Periodicals of Engineering and Natural Sciences ISSN 2303-4521
Vol. 8, No. 2, June 2020, pp.773-783

Numerical assessment of heat sink for pressure sensor connections

Ali Hussein Abdulkarim[1], Mustafa Wahby Kanbar JABER[2], Eyub Canli[3], Ayad A. Kakei[4]

[1]Kirkuk University
[2] Ministry of higher education
[3]Selcuk University
[4]Kirkuk University

Abstract

Pressure sensors, converting pressure force to electrical outputs such as 4-20 mA or 0-10 V, are used in a vast variety of areas while being facing numerous challenging thermal conditions. A common way is to design a heat sink for establishing natural convection cooling to protect the sensor. This work assesses a heat sink design and conveys its performance as a heat sink for an application interval. Special orientation as well as design geometry is introduced. Computational fluid dynamics were utilized for evaluation and assessment. A core region of heat transfer was identified. Natural convection wake boundaries were detected. It is concluded that the design can successfully protect the pressure sensor at the pressure tap. Future projections and aspects are also described in the paper.

Keywords: CFD; heat sink; natural convection; pressure sensor

Corresponding Author: Ali Hussein Abdulkarim

Third Author Name, Abdulkarim
Departement, Mechanical engineering
University of Kirkuk,
Address. Kirkuk-İRAQ
E-mail: dr.ali@uokirkuk.edu.iq

1.Introduction

An important part of engineering work is to measure. Pressure measurement is one of the key elements and pressure is sometimes, referred as a primitive variable. While so important, its measurement can be challenging. One major problem is the temperature of the medium that can deal damage to pressure sensor. Heat sink utilizing natural thermal convection is frequently used to overcome this problem. However, by its characteristics, there are too much variable that change heat sink thermal performance. Some studies that can exemplify the number of design variables are provided below.
CFD is very common in evaluating heat sink and natural convection heat transfer in the literature. Ramzan et al. investigated a moving thin needle that can be considered as a basic heat sink geometry [1]. By means of conservation laws applied to their work, they present effects of parameters such as Prandtl number or Schmidt number. Duan et al. present a heat sink that have rectangular profile fins as in this work [2]. They searched effects of an elliptical geometry on the wall surfaces between fins in terms of pressure drop and CFD. Carvalhaes-Dias et al. report a very interesting application of heat sinks having rectangular profile fins by using them in soil as heat dissipating medium [3]. Although the work is an experimental one, orientation of the heat sink and surface area enhancement are very inspiring in terms of engineering. This paper is also provides a chance to have an idea of pure conduction performance of heat sinks, which is important while evaluating conjugate heat transfer in the presence of flow over heat sink. The work of Wang et al. has common sides with the present paper by horizontal orientation of a circular heat sink, while utilizing CFD and considering buoyancy forces, though they mostly focus on internal flow [4]. Kharangate et al. show that CFD is a reliable tool for investigating heat sinks by their work combining CFD and experimental means [5]. They identified flow dead zones between heat sink fins which emphasize importance of the heat sink orientation. Saravanakumar and Kumar use rectangular profile fins on a heat sink and analyze its thermal performance by experimental and CFD means [6]. They also use temperature distribution of CFD results as a

tool for evaluating the heat sink, similar to the present paper. When CFD is used, various approaches can be done for same cases and paper of Jeng and Tzeng is a good example for this [7]. Authors applied a condition to model heat sinks easily by assuming porous media; which is rarely encountered in literature. The work of Chen et al. can be regarded as a good example for the importance of the orientation of heat sinks [8]. Hasan et al. tried phase changing materials as heat dissipation media for heat sinks [9]. They report significant changes in temperature distribution. A methodology for considering heat sinks is proposed by Ong et al. [10]. This paper is an important reference for preliminary design of heat sinks. Saadah et al. provides an important application of heat sinks having rectangular profile fins and an important instance of heat sink orientation [11]. This work is also contains a good example of conjugate heat transfer since the heat source acts on a limited area. CFD investigations of heat sinks and result presentation with contour images similar to the present work are very common and the paper of Alfellag et al. is in this category [12]. For the mesh structure of CFD heat sink studies, structured and unstructured meshes can be used. Hoi et al. present fractal mesh structure with and without structured mesh elements [13]. They manage to have a detailed variable field while mesh element transitions are insignificant. Hsu and Huang presented a new heat sink design and compared it to a porous heat sink design via CFD [14]. They find comparable performance characteristics between the two heat sinks. The study is similar to the present paper in terms of natural convection heat transfer.

In this work, CFD is used in order to assess thermal performance of a heat sink for pressure sensor connection. Complete heat sink geometry is modeled and air is used for the fluid that heat dissipates. Only natural convection is analyzed and radiation is excluded. By the complete geometry, field application is assessed. The connection orientation is also considered. The work provides an open source for vast variety of application and research.

2.Material and methods

Solid model of the heat sink is given in Figure 1. Although this rendered figure gives an idea of the heat sink, its dimensions are illustrated by Figure 2. Given dimensions are common for a variety of pressure measurement applications.

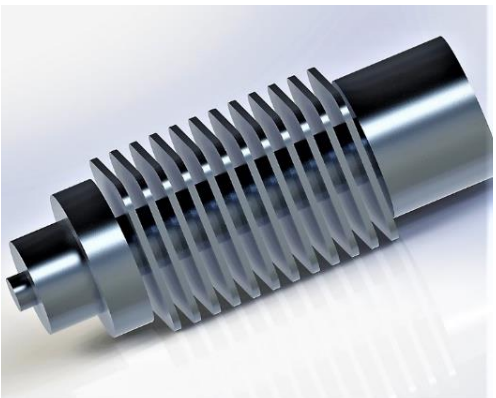


Figure 1. Rendered solid model of the heat sink

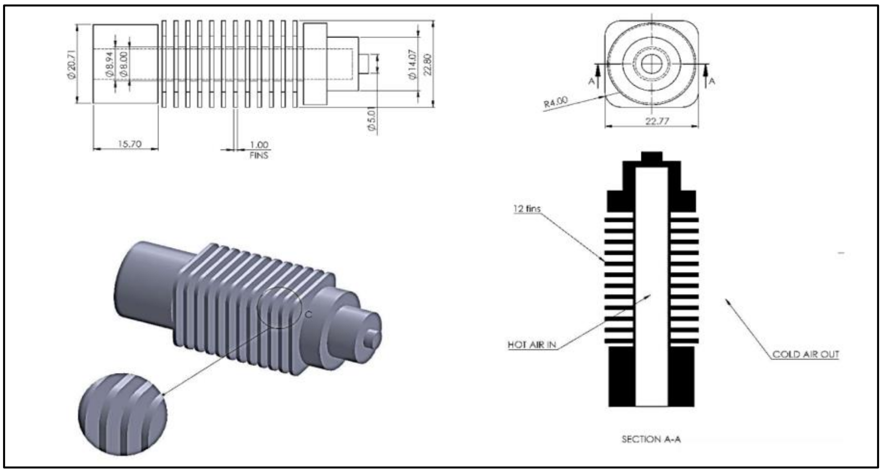
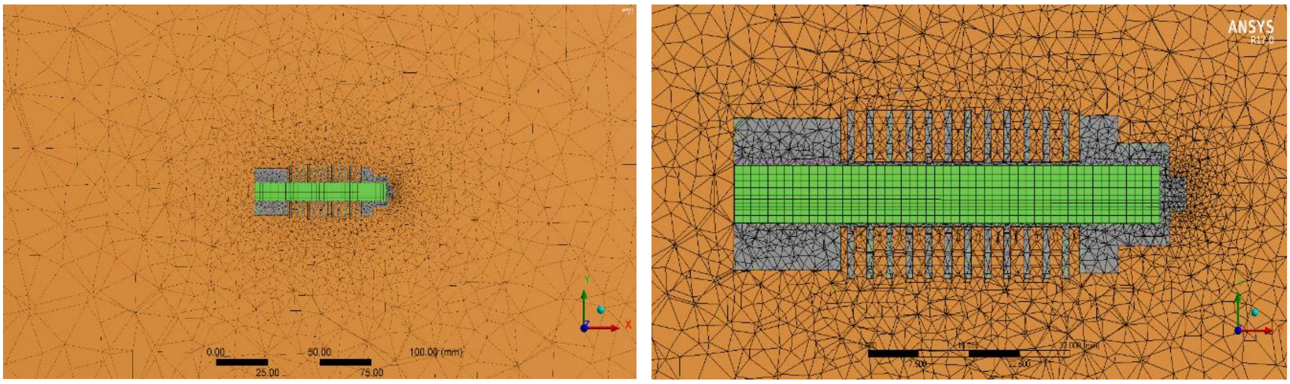


Figure 2. Dimensions of the heat sink

The particular heat sink design for protecting pressure sensors is analyzed via CFD. Ansys Fluent is used as software package. Heat sink material is chosen as aluminum having 2719 kg/m^3 density, 202.4 W/mK thermal conductivity, 871 J/kgK specific heat. For fluid domain, ideal-gas density relations are used. 1006.43 J/kgK specific heat is given as input for the fluid. Thermal conductivity of the fluid is chosen as 0.0242 W/mK. Dynamic viscosity is set to 1.7894x10^{-5}. Both solid and fluid domains are modeled. The mesh structure of the model is given in Figure 3. Triangular mesh elements are used due to complex geometry of the models. This type of elements are usually preferred in solid mechanics, however, when complex geometries make structured mesh accomplishment impossible, triangular mesh elements can be used with special treatment in the solver. Especially interpolation schemes change. Total of 222,177 mesh elements constitute the calculation domain with 52063 nodes. These numbers may be seemed low but the growing towards ambient makes them possible.

Solver is set to transient solution with gravity is activated in order to have a natural convection. 0.01 second time steps are used for transient analyses. PRESTO algorithm is selected for pressure-velocity coupling. Eddy viscosity or turbulent viscosity concept is utilized for turbulence modeling. SST k-omega turbulence model is preferred because it offers advantages of standard k-omega turbulence model near viscous sub-layer and advantages of standard k-epsilon turbulence model away from the wall.



(**a**) (**b**)

Figure 3. Mesh structure of the solid-fluid model. (a) shows growing of the mesh towards ambient (b) shows interface between solid and fluid domains

First type thermal boundary condition is used for contact regions which is constant temperature boundary condition and surface temperature is set to 350 K. For variable profiles of the iteration process in finite volume method, first order upwinding is selected as the scheme for all variables.

3.Results

All given results are instantaneous results that are subjected to spatial averaging for limited ranges in order to provide smoother graphics that are easier to read. The time instance for the results are selected in a way that iteration cycle for each time step becomes periodic and operation are close to steady state.

A rendered 3D temperature figure of the heat sink is given in Figure 4. Orientation of the heat sink and dissipation of heat towards environment are visible via temperature volume rendering. Gravity acts in -y direction and therefore Buoyancy forces make fluid to flow towards +y direction. The gradient of density, starting from the wall surface and developing in the proximity of the heat sink is seen in Figure 5. It is visible that natural convection has a major route on y axis. Still, density gradient continues for a distance at –y direction and this can be done by a conjugate heat transfer mechanism.

In order to evaluate density changes and temperature distribution as a factor on them, Figure 6 is given to illustrate the temperature distribution. It is interesting to see that general trends of the figures for density gradient distribution and temperature distribution are not completely overlap. This is only be explained by the motion of the fluid; which causes differences in pressure, velocity and turbulence properties.

The vector field in the close proximity of the heat sink is given in Figure 7. This figure reveals a slow motion between heat sink fins which is expected due to no slip condition on the wall and narrow channels formed by the fin walls. However, the consistent motion driven by the buoyancy forces is visible. Flow accelerates as the location continues towards +y direction, causing a low-pressure field and sucks more fluid mass, yielding a potential for continuous flow.

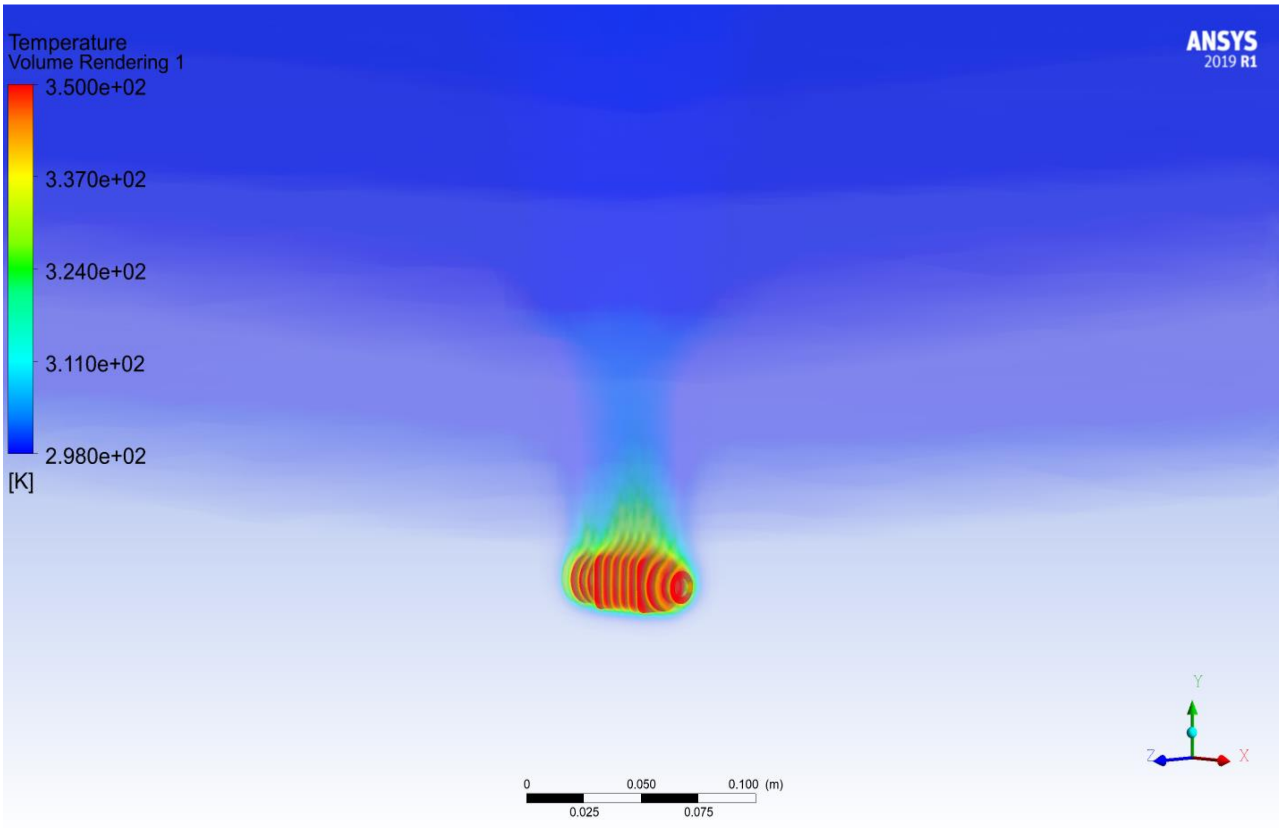


Figure 4. Temperature volume distribution during natural convection cooling of the heat sink

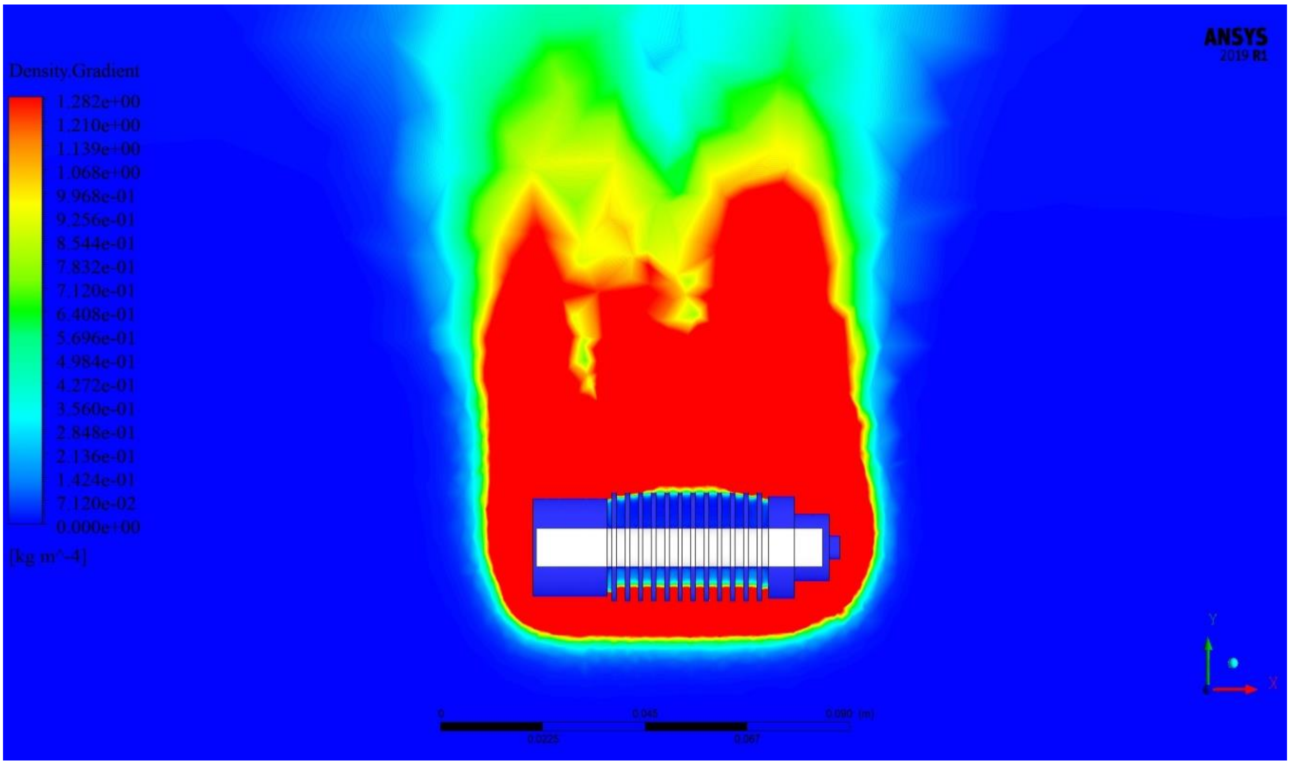


Figure 5. Density gradient of the fluid between wall and ambient

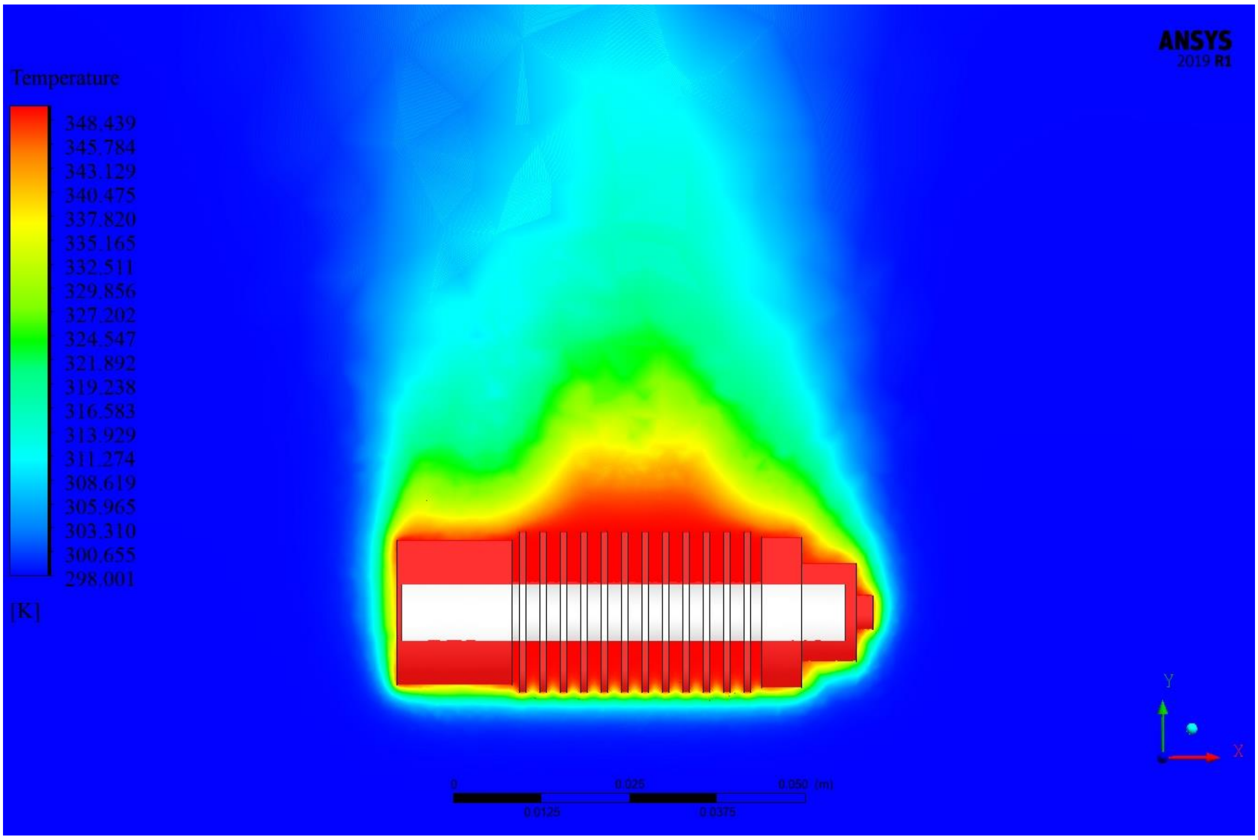


Figure 6. Temperature distribution between fluid and solid domains

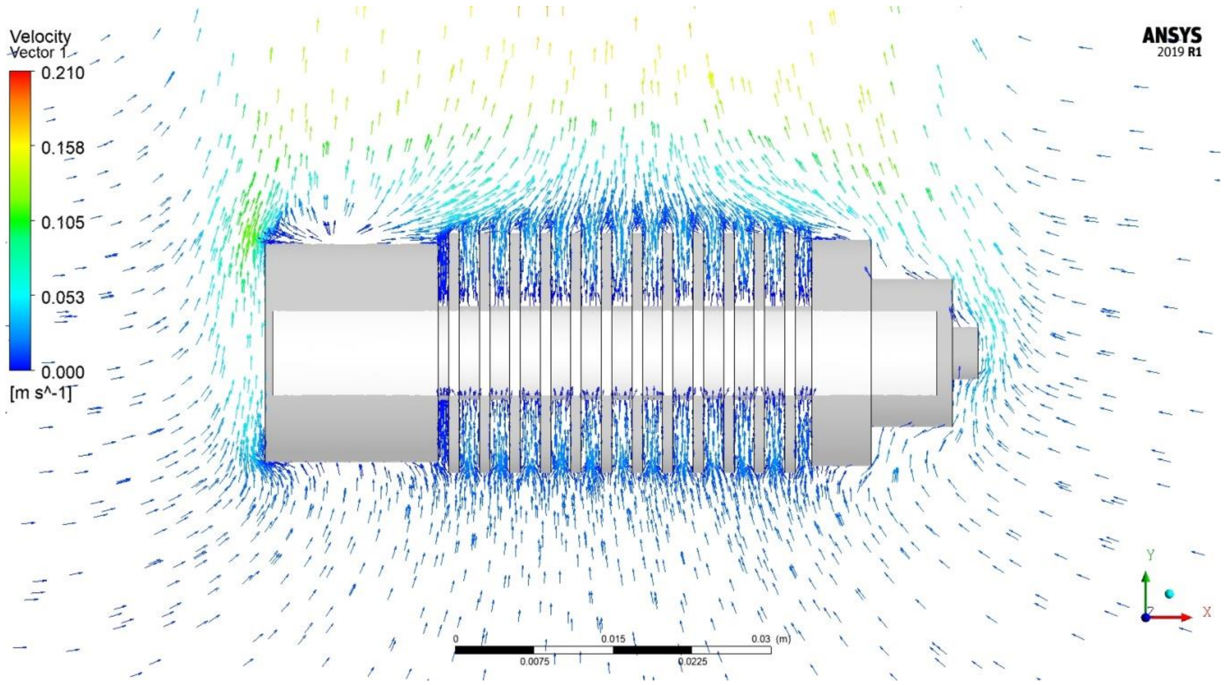


Figure 7. Vector field in the close proximity of the heat sink

While velocity vectors show direction of the flow and their magnitudes till a point, it is hard to make a concise conclusion on the flow field in terms of velocity. Therefore, Figure 8 is given for velocity distribution downstream of the heat sink. This figure identifies an interesting phenomenon about flow acceleration right after the heat sink. After solid boundaries are passed, flow accelerates due to the density gradient causing the buoyancy force and constitutes a high velocity core. Maximum velocity can be four times the velocity of the surrounding field.

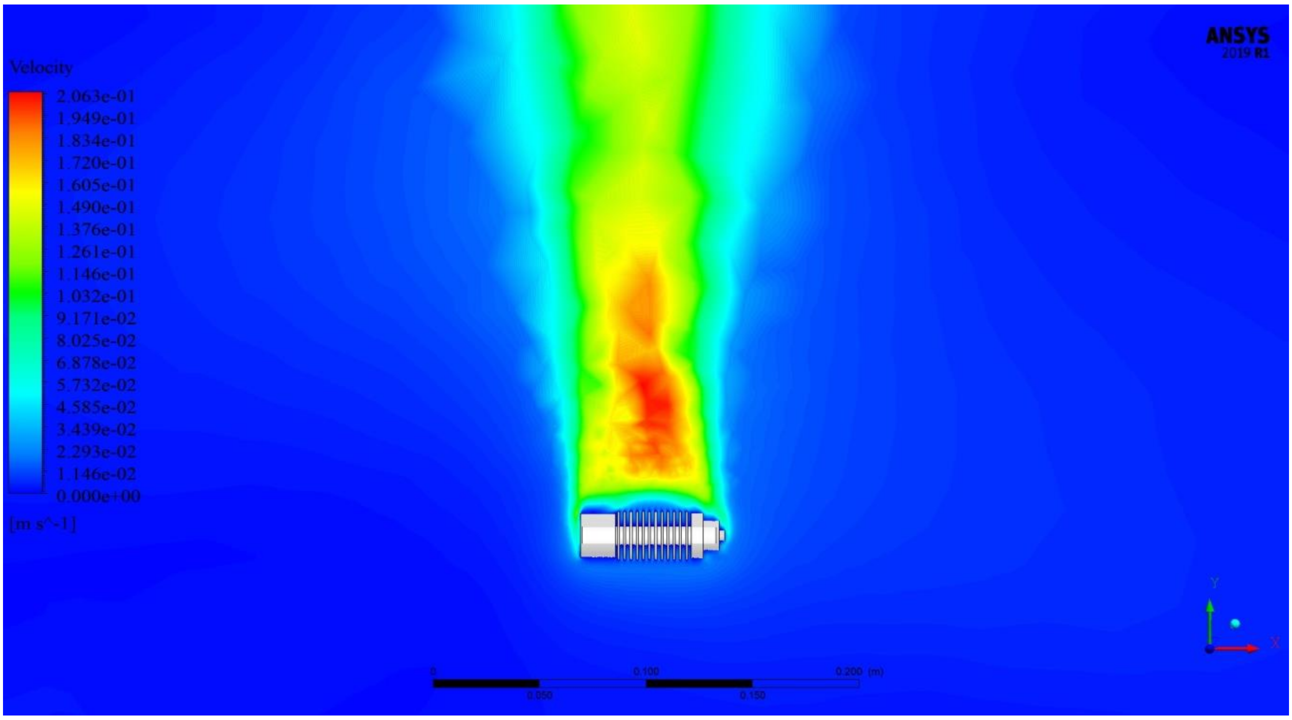


Figure 8. Velocity magnitude distribution

The last thing about flow field to be considered in respect of natural convection is turbulence. Turbulence draws energy from mean kinetic energy of the flow by random local velocity fluctuations. Those fluctuations cause to grow turbulent stresses that lower the mean flow kinetic energy and convert that energy to turbulent kinetic energy. Subsequently, turbulent kinetic energy is dissipated as molecular kinetic energy or heat in other words, due to molecular viscosity. This energy cascade is responsible of minor changes of natural convection structure. In this work, turbulence properties are evaluated by means of turbulent kinetic energy and turbulent viscosity that are given in Figure 9 and Figure 10 respectively; as magnitude value distributions.

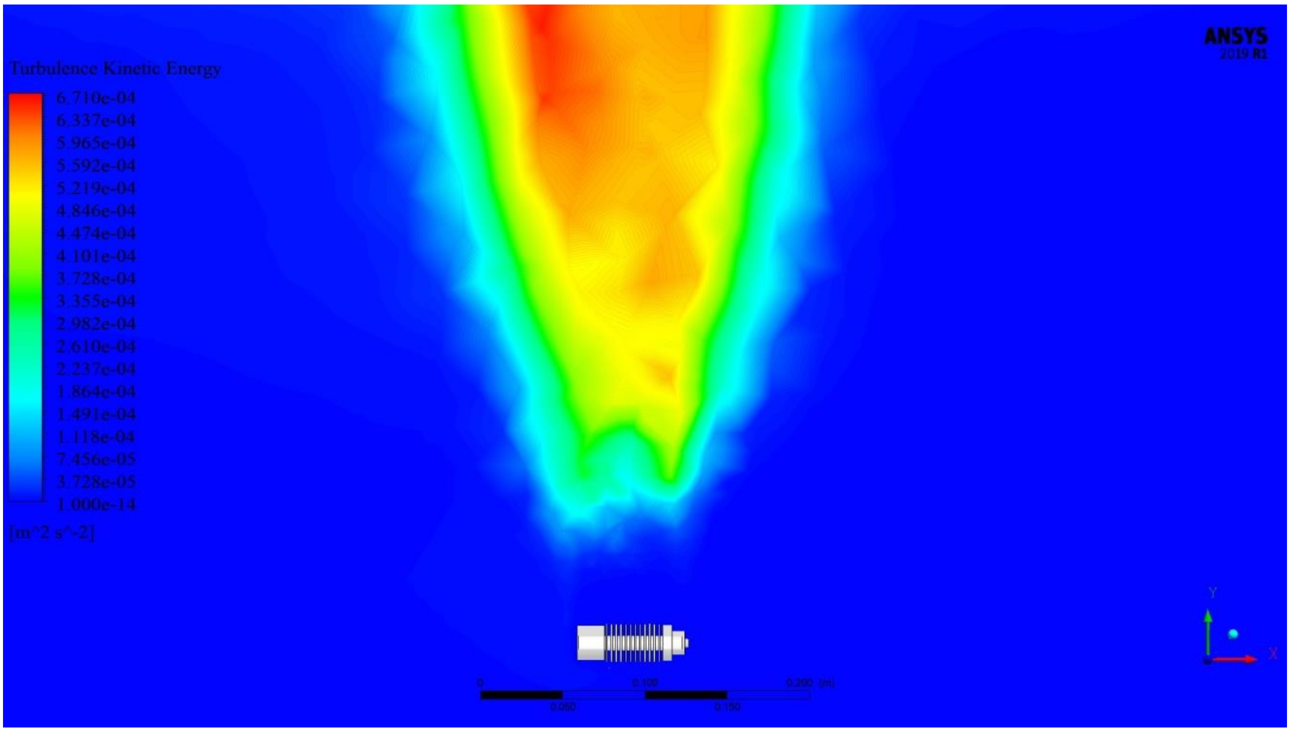


Figure 9. Turbulent kinetic energy distribution

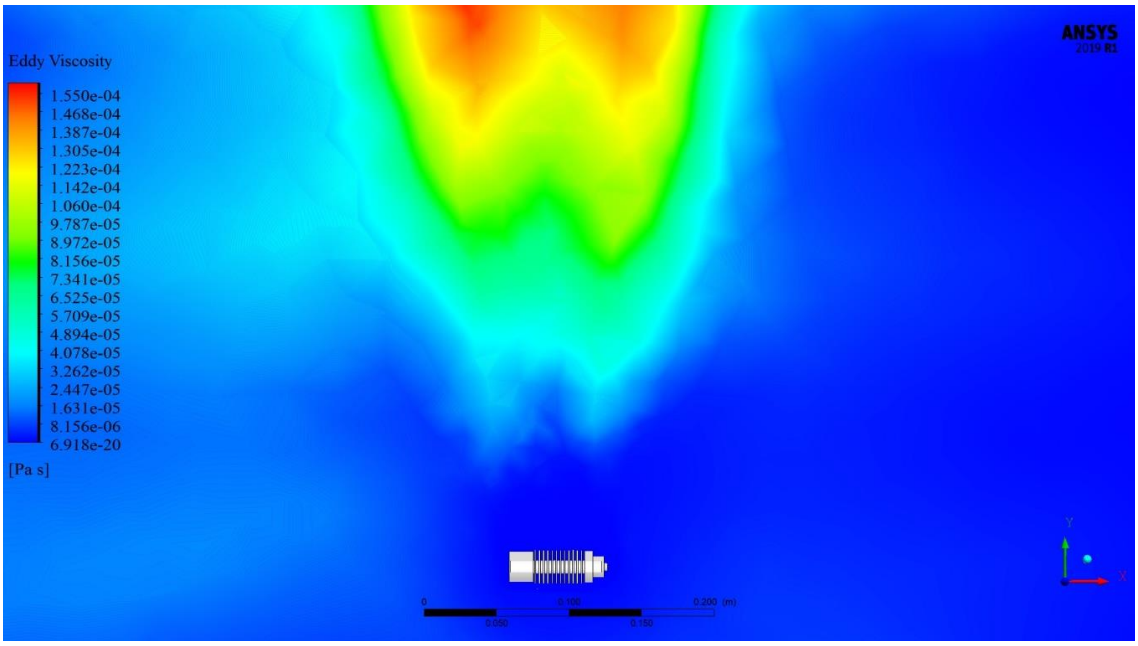


Figure 10. Turbulent viscosity distribution.

Evaluating Figure 9 and 10 together, it is concluded that turbulence generation and destruction are almost equal in the regions close to the heat sink. After almost 10 width lengths towards downstream of the heat sink, turbulent viscosity starts to have bigger values causing more energy is drawn from mean flow. This effect adds on the effect of molecular viscosity and flow decelerates. So flow field grows in width in order to satisfy continuity. Also growing turbulent viscosity is a sign of approaching to developed flow conditions. However, for natural convection, which is driven by density gradient and buoyancy forces, developed flow is hard to be mentioned if not possible.

The frequency of turbulence eddies, which is a time scale, has very high values near wall surfaces due to the damping effect of the wall (Figure 11). While it has very small role on natural convection mechanism, contours in the figure can be used to have an idea about the viscous sub layer. Especially between fins, it can be identified that viscous sub-layer are not filling the gap, so that the damping effect on eddies are not uniform between fins.

The close proximity of the heat sink solid body and surfaces also reveals vorticity due to the presence of boundaries (Figure 12). High magnitude small size structures can be identified at the circular edges of the geometry.

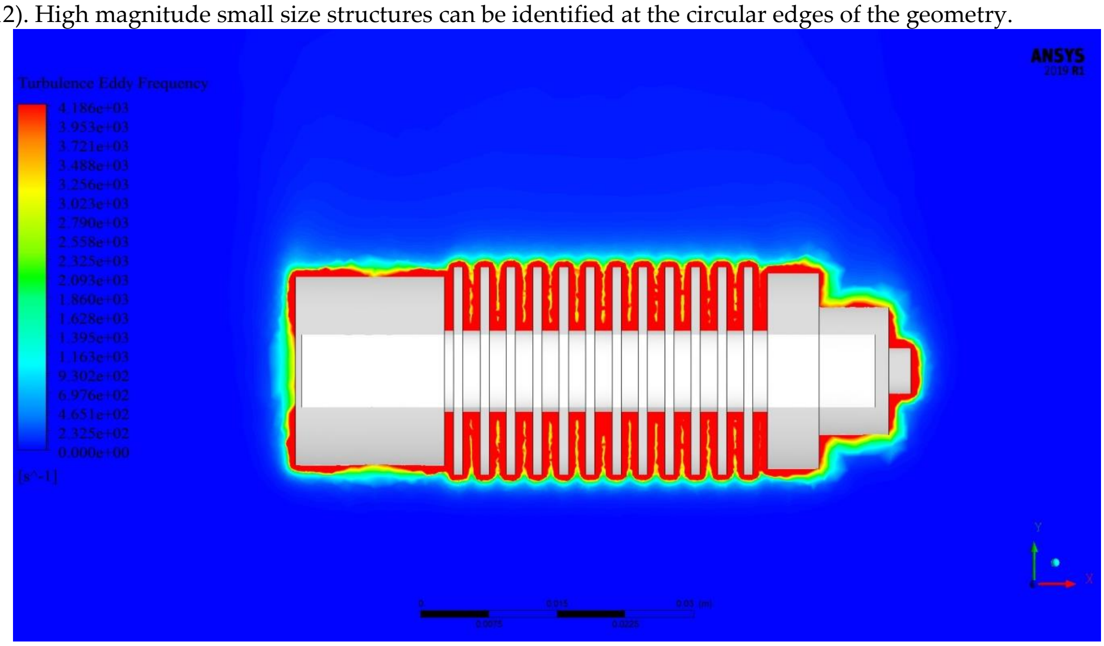


Figure 11. Turbulence eddy frequency.

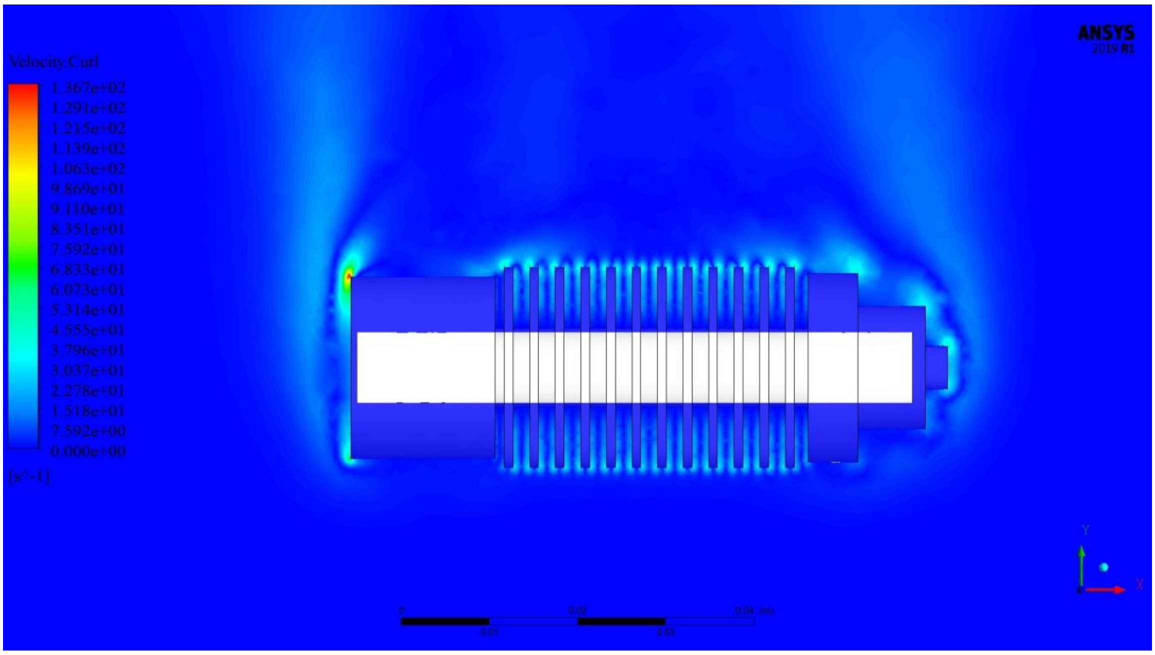


Figure 12. Vorticity due to solid boundaries

While the main flow orientation is on y axis, it is beneficial to view x axis change of the parameters in order to see the variable change near the heat sink. Figure 13 is given for density gradient in horizontal direction away from the heat sink. The peak point of the figure is very close to the heat sink, indicating intense changes in x-direction. Figure 14 is presented for turbulent viscosity change in horizontal direction away from the heat sink. The range of the figure shows a very small value of turbulent viscosity comparing to molecular viscosity. It is expected since the main motion is on the direction of a different axis. Still the general trend of turbulent viscosity, that its value increases away from the solid boundaries, can be seen in this figure also. Temperature change in horizontal direction away from the heat sink is given in Figure 15. A sharp decrease in temperature shows that heat is dissipated in another direction by means of the natural convection and conduction in fluid is very weak. It should not be forgotten that fluid sucked towards the heat sink in x-direction.

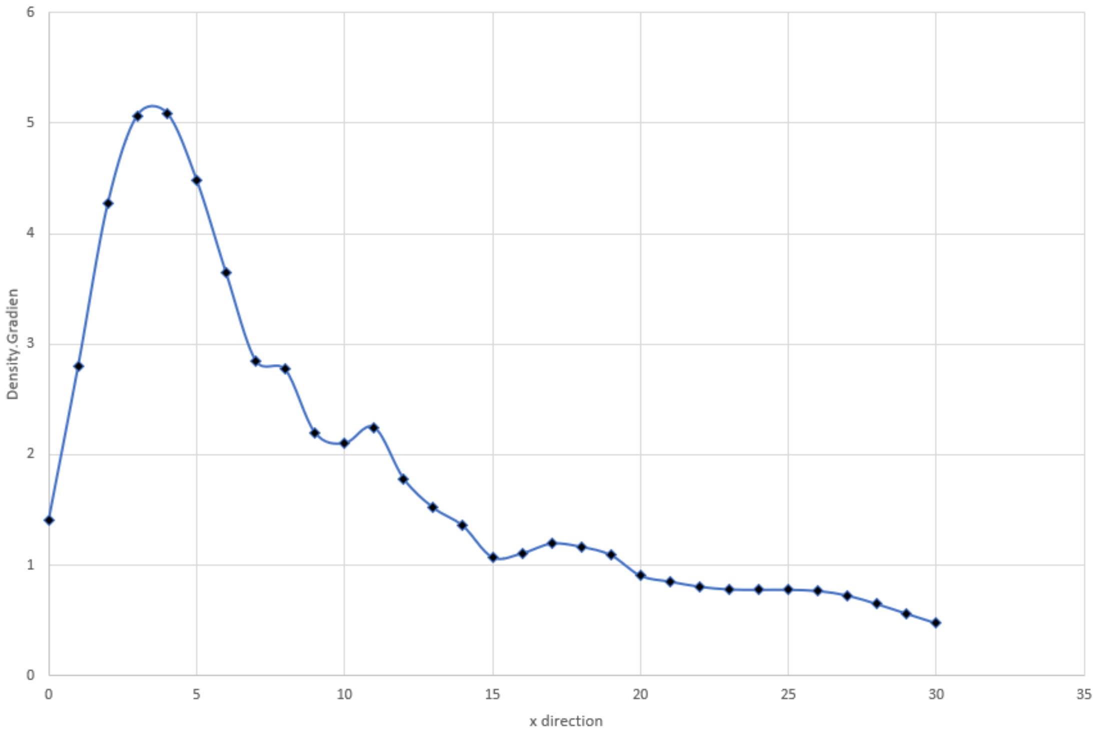


Figure 13. Density gradient in horizontal direction away from the heat sink.

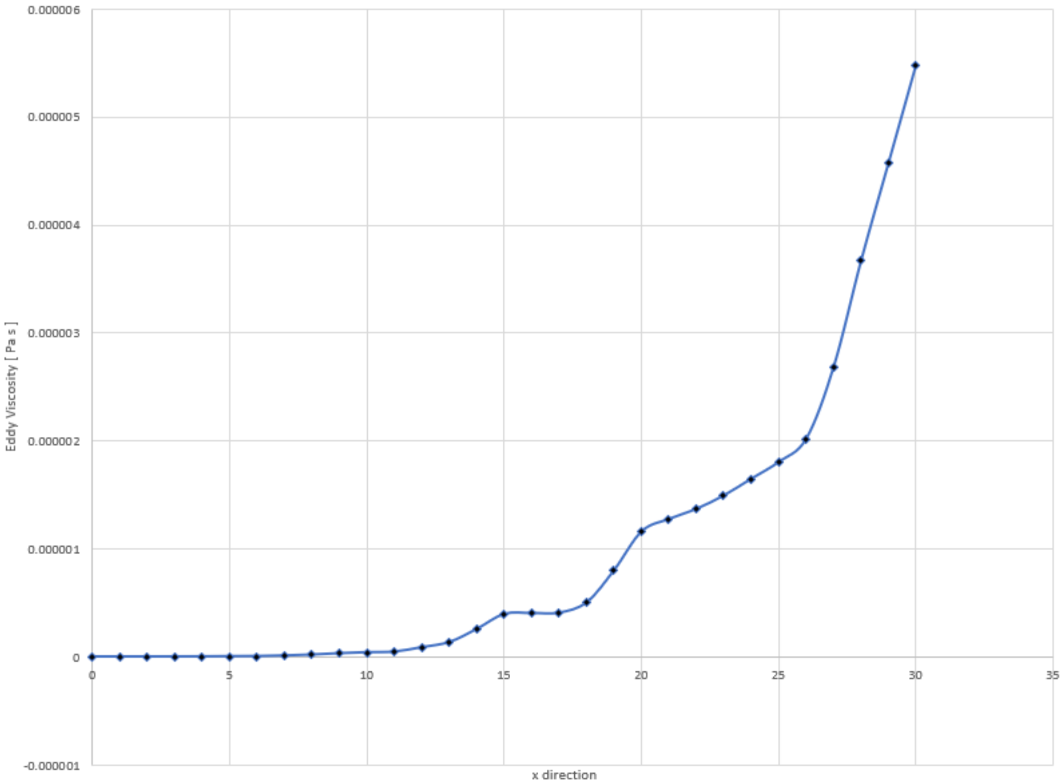


Figure 14. Turbulent viscosity change in horizontal direction away from the heat sink.

The velocity change in horizontal direction away from the heat sink in Figure 16 may mislead since it shows considerable values of velocity in x-direction. When natural convection motion towards +y direction needs more fluid due to satisfying continuity, matter is drawn from all directions.

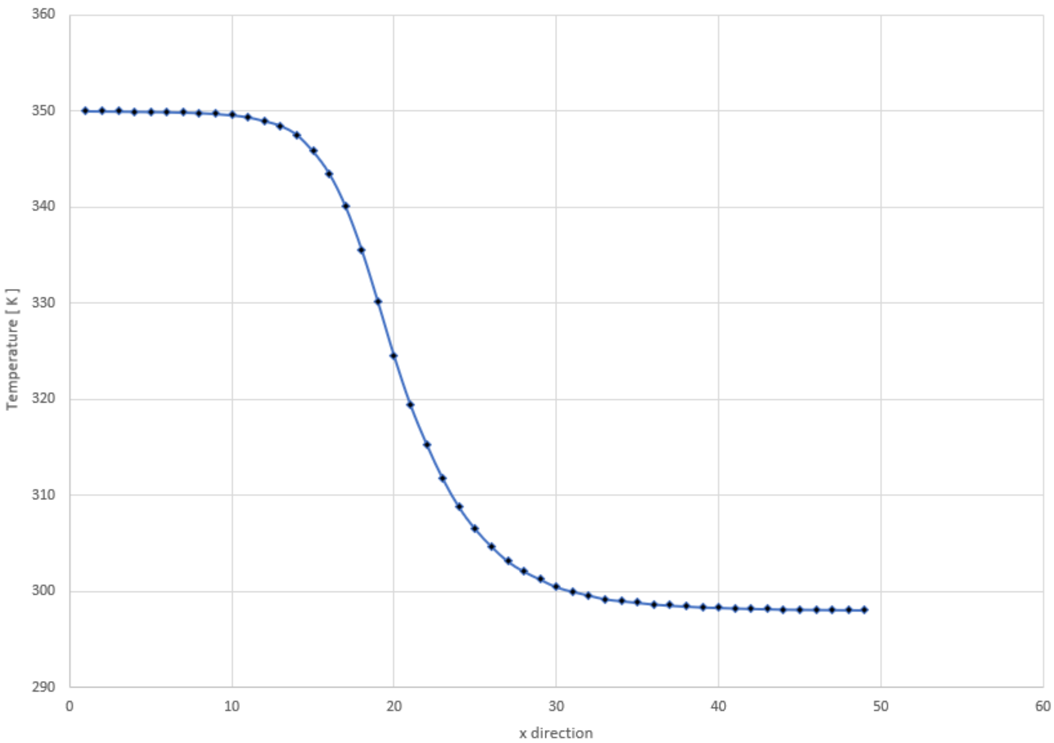


Figure 15. Temperature change in horizontal direction away from the heat sink.

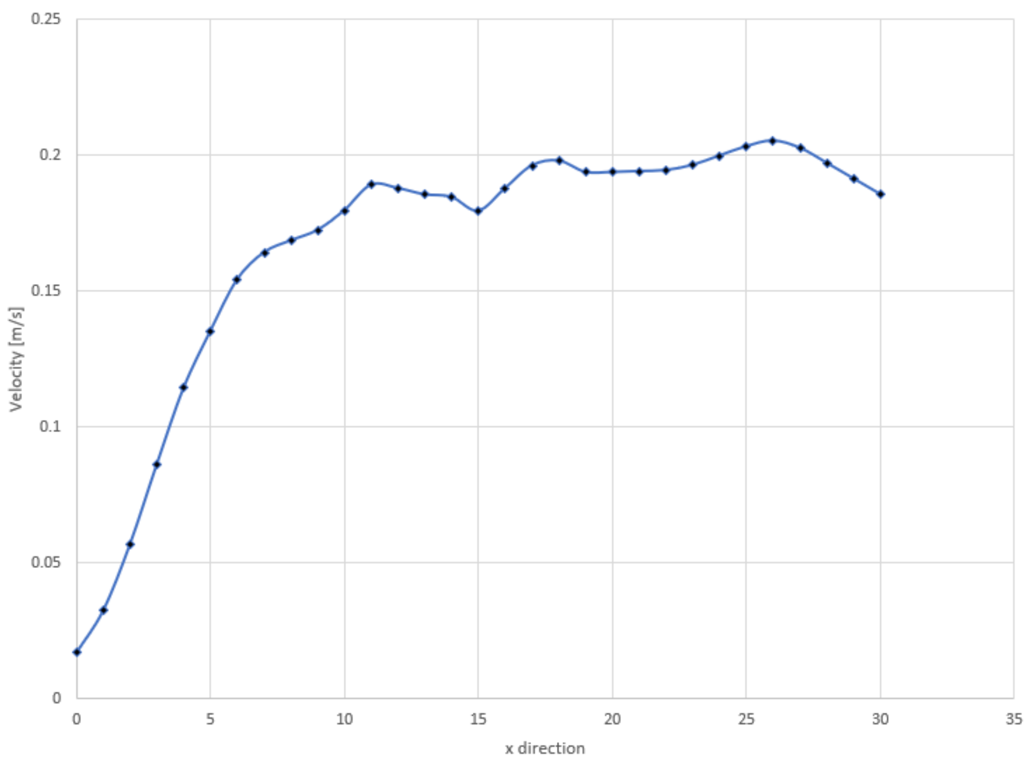


Figure 16. Velocity change in horizontal direction away from the heat sink.

4.Conclusions

This work evaluates a heat sink that is designed for protecting pressure transmitters from damage due to high temperatures. A horizontal orientation is investigated in order to see the natural convection structure and assess the operation possibility of the heat sink. CFD is used in order to have a field of distribution for variables relating to heat transfer and natural convection structure. Following points are drawn from the analysis.

- Horizontal orientation is favorable for protecting pressure sensors by the heat sink without any additional measures.
- Heat sink design is proper for obtaining natural convection cooling.
- Flow that is used for heat dissipation towards ambient is accelerating downstream of the heat sink. A blockage in a range of ten width length of the heat sink in downstream direction would reduce thermal performance.

Author Contributions: Conceptualization, Ali H. Abdulkarim and Eyub Canli; methodology, Ali H. Abdulkarim and Eyub Canli; software, Eyub Canli; validation, Mustafa Wahby Kanbar Jaber; formal analysis, Ayad A. Kakei; writing original draft preparation, Eyub Canli, MustafaWahby Kanbar Jaber; writing review and editing, Ali H. Abdulkarim and Eyub Canli; visualization, Ali H. Abdulkarim, Ayad A. Kakei, and Eyub Canli; supervision, Ali H. Abdulkarim. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding

Conflicts of Interest: The authors declare no conflict of interest.

References

[1] M. Ramzan, N. Shaheen, S. Kadry, Y. Ratha, and Y. Nam, "Thermally Stratified Darcy Forchheimer Flow on a Moving Thin Needle with Homogeneous Heterogeneous Reactions and Non-Uniform Heat Source/Sink," *Applied Sciences,* vol. 10, p. 432, 2020.

[2] Z. Duan, X. Lv, H. Ma, L. Su, and M. Zhang, "Analysis of Flow Characteristics and Pressure Drop for an Impinging Plate Fin Heat Sink with Elliptic Bottom Profiles," *Applied Sciences,* vol. 10, p. 225, 2020.

[3] P. Carvalhaes-Dias, A. Cabot, and J. Siqueira Dias, "Evaluation of the thermoelectric energy harvesting potential at different latitudes using solar flat panels systems with buried heat sink," *Applied Sciences,* vol. 8, p. 2641, 2018.

[4] Y. Wang, J.-J. Ren, C. Wang, and M.-S. Bi, "Mechanism analysis of the buoyancy effects on heat transfer of supercritical nitrogen in horizontal serpentine tube," *International Journal of Heat and Mass Transfer,* vol. 152, p. 119417, 2020.

[5] C. R. Kharangate, W. Libeer, J. Palko, H. Lee, J. Shi, M. Asheghi*, et al.*, "Investigation of 3D manifold architecture heat sinks in air-cooled condensers," *Applied Thermal Engineering,* vol. 167, p. 114700, 2020.

[6] T. Saravanakumar and D. S. Kumar, "Performance analysis on heat transfer characteristics of heat SINK with baffles attachment," *International Journal of Thermal Sciences,* vol. 142, pp. 14-19, 2019.

[7] T.-M. Jeng and S.-C. Tzeng, "Numerical simulation of laminar forced convection of pin-fin heat-sink array in a Channel by Using Porous Approach," *Applied Sciences,* vol. 5, pp. 1846-1868, 2015.

[8] Y.-W. Chen, C.-C. Wu, C.-C. Hsu, and C.-L. Dai, "Fabrication and testing of thermoelectric CMOS-MEMS microgenerators with CNCs film," *Applied Sciences,* vol. 8, p. 1047, 2018.

[9] A. Hasan, H. Hejase, S. Abdelbaqi, A. Assi, and M. O. Hamdan, "Comparative effectiveness of different phase change materials to improve cooling performance of heat sinks for electronic devices," *Applied Sciences,* vol. 6, p. 226, 2016.

[10] K. S. Ong, C. F. Tan, and K. C. Lai, "Methodological considerations of using thermoelectrics with fin heat sinks for cooling applications," *Applied Sciences,* vol. 7, p. 62, 2017.

[11] M. Saadah, E. Hernandez, and A. A. Balandin, "Thermal management of concentrated multi-junction solar cells with graphene-enhanced thermal interface materials," *Applied Sciences,* vol. 7, p. 589, 2017.

[12] M. A. Alfellag, H. E. Ahmed, and A. S. Kherbeet, "Numerical simulation of hydrothermal performance of minichannel heat sink using inclined slotted plate-fins and triangular pins," *Applied Thermal Engineering,* vol. 164, p. 114509, 2020.

[13] S. M. Hoi, A. L. Teh, E. H. Ooi, I. M. L. Chew, and J. J. Foo, "Plate-fin heat sink forced convective heat transfer augmentation with a fractal insert," *International Journal of Thermal Sciences,* vol. 142, pp. 392-406, 2019.

[14] H.-C. Hsu and Y.-C. Huang, "Numerical simulation and experimental validation for the thermal analysis of a compact LED recessed downlight with heat sink design," *Applied Sciences,* vol. 7, p. 4, 2017.