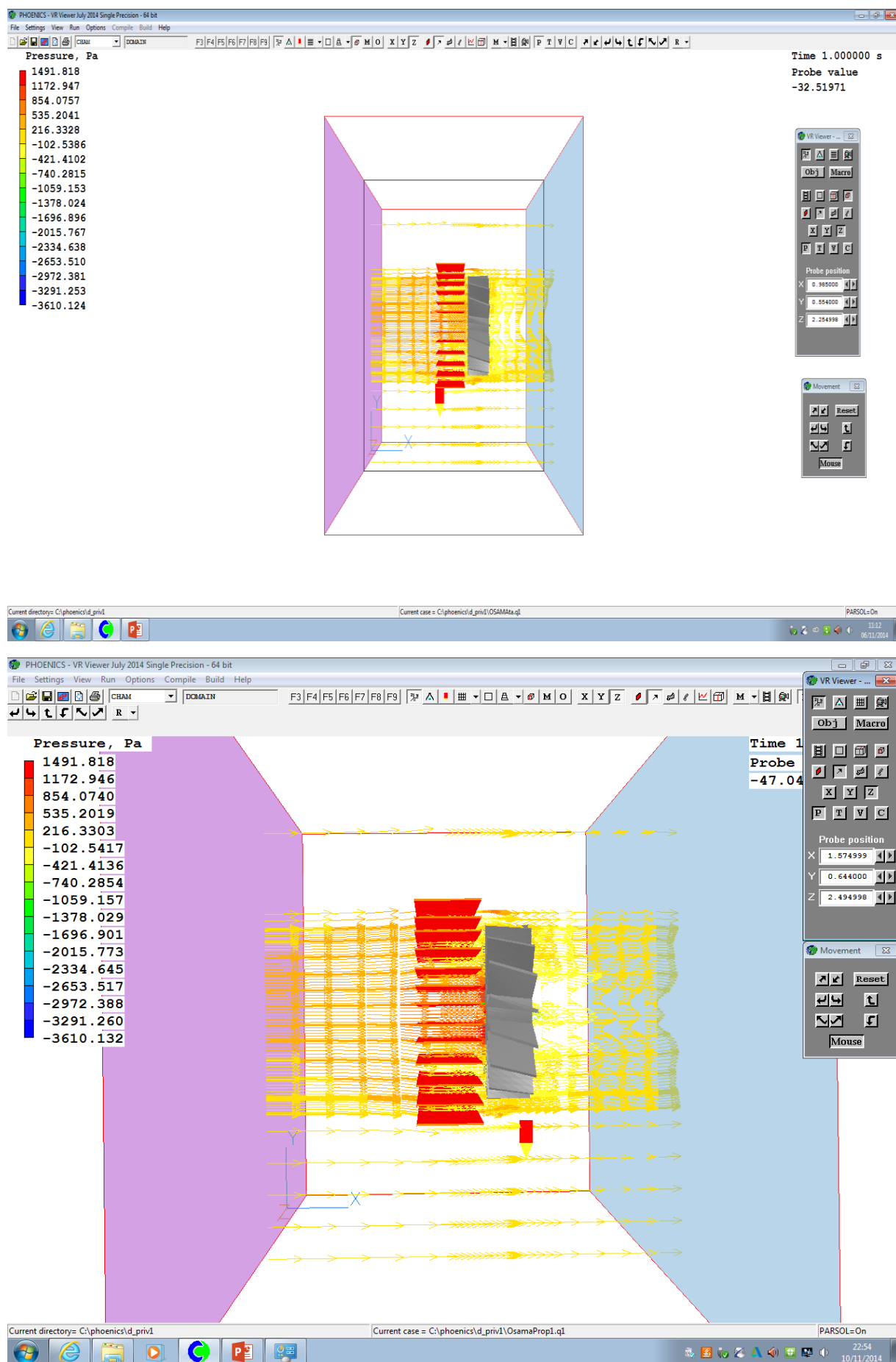


Figure 18: Comparing between the values of pressure for three different temperature (50°C, 70°C, 90°C) fan (00-00).

However, the situation changed a bit downstream after the impeller; the velocity value for impeller with hybrid hub suddenly dropped and remained the lowest throughout the flow domain, followed by the impeller with flat surface and the impeller with dome hub respectively. The velocity

value for impeller with cylindrical hub remained highest at the downstream region of the flow domain. The order of increasing influence of impeller shape hub on increment of velocity downstream of impeller could be summaries.

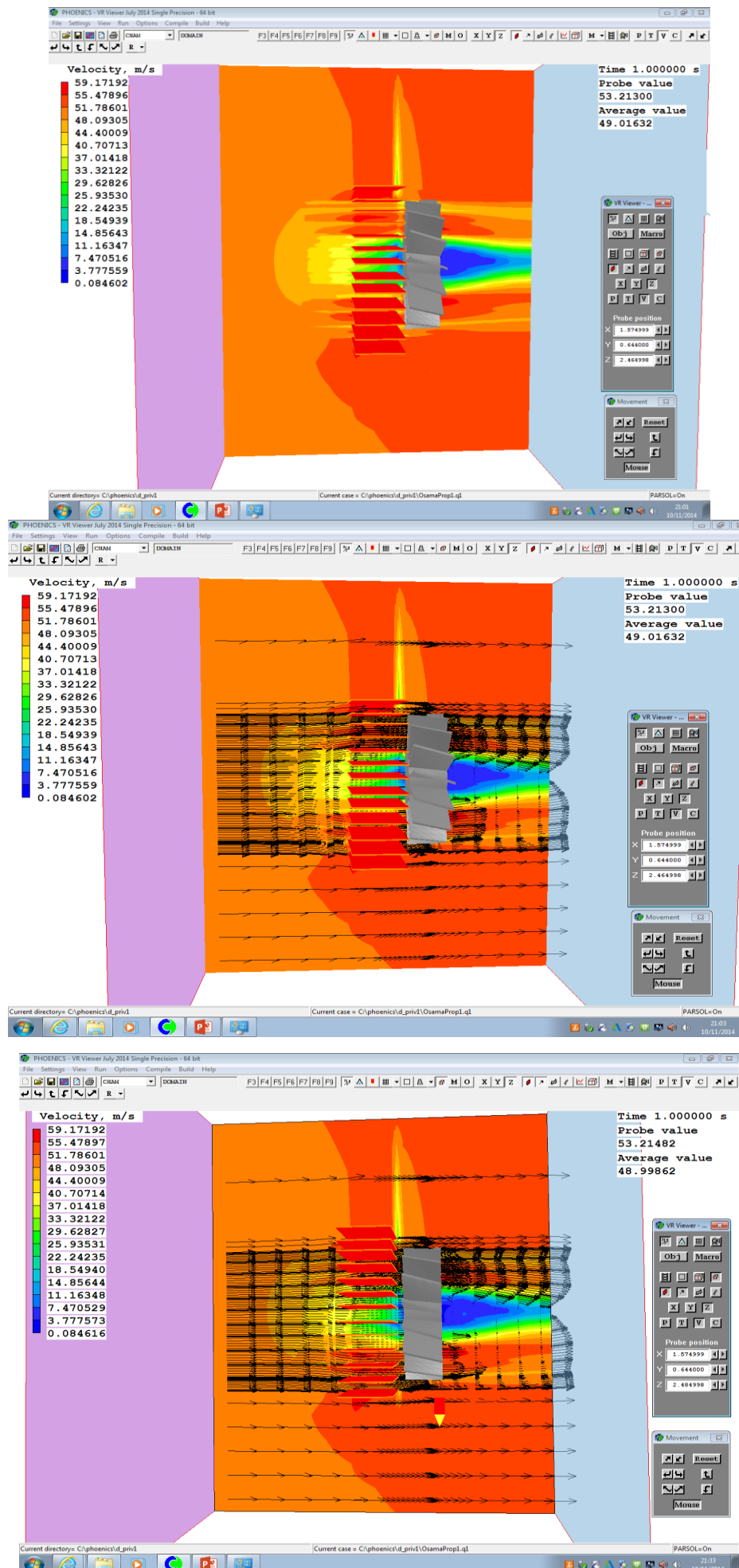
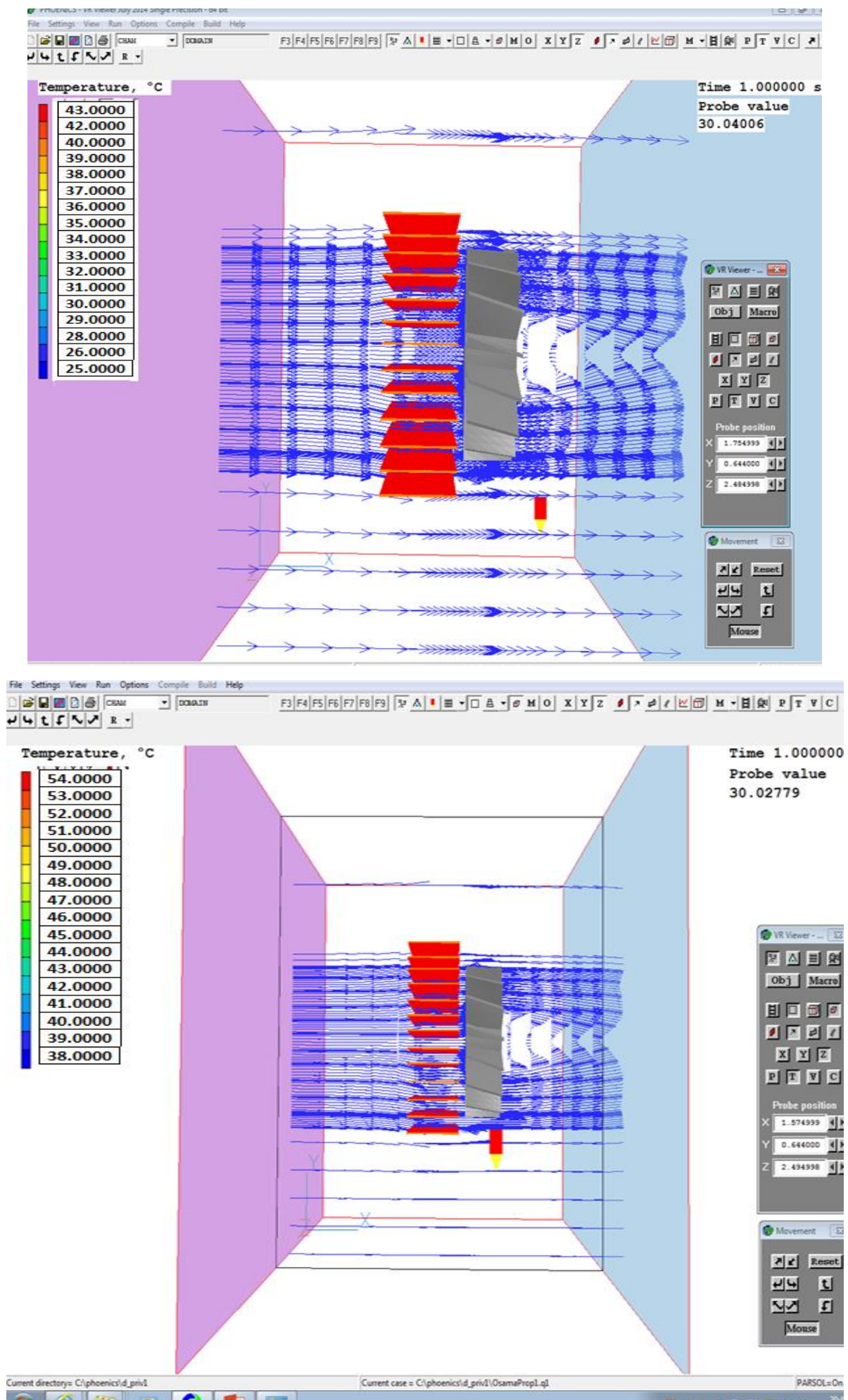


Figure 19: Comparing between the values of velocity for three different temperature (50°C, 70°C, 90°C) fan (11-11)



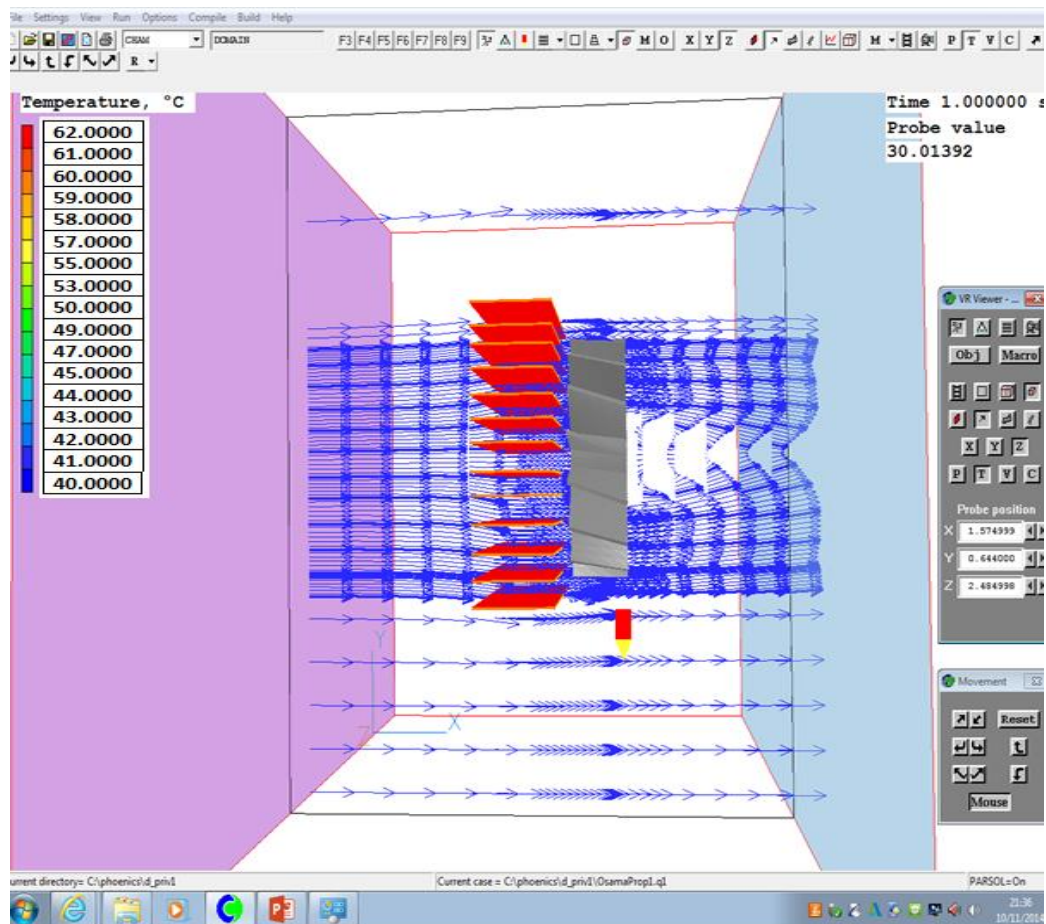


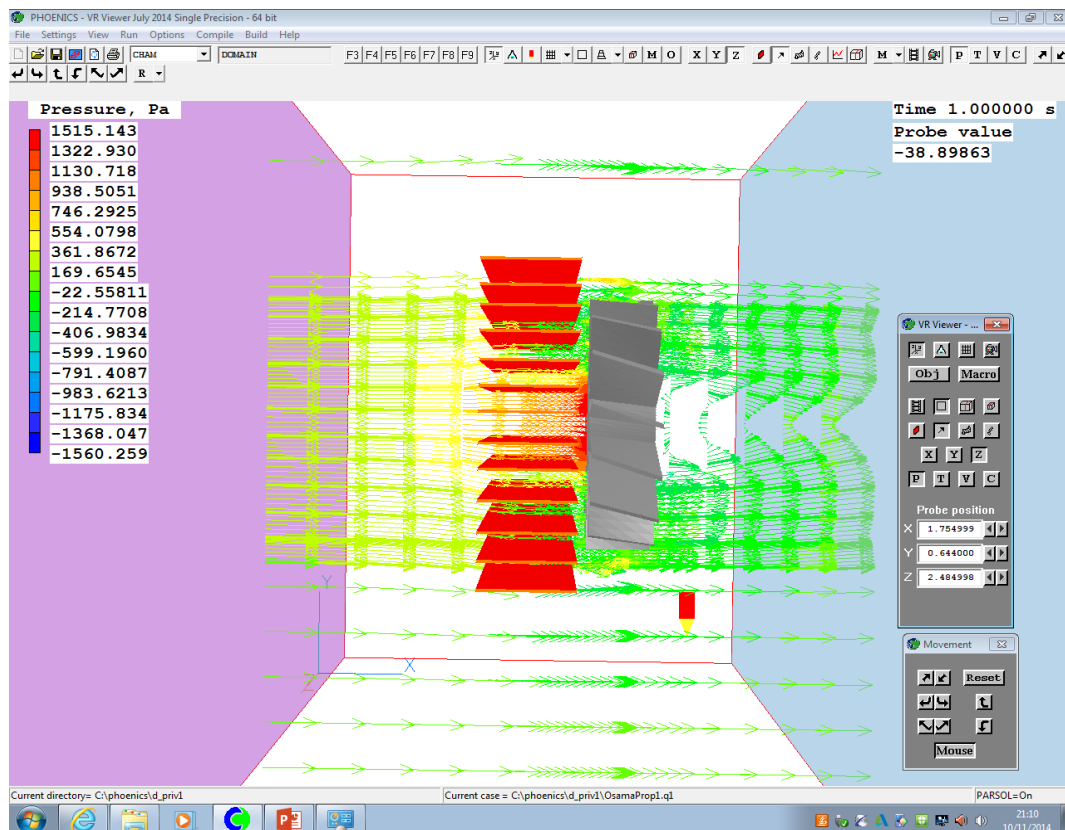
Figure 20: Comparing between the values of temperature for three different temperature (50°C, 70°C, 90°C) fan (11-11)

(the solid state) and the symbolized state (11 - 11) It is a fan similar to the original one recommended, except that it is solid in the middle and not hollow like the original one. We note the following:

1. When the coolant temperature is 50°C, the temperature ranges between 25°C - 48°C.

2. When the coolant temperature is 70°C, the temperature ranges between 38°C - 54°C.

3. When the coolant temperature is 90°C, the temperature ranges between 40°C - 62°C.



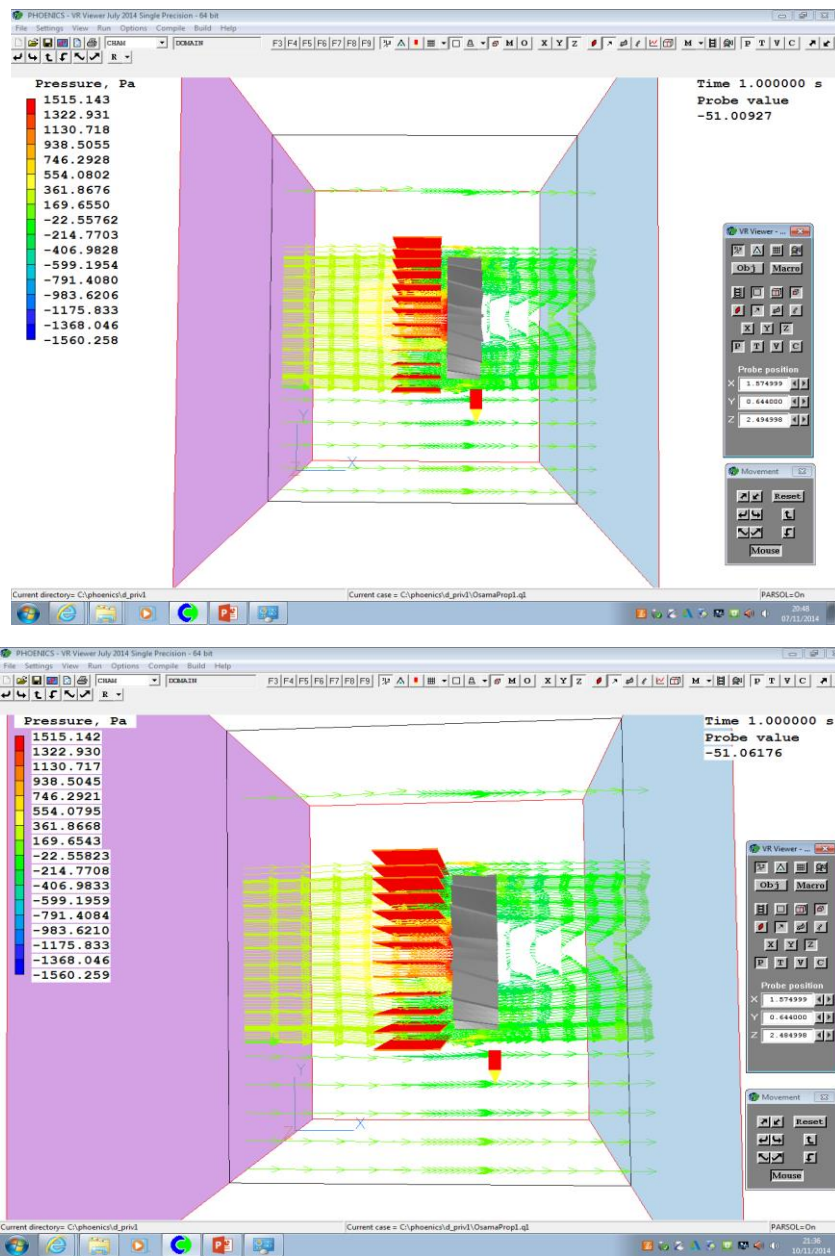
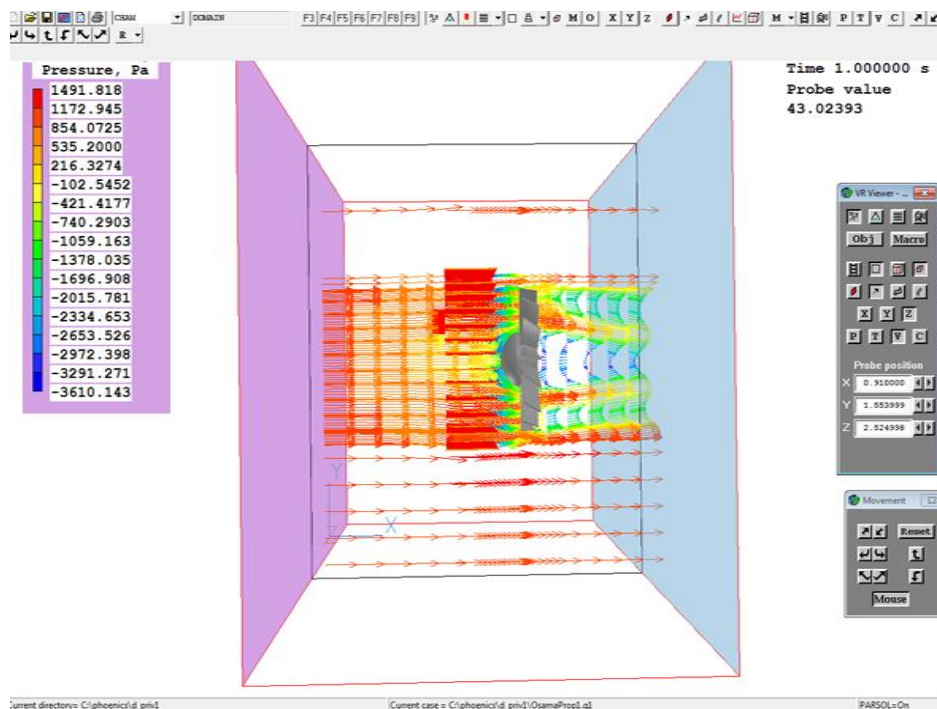


Figure 21: Comparing between the values of pressure for three different temperature (50°C, 70°C, 90°C) fan (11-11)



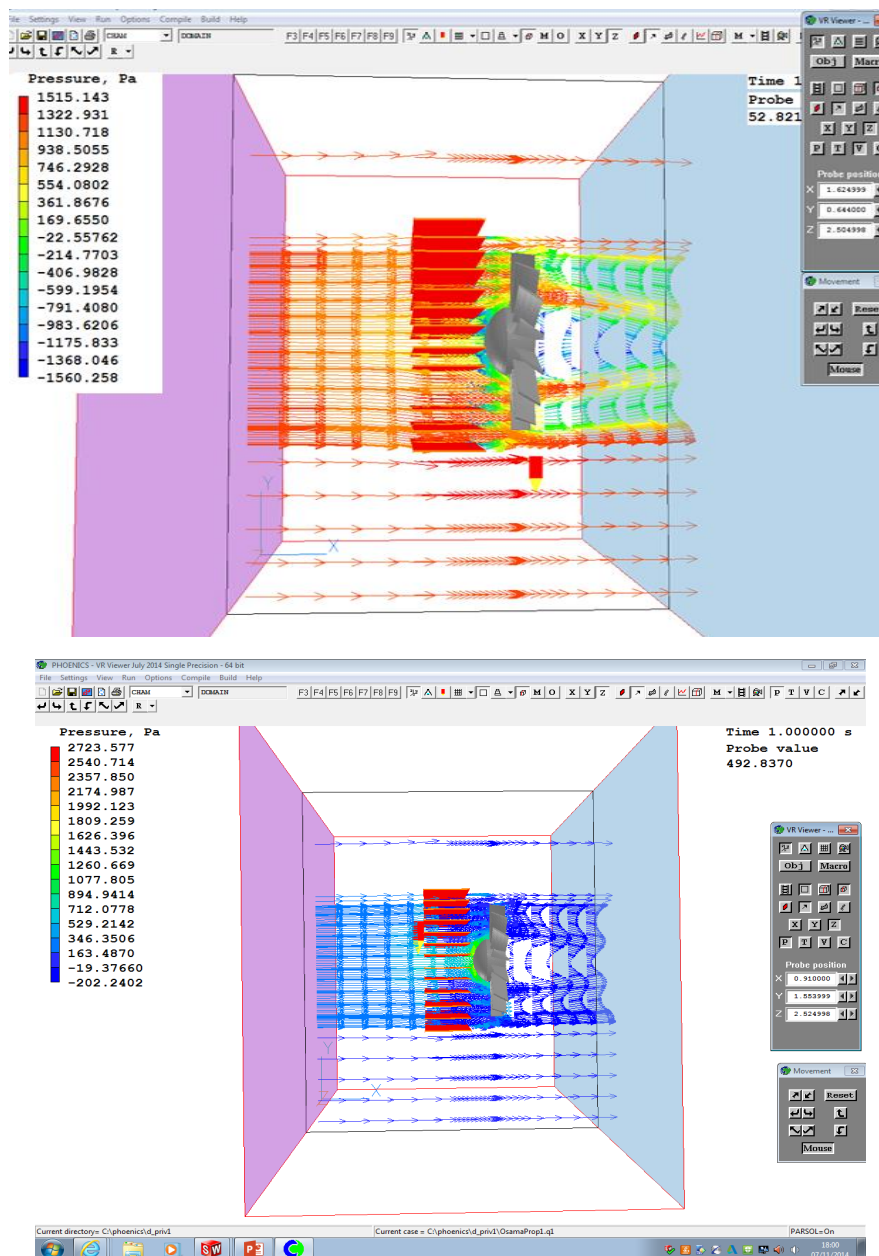
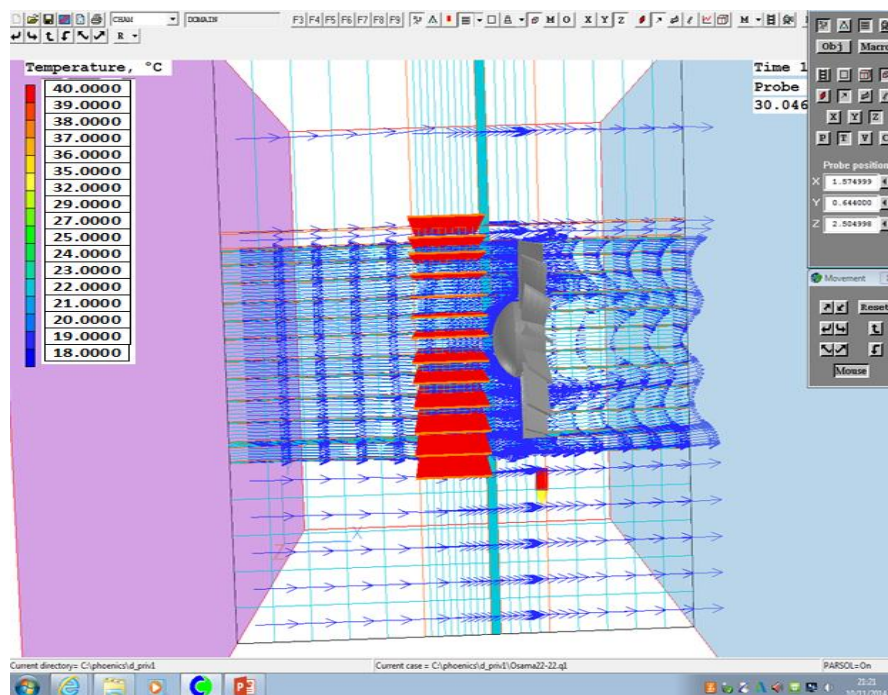


Figure 22: Comparing between the values of pressure for three different temperature (50°C, 70°C, 90°C) fan (22-22).



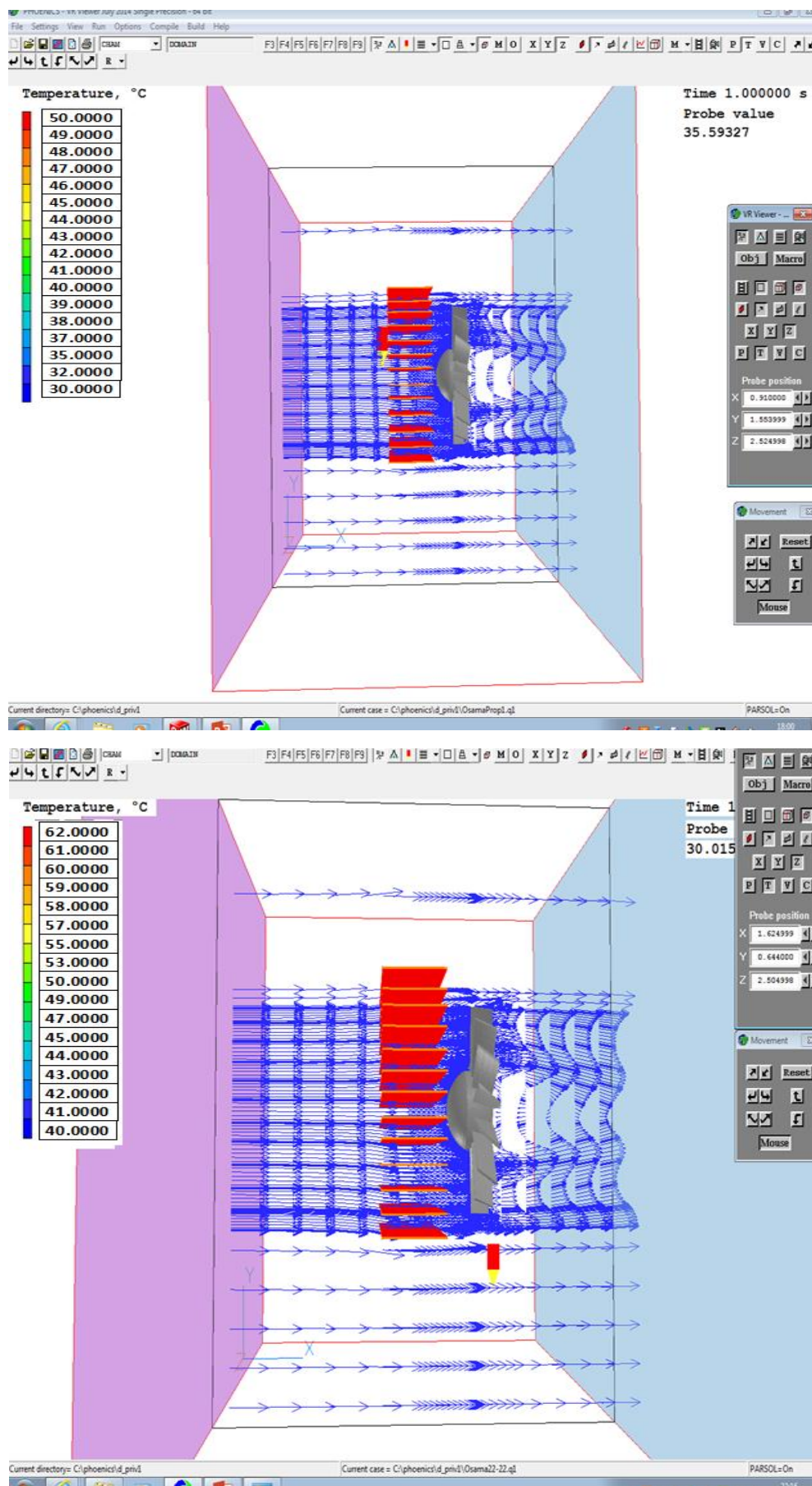
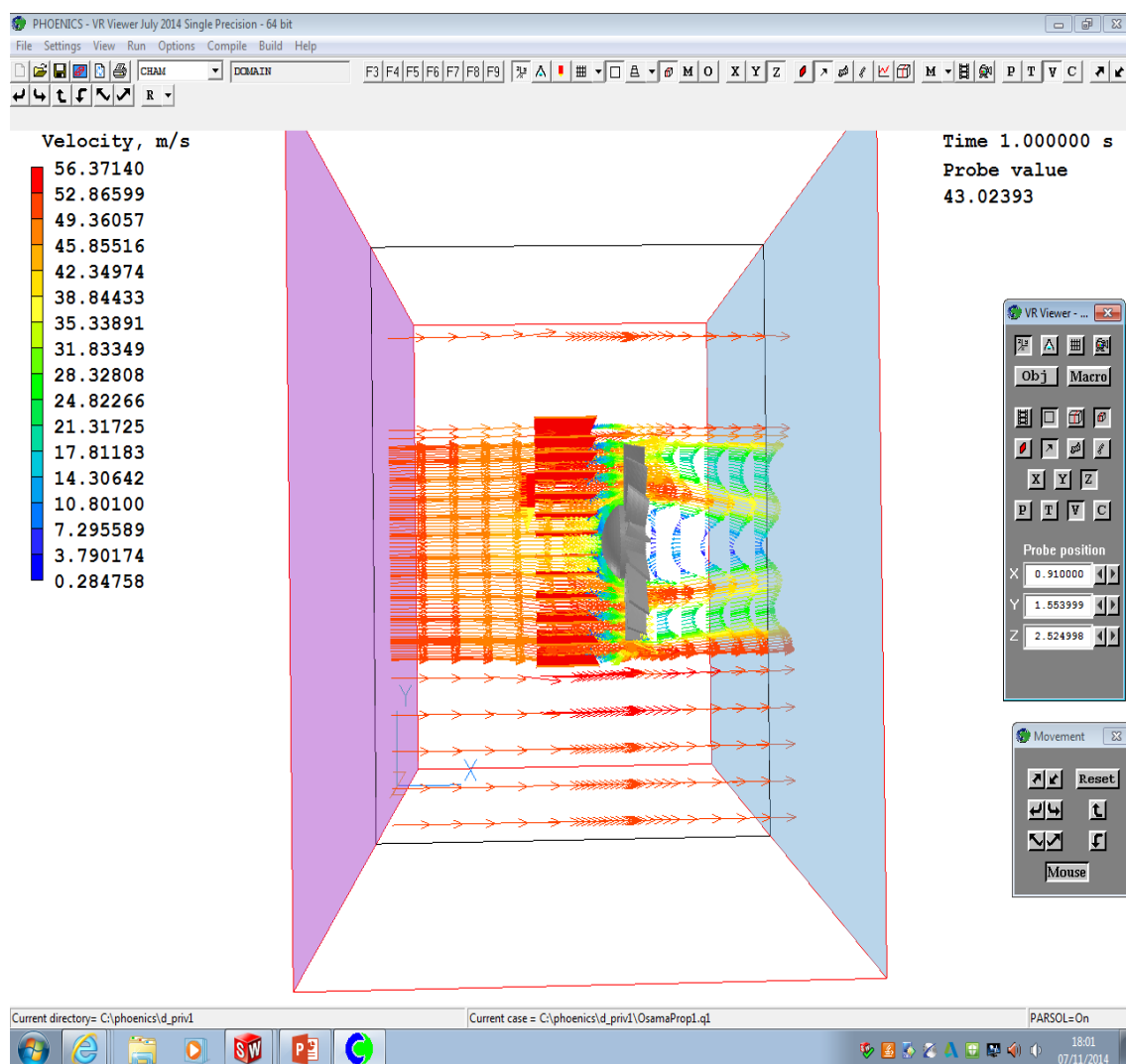
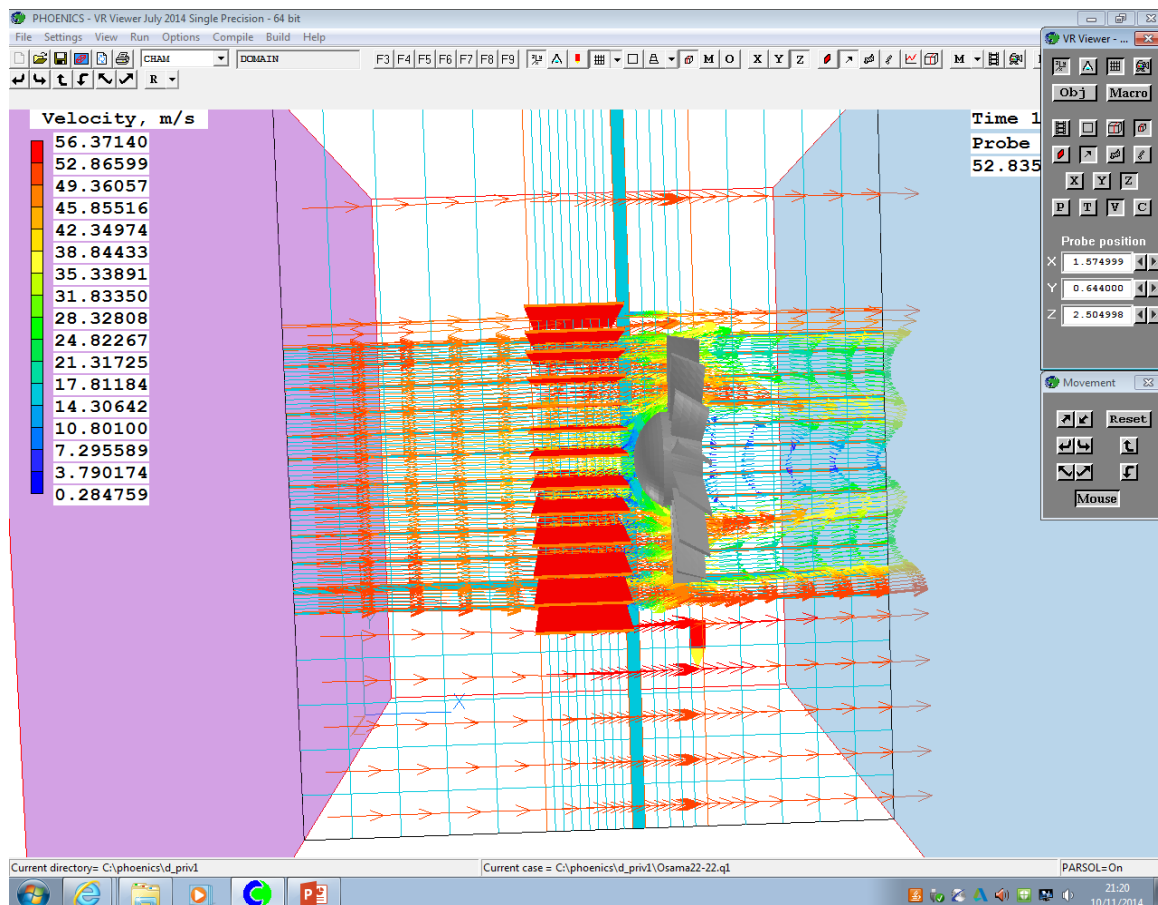


Figure 23: Comparing between the values of temperature for three different temperature (50°C, 70°C, 90°C) fan (22-22).

The case with a convex turbine and its symbol is (22 - 22) It is a fan similar to the original recommended one, except that it has a convex (spherical) center blade. We note the following:

1. When the coolant temperature is 50°C, the temperature ranges between 18°C- 40°C.
2. When the coolant temperature is 70°C, the temperature ranges between 30°C- 50°C.
3. When the coolant temperature is 90°C, the temperature ranges between 40°C- 62°C.



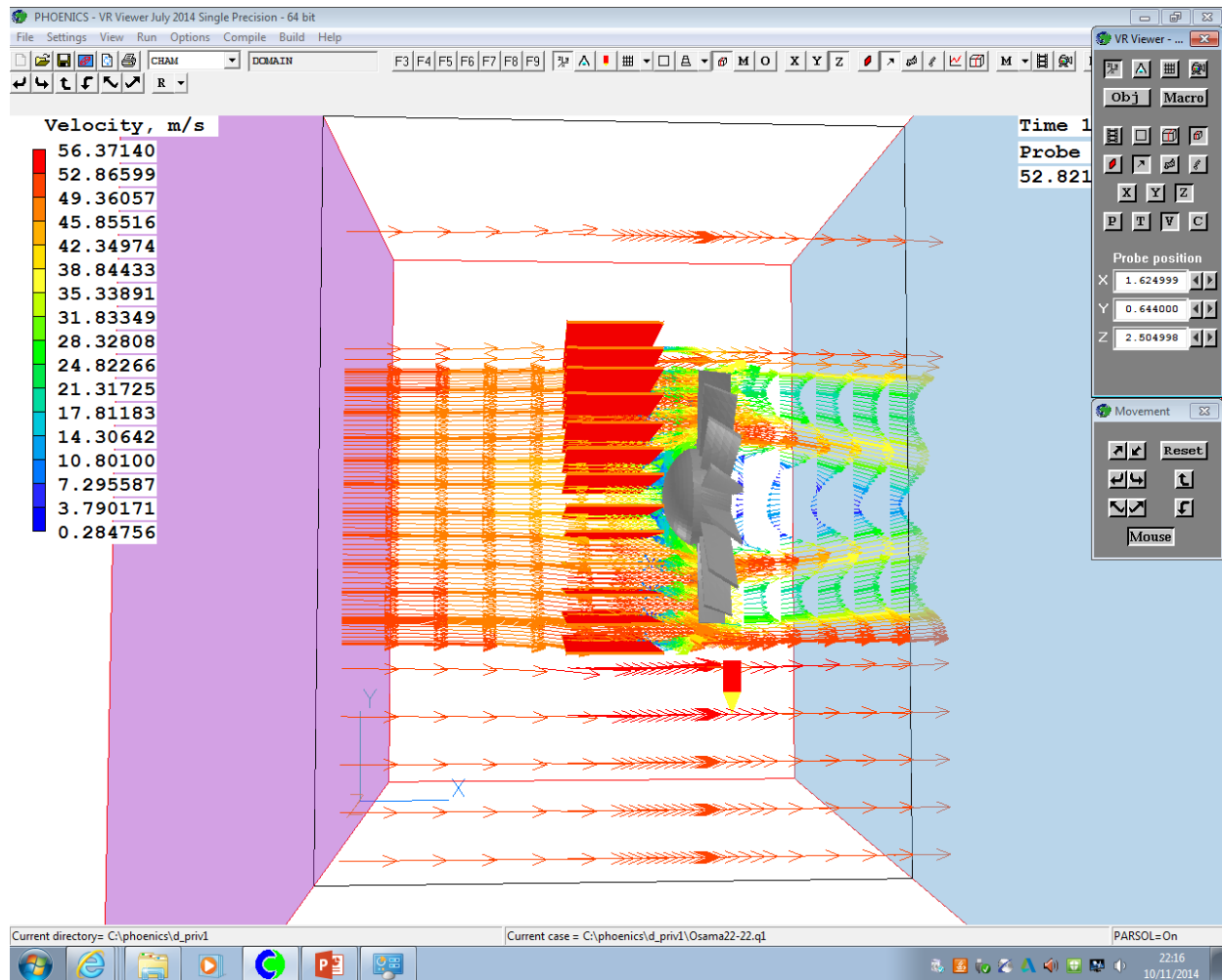
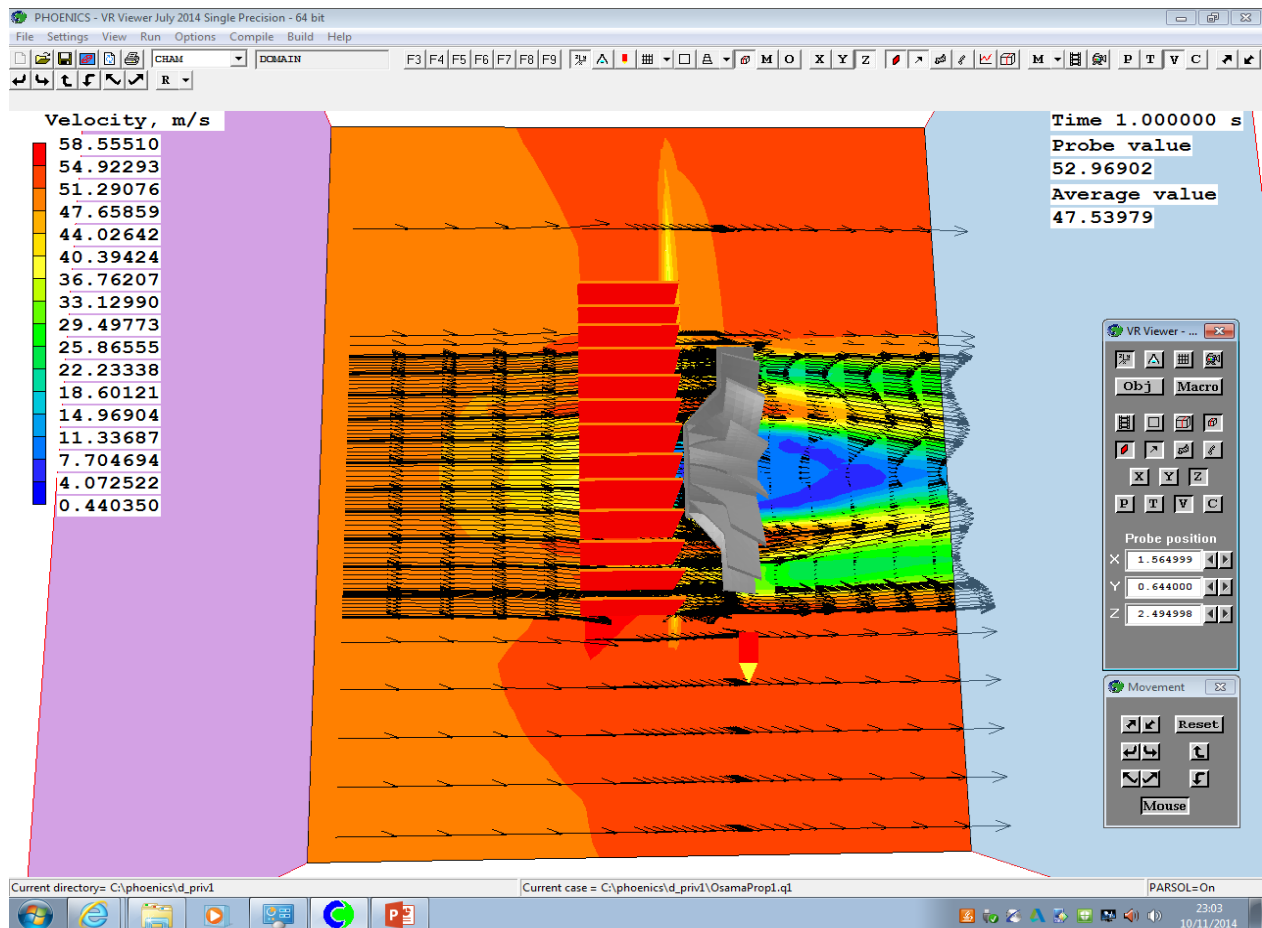


Figure 24: Comparing between the values of velocity for three different temperature (50°C, 70°C, 90°C) fan (22-22).



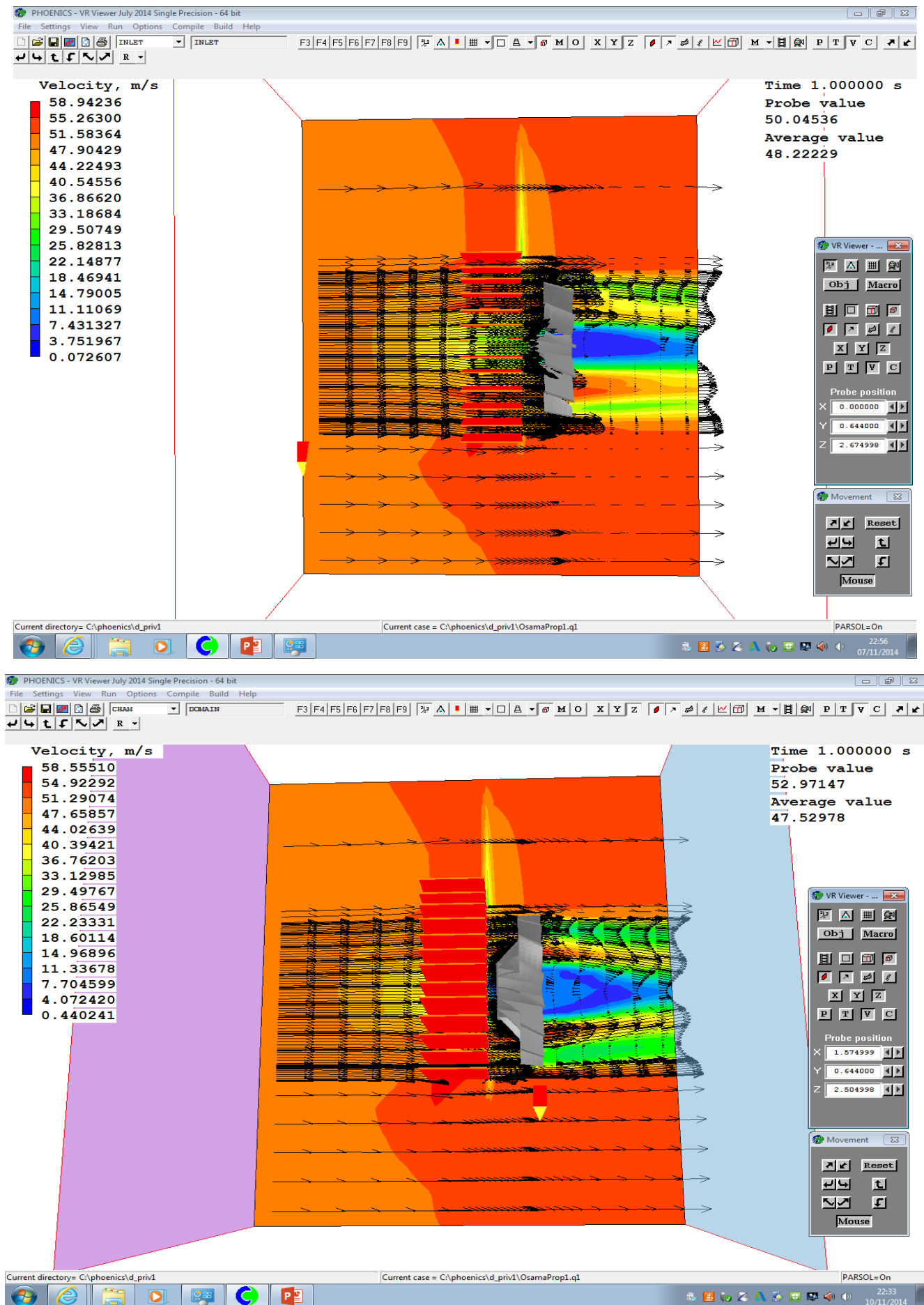
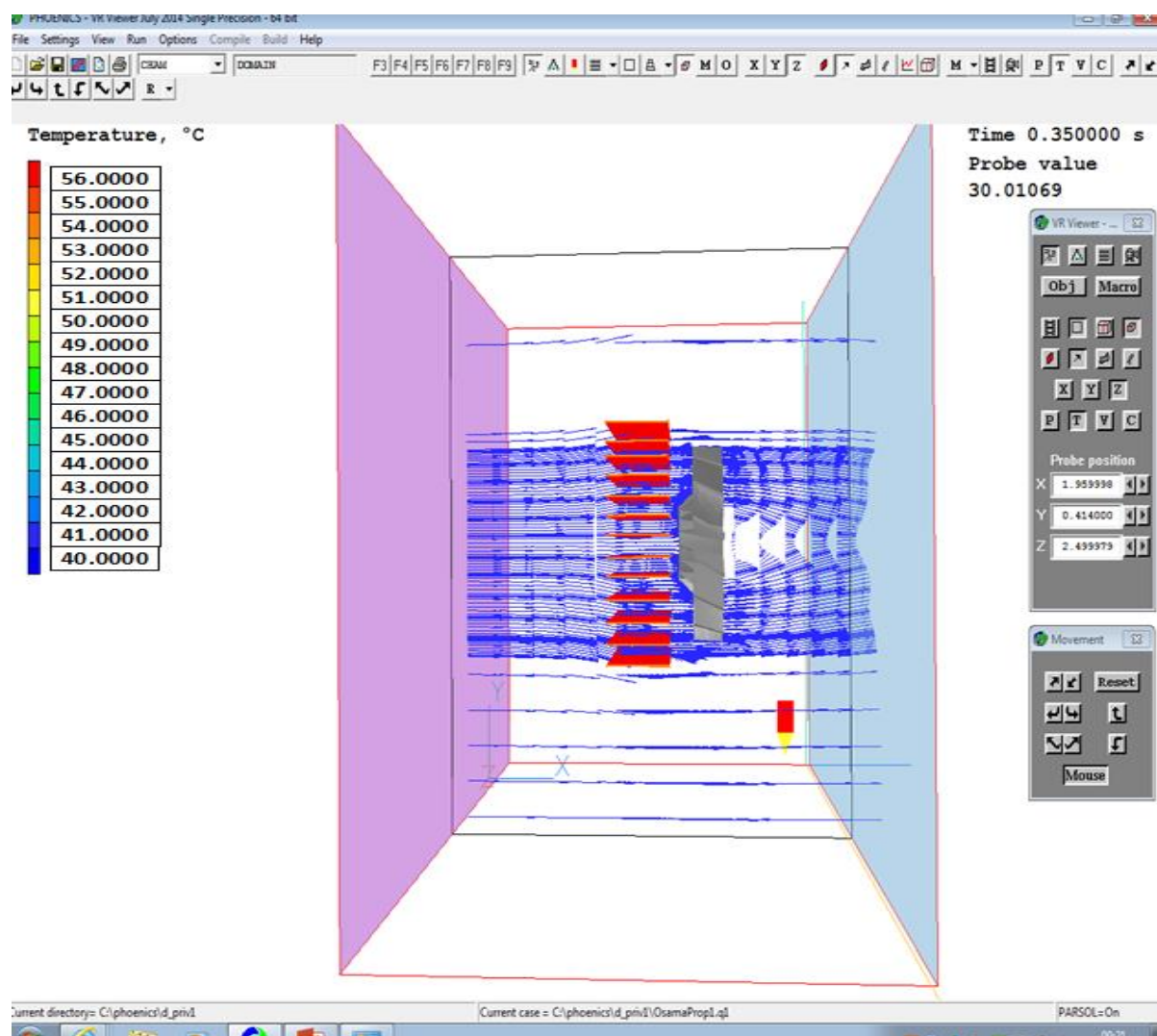
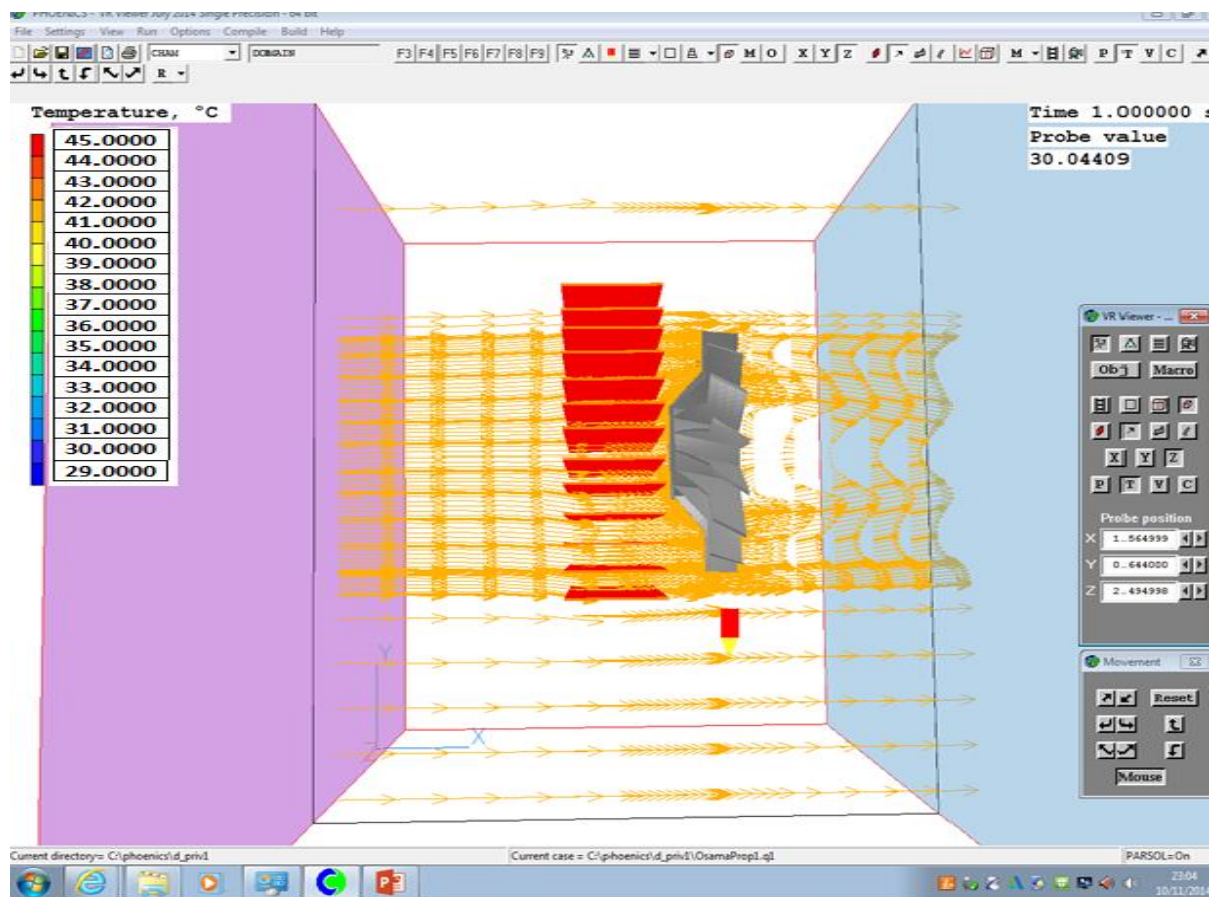


Figure 25: Comparing between the values of velocity for three different temperature (50°C, 70°C, 90°C) fan (33-33)

This suggested that impeller with partially raised cylindrical hub and a nice curvature end finish could actually reduce the effect of velocity fluctuation and turbulence initiation in a region of high moving fluid

domain; followed by an impeller with flat hub and it could be inferred from this study that impeller with sharp hub are not good at handling velocity and turbulence fluctuation in flow domain.



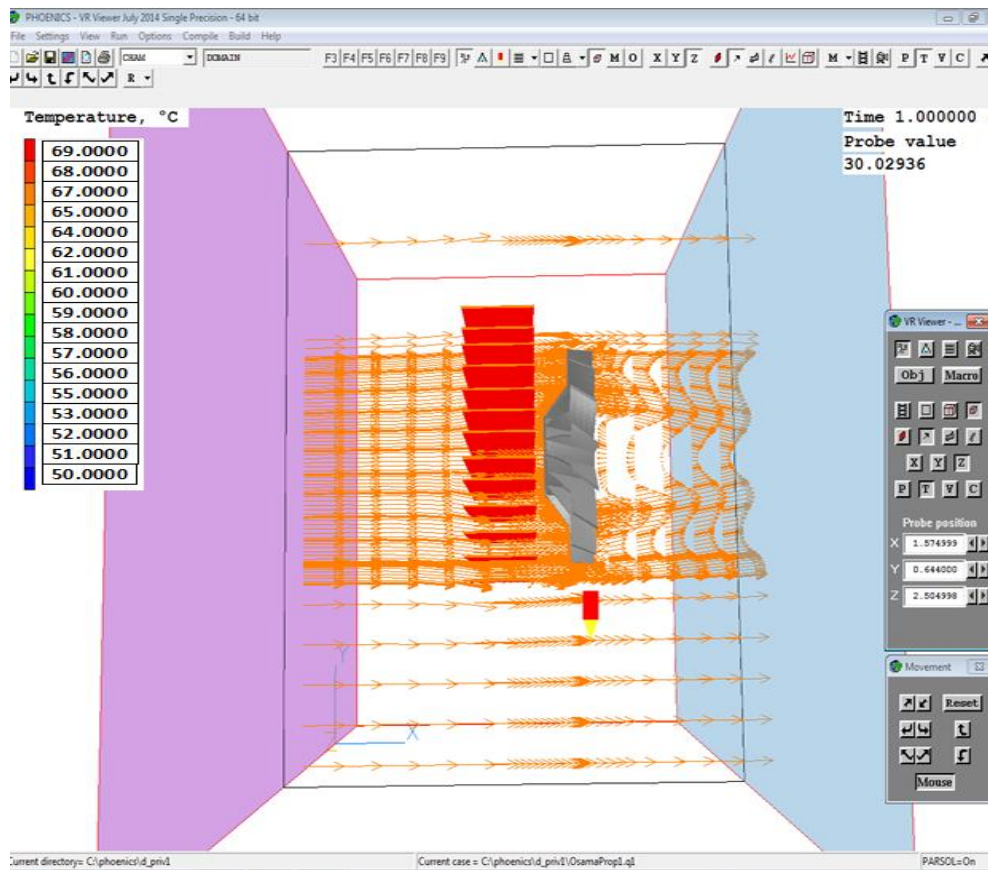
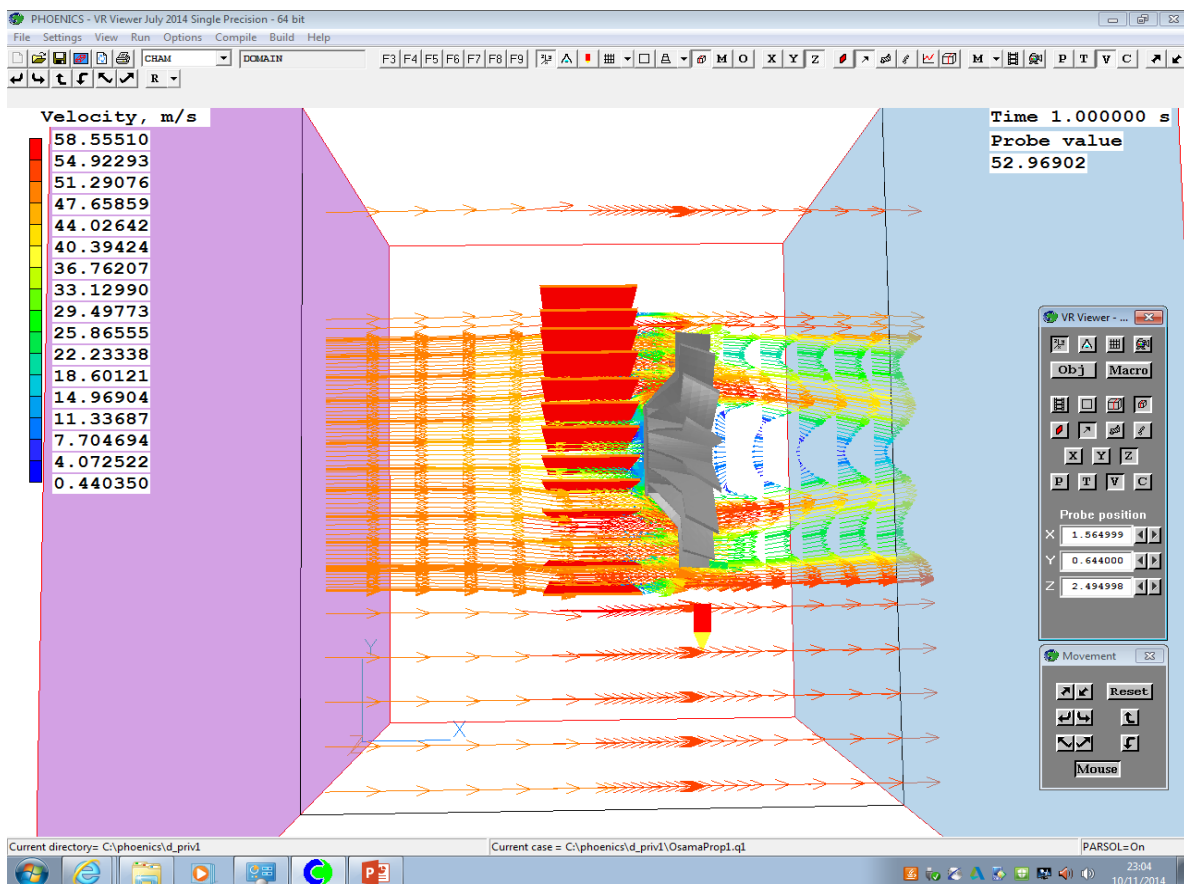


Figure 26: Comparing between the values of temperature for three different temperature (50°C, 70°C, 90°C) fan (33-33)

(The case with the prominent spiral turbine) and its symbol is (33 - 33) It is a fan similar to the original one recommended, except that it has a prominent spiral blade. We note the following:

1. When the coolant temperature is 50°C, the temperature ranges between 29°C- 46°C.
2. When the coolant temperature is 70°C, the temperature ranges between 40°C- 56°C.
3. When the coolant temperature is 90°C, the temperature ranges between 50°C- 69°C.



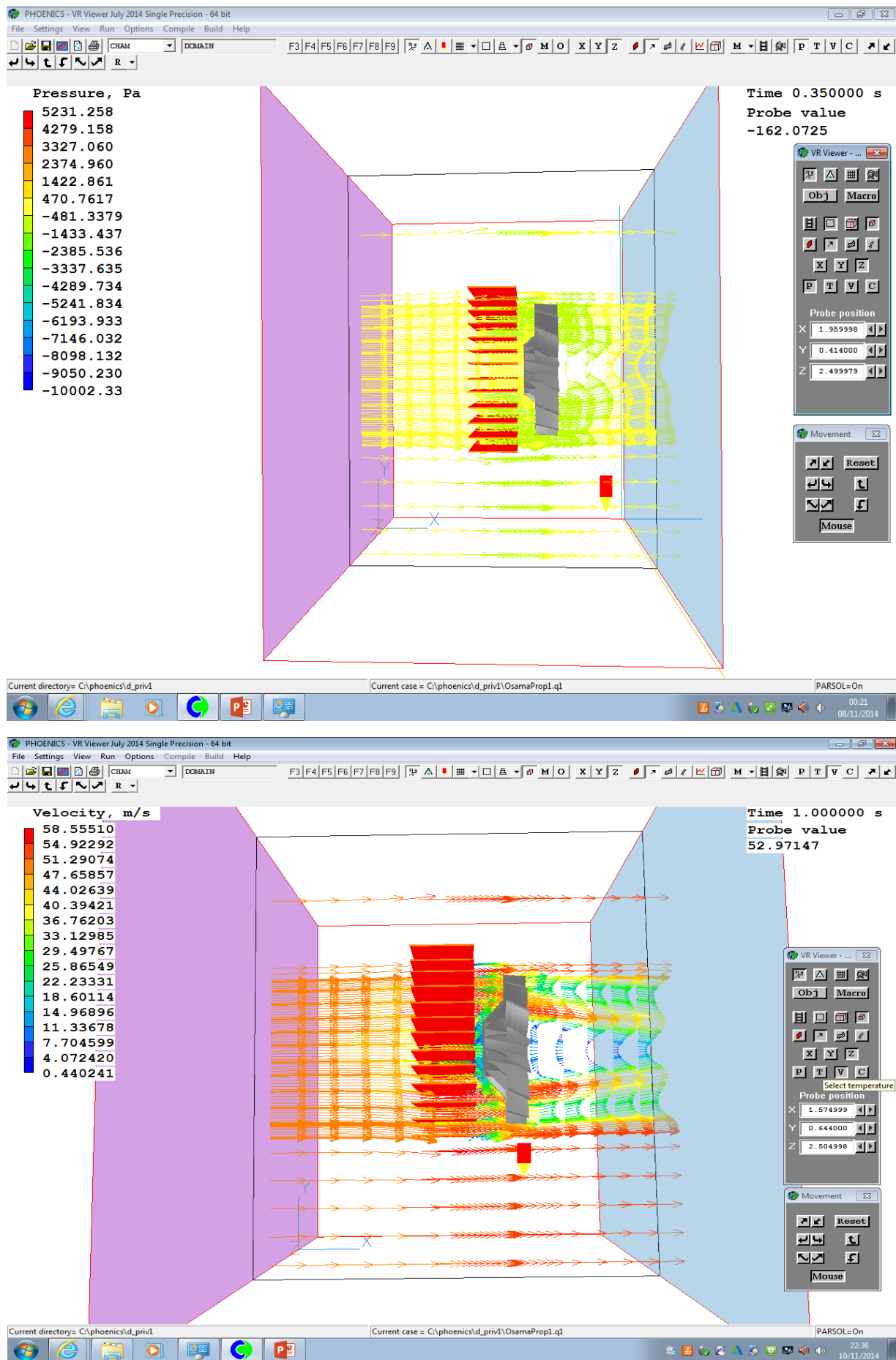


Figure 27: Comparing between the values of pressure for three different temperature (50°C, 70°C, 90°C) fan (33-33).

Dimensional pressure figure for the same flow but divided into two different flow regions; upstream of impeller and downstream of impeller. Fluctuations in the pressure values were observed which could be attributed to the momentum transfer from the rotating impeller. The

higher the RPM, the higher the fluctuation variation as showed by both graphs, though this could actually affect the pressure profile in the flow domain.

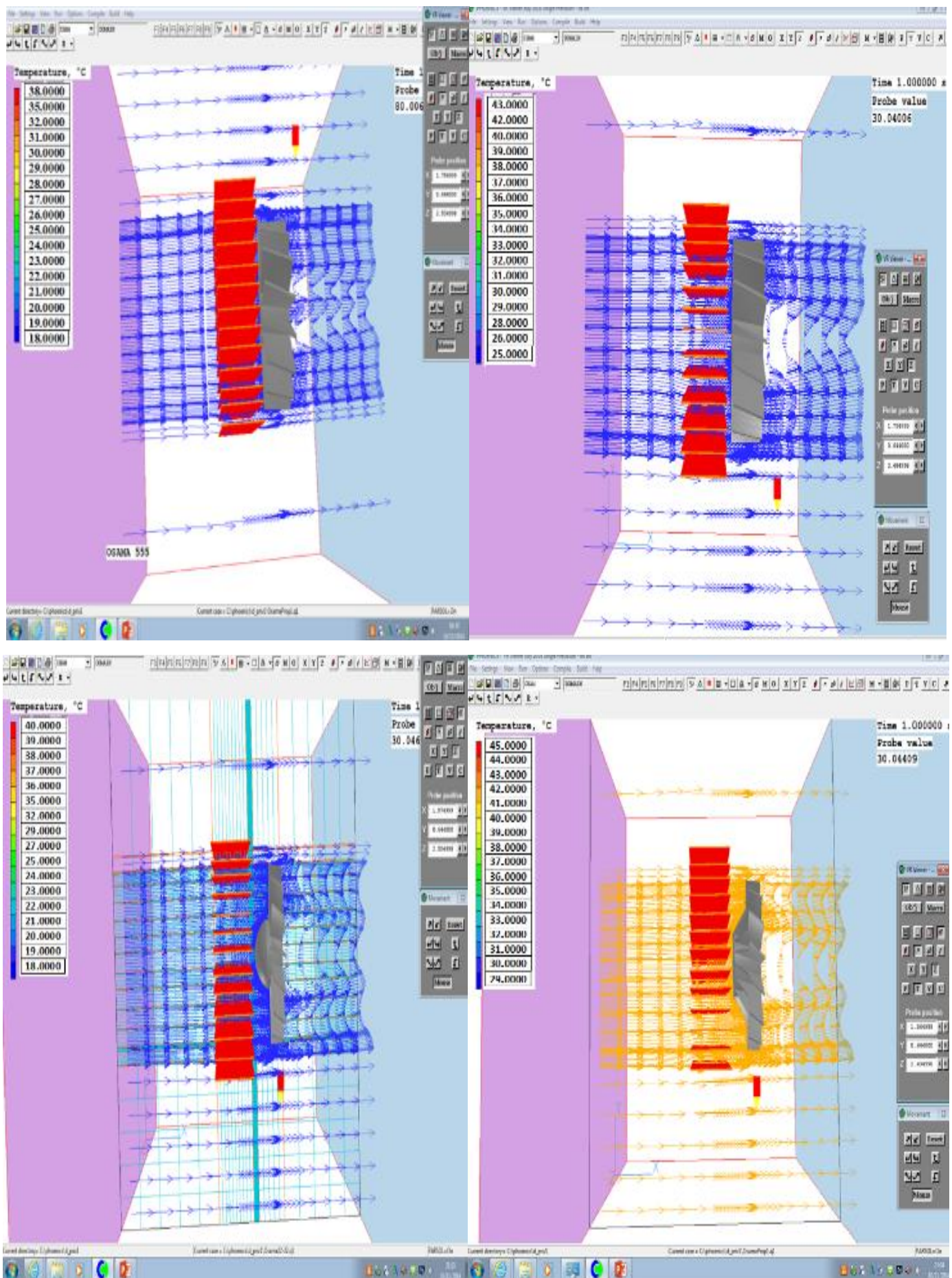
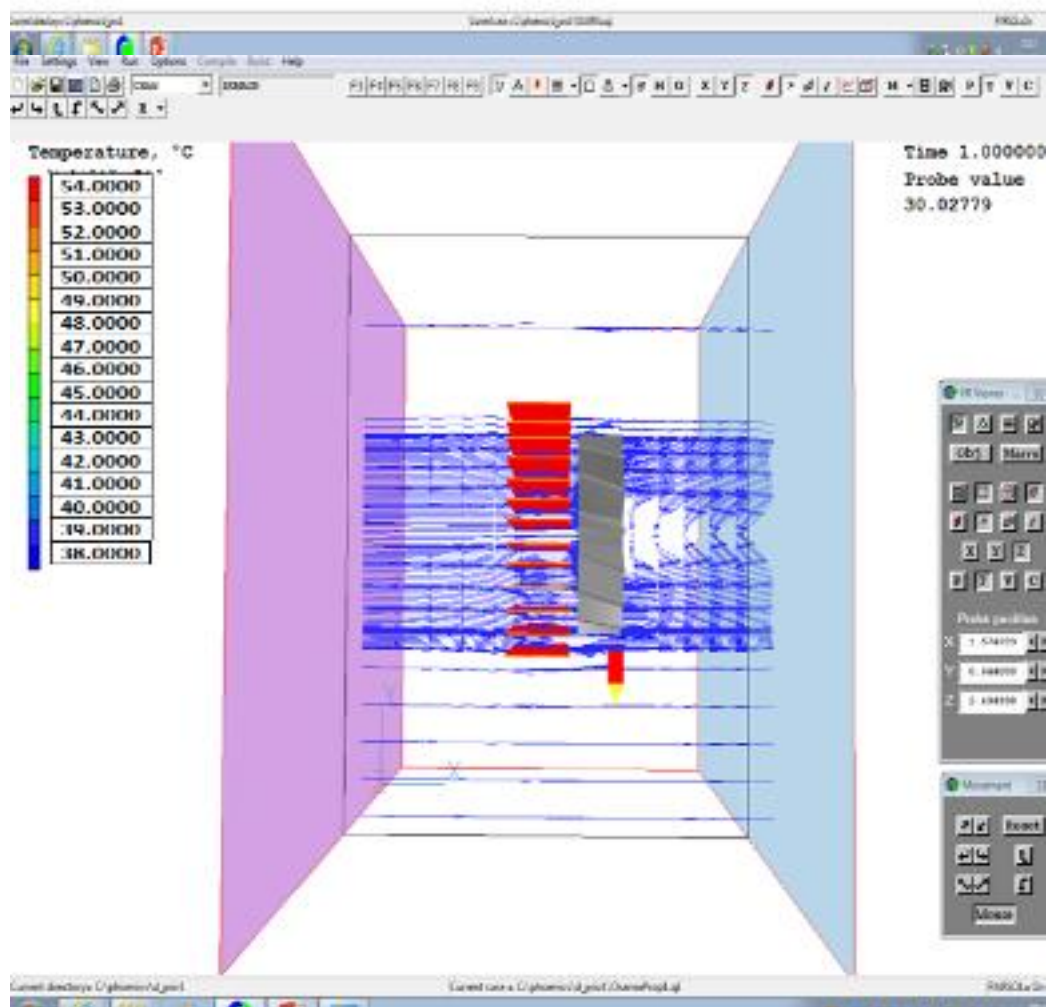
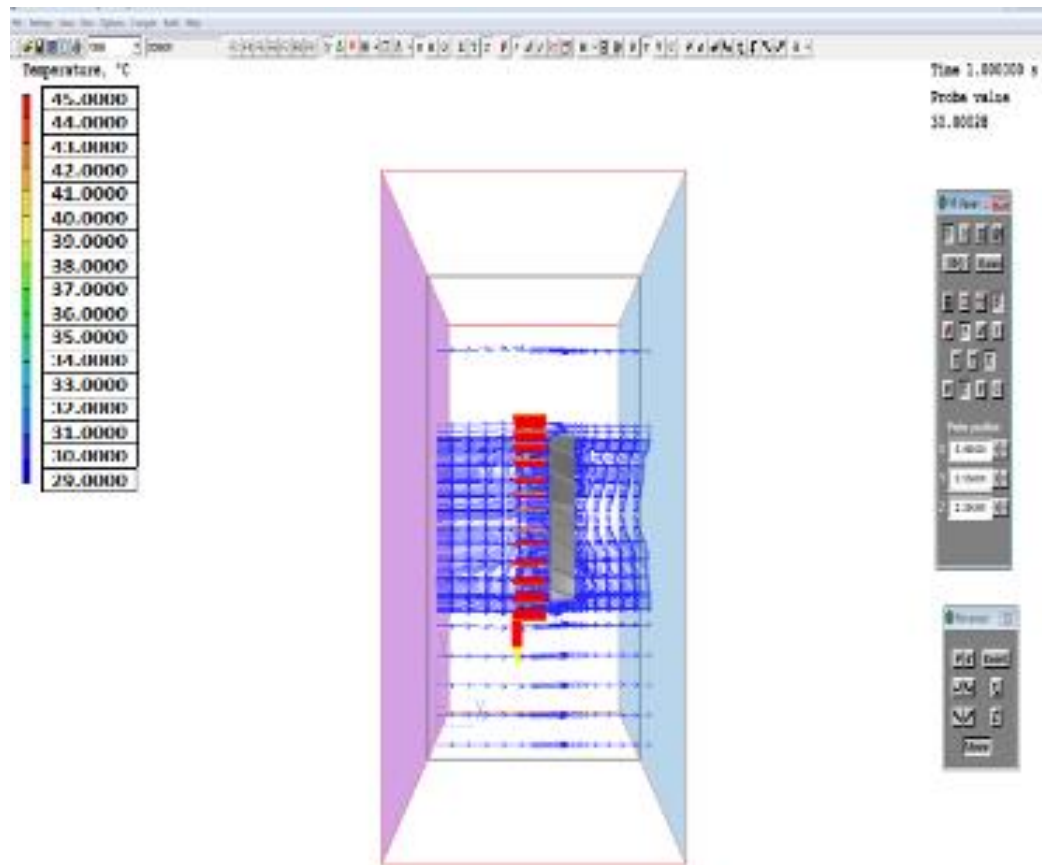


Figure 28: Comparing between the values of temperature for the four fans at temperature 50°C.

We observe through comparison and simulation of the four types at temperature 50°C. The fan with a convex turbine, coded (22 - 22), is the closest to conforming to the original recommended (zero state) and coded

(00 - 00), while the farthest is the fan with a prominent spiral turbine and Symbolized by (33-33).



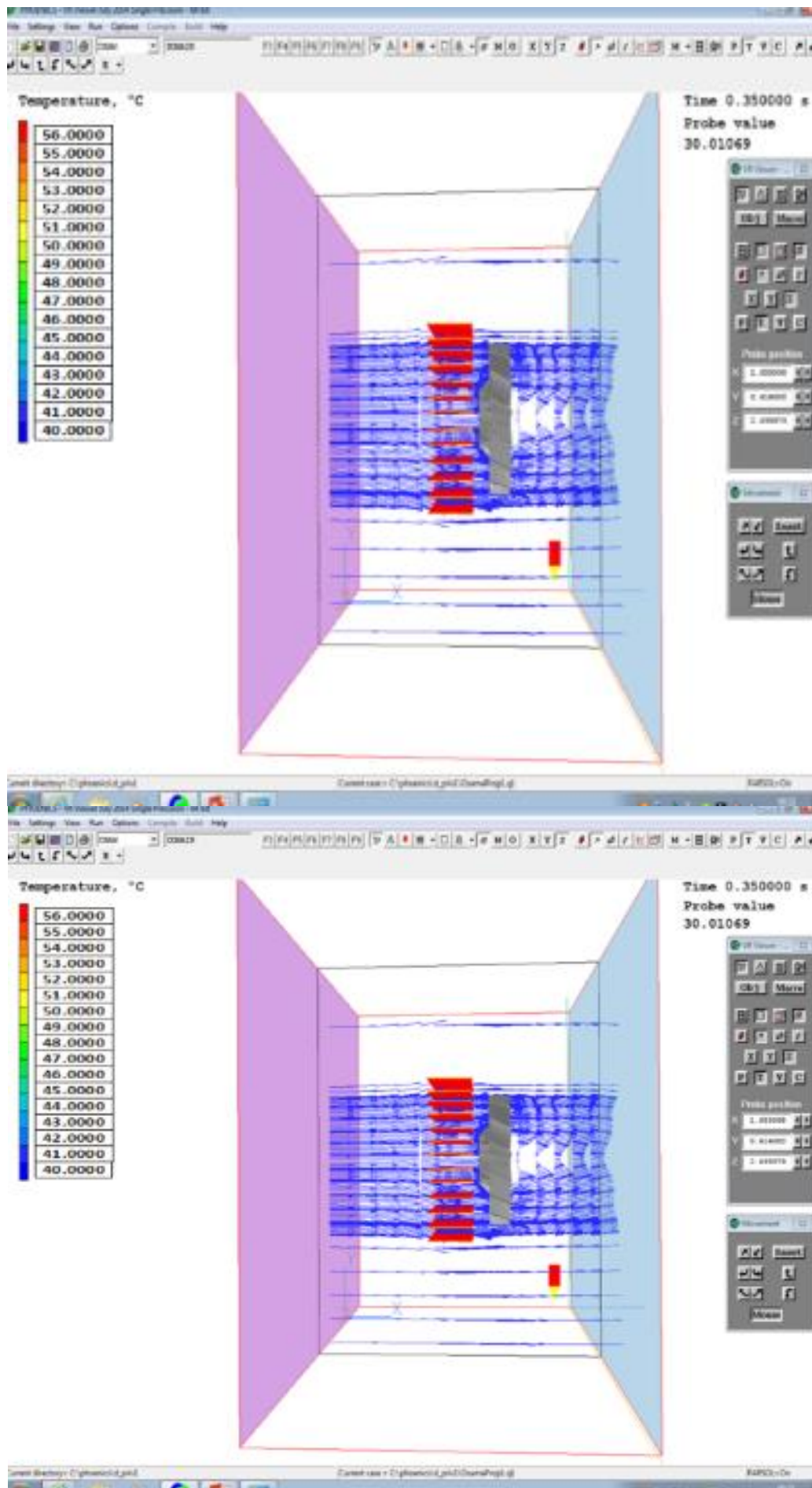


Figure 29: Comparing between the values of temperature for the four fans at temperature 70°C

Also through comparison and simulation of the four types at temperature 70°C. The fan with a convex turbine, coded (22 - 22) is the closest to the

original recommended (zero state) and coded (00 - 00), while the fan with a prominent spiral turbine is farthest away. Symbolized by (33-33).

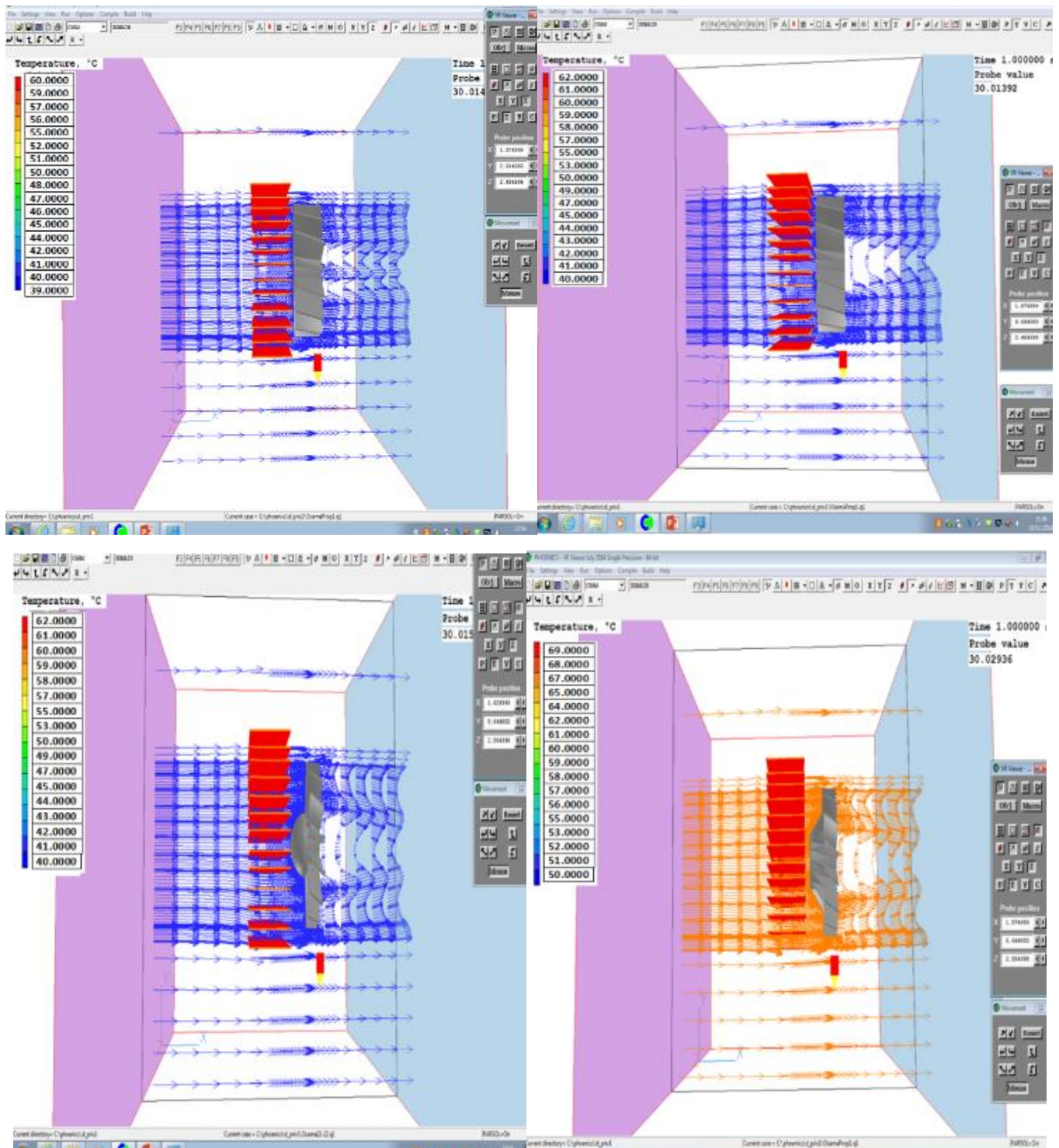


Figure 30: Comparing between the values of temperature for the four fans at temperature 90°C

On the other hand, through comparison and simulation of the four types at temperature 90°C.

The solid fan, symbolized (11 - 11), as well as the convex turbine fan, symbolized (22 - 22), are the closest to the original recommended (zero state) and symbolized (00 - 00). While the furthest and largest is the fan with a prominent spiral turbine, symbolized by (33-33).

5. CONCLUSIONS

CFD analysis was performed for the results were then compared and verified against known theoretical observations and actual experimental data. This study shows that a CFD simulation can be very useful for predicting and understanding the flow distribution from a radiator fan for further research work. The results from the numerical simulations provided an insightful understanding of the behaviour of fluid flow around the different fan blade orientations. Numerical CFD analysis was performed for a fan with a left and right oriented blade. It was discovered that at low rotational speed of impeller of 60 rev/min, [9]. the vortices

formation and flow separation was visible; vortices and turbulence formation were made more pronounced with introduction of more agitation in the form of increase in the rotational speed. The inference drawn is also supported by Euler turbine equation that states that conservation of energy and concept of angular momentum must be satisfied when power is added to or remove from a flow through a rotating blade.

Through the course of this study, the motion of the fluid the transfer of energy from the blade of a rotating impeller; thereby disturbing the laminar stream of the fluid which was maintained by the inertia of the fluid upstream of the impeller. However, since the viscous shear stress of the fluid was not sufficient enough to eliminate the effects of the deviation, turbulence flow was initiated throughout the various models used in this study. Therefore, one of the criteria that determines if a flow will be viscous or turbulence is the ratio of the introduced internal force to the viscous force acting on the particles of the fluid. It could be said that the internal force was not enough to sustain the shear stress introduced by the rotating impeller, thereby leading to turbulence flow which was visible

with the formation of vortices behind the impeller. It is also evidence has shown by of the study that shape factor affects the performance of impellers. It was clear by work done that impellers with cylindrical shaft and spherical hub end could turn turbulence into laminar flow; it was also discovered that impeller with flat surface hub could be better at converting turbulence flow into laminar flow than cylindrical and spherical shaped hub. This conclusion drawn was also supported by work of Shojaeeford, Through comparison of all designs for the third fans with the original fan recommended by car manufacturers, and from the results obtained from simulations of pressure, speed, and temperature, the best design and the one closest to matching is the fan with a convex turbine, while the worst designs are significantly farthest. It is a fan with a prominent spiral turbine.

REFERENCES

- Bredell, J.R., Kroger, D.G., and Thiert, G.D., 2005. Numerical investigation of fan performance in a forced draft air-cooled steam condenser. *Applied Thermal Engineering*, 26, Pp. 846–852.
- Duvenhage, K., Vermeulen, J.A., Meyer, C.J., and Kroger, D.G., 1996. Flow distortions at the fan inlet of forced-draught air-cooled heat exchanger. *Applied Thermal Engineering*, 16 (8/9), Pp. 741–752.
- Elmnifi, M., Al-Tajer, A.M., Al-Asadi, H.A., Al Saker, M., and Habeeb, L.J., 2023. Computational Fluid Dynamics Analysis of a Parabolic Trough Solar Collector for Enhanced Efficiency and Thermal Performance. *Mathematical Modelling of Engineering Problems*, 10 (3).
- Elmnifi, M., Mansur, A.N., Abdul-Ghafoor, Q.J., Alrubaiy, A.A., Mustafa, M.A.S., Khaleel, M., and El-Khozondar, H.J., 2023. Induction Heating for Residential Water Desalination: A Numerical Simulation and Experimental Evaluation. *International Journal of Heat & Technology*, 41 (6).
- Elmnifi, M., Mansur, A.N., Hassan, A.K., Abdullah, A.R., Ayed, S.K., Majdi, H.S., and Habeeb, L.J., 2024. Experimental And Simulation Study For Improving The Solar Cell Efficiency By Using Aluminum Heat Sinks. *Proceedings On Engineering*, 6 (1), Pp. 161-170.
- Elmnifi, M., Moria, H., Elbreki, A.M., and Abdulrazig, O.D., 2021. Possibilities study of using hybrid solar collectors in northeastern libya residential home. *International Journal of Renewable Energy Research*, 11 (2), Pp. 654-661.
- Ho, S.H., Rosario, L., and Rahman, M.M., 2009. Thermal comfort enhancement by using a ceiling fan. *Applied Thermal Engineering*, 29, Pp. 1648–1656.
- Jenkins, P., Elmnifi, M., Younis, A., Emhamed, A., and Alshilmany, M., 2020. Design of a solar absorption cooling system: Case study. *Journal of Power and Energy Engineering*, 8 (01), Pp. 1.
- Kreith, F., 1999. *Fluid Mechanics: A Handbook of Mechanical Engineering*. Bacon Raton, CRC Press.
- Majdi, H.S., Benameur, A., Elmnifi, M., Benkrima, Y., and Al Saker, M., 2023. Modeling and Analysis of Desalination in A Solar Dome Using CFD. *Mathematical Modelling of Engineering Problems*, 10 (5).
- Majdi, H.S., Habeeb, L.J., Elmnifi, M., and Alderoubi, N., 2022. Transient free convection between two horizontal concentric cylinders. *Int. J. Mech. Eng*, 7, 596.
- Mousa Al-Musawi, S.T., Elmnifi, M., Abdulrazig, O.D., Abdullah, A.R., Jassim, L., Majdi, H.S., and Habeeb, L.J., 2024. Water Heating Rate as a Function of Magnetic Field and Electrical Induction Using Solar Energy. *Mathematical Modelling of Engineering Problems*, 11 (2).
- Shams, O.A., Abdulrazig, O.D., Sanousy, S.M., Elmnifi, M., Jassim, L., Shakir, H., and Majdi, L.J.H., 2023. Modeling of Electricity Generation Using Smart Piezoelectric-Materials. *Journal homepage: <http://ieta.org/journals/ijht>*, 41 (6), Pp. 1655-1660.
- Shojaeeford, M.H., Tahani, M., Eghaghi, M.B., and Beglari, M., 2011. Numerical Study of the effect of some geometric characteristics of a centrifugal pump impeller that pump a viscous fluid. *Journal of Computer & Fluid* (60): Science Direct Series. Available at: www.elsevier.com/locate/complfluid (Accessed on: 19 Nov, 2023)
- Stafford, J., Walsh, E., and Egan, V., 2010. Local heat transfer performance and exit flow characteristics of a miniature axial fan. *International Journal of Heat and Fluid Flow*, 31, Pp. 952–960.
- Stafford, J., Walsh, E., Egan, V., and Grimes, R., 2010. Flat plate heat transfer with impinging axial fan flows. *International Journal of Heat and Mass Transfer*, 53, Pp. 5629–5638.
- Stokse, W.S., 2004. Centrifugal fan impeller mechanism of machines. A journal of Mechanism and Machine Theory: Science Direct Series, 39. Available at: www.elsevier.com/locate/mechanismmachinetheory (Accessed: March 3, 2023).
- Twizell, E.H., and Bright, N.J., 1981. Numerical modelling of fan performance. *Applied Mathematical Modelling*, Pp. 5.
- Wang, F.J., 2004. *Computational fluid dynamics-principle and application in CFD software* (Beijing: Tsinghua University Press).
- Yu, F.W., and Chan, K.T., 2007. Modelling of a condenser-fan control for an air-cooled centrifugal chiller. *Applied Energy*, 84, Pp. 1117–1135.

