

The influence of baffle geometry on turbulent heat transfer on air flow and convective heat transfer enhancement in air-filled solar collectors

Mohammed Amine Amraoui ^{a,*}

a. Laboratory of Mechanics of Structures and Solids (LMSS). Faculty of Technology, Department of Mechanical Engineering, University Djillali LIABES Sidi-Bel-Abbès, BP 89 22000 Sidi-BelAbbès, Algeria

* Corresponding author: mohamed.amraoui@univ-sba.dz

Keywords	Abstract
Solar drying; Solar heating; Thermodynamics; Turbulence; CFD;	Solar heating and drying using solar air collectors is an environmentally friendly and cost-effective method for drying various products such as grain, fruit, or even wood. These collectors, installed on the roof or in a sunny location, capture the sun's energy. Air circulates through these devices, where it is heated by solar heat, and is then directed into the building or into a storage system for later use. The technique uses solar energy to heat the air, which is then directed toward the product to be dried. Recent innovations focus on improving their performance, durability, and their integration into more complex systems to maximize solar capture. In the study, three types of solar collectors were analyzed using three-dimensional numerical simulation: the collector with four-tooth obstacles (SCFT), the collector with three teeth (SCTT), and the collector with two teeth (SCTWT). The thermal efficiency factor (THEF) exceeded 1 for all three models, indicating improved thermal efficiency, even when considering additional pressure losses. Among them, the SCTWT model displays the highest values, reaching 1.3670, followed by the SCTT model with a maximum value of 1.3331, and then the SCFT model, which reaches 1.3123.

1. Introduction

Flat-plate air solar collectors have been the subject of extensive research in recent years due to their potential for heating and ventilation in buildings. Studies have focused primarily on improving their thermal efficiency by optimizing the design of the absorbers, the materials used, and the system configuration.

Due to the environmental and financial constraints associated with nanofluids, there are few large-scale direct absorption solar collectors (DASCs) operational in the field. To fill this gap, Pavel G. Struchalin et al [1]. analyzed five months of fieldwork for large-scale DASC, using an environmentally friendly and cost-effective nanofluid. During this period, the DASC was compared to a commercial flat solar heating device installed in the same location. Hekal et al [2]. present a laboratory study on the flow of hot and cold air through textiles. It is necessary to develop a precise method to determine the friction coefficient and Nusselt number of the airflow within these ducts, given their major influence on the design of centralized air conditioning systems. The principle involves conducting experimental research to evaluate the thermodynamic parameters of air movement and performing a CFD numerical simulation to provide refined and detailed profiles. As part of the development of a clean-energy ground-to-air geothermal heat exchanger (GATHE), Mr. Rana et al [3]. selected Fluent for system modeling. Three soil samples, with thermal conductivities of 0.65, 1.25, and 3.50 W·m⁻¹·K⁻¹, were used to study the thermal properties. The critical soil thickness was found to be ten times the duct diameter, and a minimum outlet temperature of 300.1 K was obtained for a 60 m duct when the soil thermal conductivity was 3.50 W·m⁻¹·K⁻¹. To further improve performance, several fin arrays were introduced into the duct with pitches of 0.5 m, 1.0 m, and 2.0 m. Thus, for a 10 m long pipe equipped with fins every 0.5 m, the observed outlet temperature was 306.7 K.

The study by Firas A. Abbas and Mohammed H. Alhamdo [4] focused on a study with computer simulation and they validated the work by experimenting the heat transfer performance of an asphalt solar heater. To do this, 3,000 mm parallel copper tubes placed in a serpentine shape were integrated into the middle of a hot-mix asphalt model measuring 1,000 × 600 × 80 mm. The end and bottom of the test sections were insulated to prevent energy loss to the environment. Four water quantity values and three simulated solar radiation values were used to examine the device performance. The interest of the study by Armita Soleimani Ghalati et al [5]. was to study and predict the effect of solar water heater absorber tilt and type of solar heater on thermal efficiencies Using a simulated neural network (ANN) simulation, two distinct models ((MLP) and (RBF)) were used to guess the energy characteristics of the device that heats the water by the sun rays. Parabolic solar heaters (PTSC) suffer from low thermal efficiency, which limits their use as a clean energy source. To overcome this problem, Guoliang Hou et al [6] proposed the installation of internal fins in the receiver tube. This passive technique is recognized for being reliable, economical and simple to implement. However, although these internal fins significantly increased heat transfer in swirling flows, they also significantly increase the pumping work required, which can be a disadvantage.

It appears that the study by Peng et al [7]. presented a new solar air heater with an absorber fitted with fins, aimed at improving heat transfer. At a constant flow rate, they experimentally and theoretically studied twenty-five finned collectors and one flat-plate collector. From measurements and theoretical predictions, the useful heat flux, collector efficiency, and air-absorber heat transfer coefficients were determined. Actual work gives results on the average efficiency of twenty-five different finned radiators varies between 0.5 and 0.74, even the glazing gives a solar transmittance of 0.83. They also propose a correlation equation to estimate the maximum thermal efficiency (hmax) of these collectors as a function of the dimensionless fin span (s/d) and the dimensionless fin height (h/d). In their study, Peng et al [8]. proposed a novel receiver tube model based on the cavity concept,

designed for a direct absorption solar collector (DASC) featuring two linear parabolic reflectors (LPR). This receiving tube uses both an insulated wall and a cavity shape to minimize energy loss. A notable advantage is the increase in the path traveled by the solar rays thanks to successive reflections inside the receiving tube, which reduces the agglomeration of the nanoparticles and thus improves the efficiency of the system. The use of a DASC sensor with $\alpha = 0.07$, CC2, a pentagonal reflection configuration, an outer diameter d_o of 12 cm, and $T_o = 310$ K yields better results than the other conditions studied. The efficiency of the new DASC sensor presented exceeds that of the SBSC sensor examined by 5%, thanks to the addition of nanoparticles. In the research conducted by Famiglietti et al [9], a real analysis made to the first small-scale solar field model, occupies an area of 79.2 m². This investigation makes it possible to characterize the energetic and mechanical behavior of the turbocharger, as well as to refine and validate the developed numerical model. Particular attention is paid to the diabatic behavior of the two turbomachines and to the establishment of the limiting efficiency of the turbocharger in the context of advanced research. A detailed model of the turbocharger was developed, incorporating a conventional approach based on the compressor and turbine temperature curves provided by the manufacturer, with a semi-empirical characterization of the heat transfer phenomena influencing its performance—that is, a diabatic turbocharger. The model was validated and optimized using experimental data obtained directly from the constructed installation, thus offering rapid and cost-effective commissioning methods for similar installations. The pilot plant did not achieve optimal operating conditions.

The essential principle of the article by Simo-Tagne et al [10], was to enhance a numerical model for simulating the wood drying process using a solar drying system. Two sets of equations were established: one for the solar air heater and the other for the drying room. The calculated and measured outlet temperatures of the solar heater were compared, and the model showed a very satisfactory prediction. To facilitate understanding of the modeling of the solar collector and the drying chamber, heat transfers are schematically represented and the energy balance of each component of the dryer is defined as follows: the rate of thermal energy accumulation in a component is equal to the sum of the energy flows exchanged with all other components, plus the sum of the gains and losses due to solar irradiation. In the work of Singh et al [11], a device for drying with indirect type solar was studied using (CFD) to simulate the process. Ansys Fluent CFD software was used to simulate and analyze the thermodynamic performance of the dryer in different mass flow rates. Numerical analyses were confirmed by identification with experimental data. The predicted values are validated by experimental results. At the sensor outlet, the average velocity is 2.89 m/s for a mass flow rate of 0.0872 kg/s, and 1.36 m/s for 0.0447 kg/s. The maximum temperature at the sensor outlet reaches 44.6 °C, while that of the absorber plate is 69.57 °C for the same flow rate of 0.0872 kg/s. The temperature profiles of the plates were examined, and the plate adjacent to the sensor exhibits the highest average temperature. The heat transfer enhancement characteristics of a flat plate unit heater were analyzed experimentally by Tong et al [12], by using various nanofluids in the solar heater and comparing the sensitivity of its efficiency under different operating conditions. This study evaluates the performance of a flat-plate solar collector using water and nanofluids based on Al₂O₃, CuO, Fe₃O₄, and multi-walled carbon nanotubes (MWCNTs) as working fluids. The nanofluids used were manufactured using a two-step approach, the most common method in this field. Nanoscale materials with good thermal and rheological properties are produced as dry powders and then dispersed in the base fluid using intensive mixing techniques (magnetic stirring, ultrasound, ball milling, or high-shear stirring).

The objective of the study carried out by Ullah et al [13], was to evaluate the performance of a parabolic solar air heater used for the dehydration process of seasonal fruits according to air flow rates. The experiment was conducted using three levels of air mass flow (F1, F2, and F3) and two absorber diameters (D1 and D2), with a concentration ratio of 60. Each day, the intensity of solar radiation was recorded using a pyrheliometer, a mechanical instrument used to measure direct radiation during the experiment. They also recorded the temperature with a thermometer placed in the absorber tube of the parabolic solar collector, in the drying chamber, and in the ambient air. The water content of the apples was determined hourly until a residual moisture level of 10% was reached, in order to limit insect infestations, mold, and bacterial growth. In the work of Ural et al [14], a new creation was designed and tested, integrating an air-water heat pump and a solar air heater equipped with a fabric absorber. This system aims to solve the problem of frost deposition on the evaporator under conditions of increasing temperature. This enables synchronized thermal exchange within an integrated system utilizing R410A as both a gaseous and liquid heat source. This study uses a novel solar air collector (SAC) to evaluate the thermodynamic performance of a solar heat pump (SHPP) system. Unlike conventional SACs, this collector is economical and lightweight thanks to the use of a black non-woven fabric as an absorber, replacing metallic materials. The fabric is positioned diagonally along the collector. Air is drawn in through the inlet located beneath the collector, then circulates between the fabric-upper contact area, and then between the fabric and the lower part of the collector. Simultaneously, the air passes through the heated fabric, ensuring heat transfer. A fan located at the SAC outlet ensures air circulation within the collector. Zarei et al [15], propose a novel polygeneration system driven by two distinct solar energy sources. This system uses thermal and parabolic photovoltaic collectors configured in series with water as the working fluid. The integrated system also includes an ejector-compression refrigeration cycle (VCRC) with two temperature levels (above and below zero) as well as an organic Rankine cycle (ORC). These components are designed to simultaneously provide cooling, heating and electricity, illustrating an innovative approach to solar polygeneration. The objective of this research is to improve the use of solar energy for domestic cooling, heating, and electricity. The proposed system consists of three subsystems: (a) a vapor-ejector refrigeration and compression cycle with two evaporators (positive and negative) for refrigeration and freezing; (b) a regenerative organic Rankine cycle for generating electricity; and (c) PVT PTC modules.

In the study by Zhou et al [16], a dynamic resistance and capacitance (3R2C) network model for a high-performance air-to-ground solar energy storage system with intermittent through-wall ventilation (SAC-HVIW) has been developed. This model is developed to analyze the thermal performance of the system. It uses theoretically identified values of resistance and capacitance to model the building, as well as the number of transfer units method to represent heat exchange with intermittently ventilated walls. Heat transfer in this system is primarily due to radiation and convection. The solar collector absorbs radiation and heats the air circulating inside. The air passing through the solar air collector (SAC) can reach temperatures exceeding 50°C. This hot air then circulates through the HVIW air duct and exchanges heat with the inner surface of the wall by convection, keeping this surface temperature above 20°C. Through transient conduction, heat spreads from the warm inner surface to the outer surface of

the wall. As the outer surface heats up, it transfers heat to the ambient air and other surfaces by convection and infrared radiation. The room is also influenced by external conditions, with heat exchanged by convection and radiation with the external environment. This study conducted by Zhu et al [17]. presents a new type of compound parabolic concentrator (CPC) solar air collector using flat microheat pipe arrays (FMHPA). The system features a cylindrical absorber housing the FMHPA within an evacuated glass tube, engineered for efficient heat transfer during operation. The collector, referred to as the FMHPA-CPC SAC, is designed as an edge-ray concentrator with a concentration ratio of 1.8, indicating its ability to concentrate incident sunlight by a factor of 1.8. This study presents an innovative concentrating solar collector (CPC-SAC). A series of experiments was conducted in 2014 in Beijing, China, to characterize the collector's thermal performance. Measurements revealed stable performance throughout the experimental period, with an average annual thermal efficiency of 52%. The new configuration resulted in a substantial improvement in thermal performance. The article by Zhu et al [18]. presents a new type of solar flat air heater (SAC) equipped with micro heat pipe arrays (MHPA). This innovation is distinguished by a unique structure that separates the processes of heat collection and transfer from those of ventilation and heat exchange. This innovation in the design of flat air solar collectors with micro-heat pipe arrays represents a significant advancement in solar energy technologies, promising more efficient and optimized systems for various thermal applications. Solar energy is captured by the MHPA evaporation section via the organic compound of the high-performance absorbent film, producing a heat pipe effect. This heat is then released into the air at the vent and heat exchange section of the collector, warming the air. Simultaneously, the heat transfer fluid in the MHPA undergoes a continuous phase transition cycle and efficiently transfers the heat from solar radiation to the air circulating in the duct. The core of the MHPA absorbent plate ensures optimal heat absorption, efficient conduction, and a large dissipation surface area, resulting in high synchronized efficiency.

The articles by Amraoui et al [19-26]. focus on the study of the aerodynamic and thermal behavior of various types of solar plate heaters. Their research uses computational fluid dynamics (CFD) to analyze turbulent airflow in the presence of different types of baffles. The work of Amraoui et al. make important contributions to understanding and improving the performance of solar collectors, using advanced simulation and analysis tools to explore innovative solutions in the field of solar energy. One of the essential problems of using solar energy is the low thermal exchanges carried out with the air in the dynamic vein of the solar collector. The study conducted by Arkhazloo et al [27]. Provides a comprehensive transient analysis of the heating behavior of large forged blocks within a gas heat treatment furnace. This analysis integrates experimentally measured temperatures as well as numerical simulations (CFD) to evaluate the complex interactions of heat and fluid flow in the furnace. This study represents an important advance in the use of CFD simulations to analyze and optimize heat treatment processes, offering promising prospects for the improvement of industrial processes in various sectors.

The article by Zheng et al [28]. Proposes a dry air dehumidification method based on air conditioning to prevent condensation in switchgear. This study uses numerical simulations to analyze different air flow distributions and their impact on the thermohygrometric environment within the equipment. This study provides an innovative and practical solution to prevent condensation in switchgear, thus contributing to improving their reliability and performance. The study by Boonloi and et al [29]. explores the performance and patterns of flow and energy performance in a square duct fitted with six different types of modified corrugated baffles. Researchers examined the effects of fluid blockage rates, baffle heights, flow directions, and baffle shapes on Nusselt number and fluid flow mechanisms under laminar flow conditions. The research by (Xu et al [30]. Proposed two models: an air entrainment model (AEM) and a volumetric mass transfer heating source model (MT-VHS). Then, different physical tunnel models, gas description models and combustion models were used to refine their analysis. The results obtained by computational fluid dynamics (CFD) were compared to experiments carried out on a reduced scale as well as to previous theories, making it possible to validate the efficiency and precision of the new proposed models. Unlike some CFD tools, such as FDS, FLUENT can better capture the geometric characteristics of complex tunnels. However, in previous work, the temperature predicted by FLUENT was consistently higher than experimental values and theoretical predictions, and the smoke layer height was rarely verified. To address this, this paper introduces an air entrainment model (AEM) and a mass transfer model with a volumetric heating source (MT-VHS). Then, using a small-scale fire experiment as a scenario, various physical models, combustion mechanisms, and gas descriptions were tested and compared with respect to temperature profiles and smoke layer height.

The work of (Razzaghi et al [31]. presents a numerical procedure for the study of turbulent fluid flow and heat transfer in a twisted flat tube. In this study, three different geometries of the flat tube cross section are examined. Water, used as active fluid, has constant thermophysical properties throughout the experiments. The simulations cover a range of Reynolds numbers from 5,000 to 20,000, allowing the impact of different geometries on heat transfer performance and turbulent flow characteristics to be assessed. The study by (Shafieian et al [32]. Presents a validated three-dimensional transient computational fluid dynamics (CFD) analysis to investigate the impact of inlet flow conditions on the thermal performance of solar storage tanks. Researchers analyzed the impacts of different mass flow rates on the thermal performance of various reservoirs, thereby better understanding and optimizing flow conditions to improve the efficiency of solar energy storage systems.

Donghui Wei et al [33]. studied the heat pipe solar heater. They simulated the energy controls of a heat pipe solar receiver using the anisotropic solar sky emission model. This simulation was carried out for different inclination angles and azimuths in the city of Isfahan, Iran. Theoretical results of this simulation were compared to experimental results, revealing an average deviation of 5%. K. The results are presented through two experimental protocols. The study separately examines the influence of the tilt angle and orientation angle on heat absorption by heat pumps, by analyzing the temporal evolution of the hot water temperature. To facilitate comparison, experimental data and theoretical results are displayed on the same graph in each protocol, and the accuracy of the proposed model is verified. After model validation, the optimal orientation and tilt angles are determined, and relevant correlations are established. Goudarzi et al [34]. Focused on creating a device for a conical sun heating device and the improvement of its convection according to the ASHRAE rule. The goal of their study was to achieve a significant energy transfer to that of other available heating. Due to experimental shortcomings, they determined the ideal shape of the conical sun water heater by computer simulation. Work on the heat transfer performance of solar heaters placed in buildings is limited. This article by Wenjie Liu et al [35]. carried out an experimental and computational study using Fortran computer programs they

developed themselves. The HPR decay study in solar-powered home-integrated systems (BISS) showed the solar heater performance. The results highlight the advantage of a selective chromium-based coating for reducing energy losses, with glazing having a less pronounced impact. Optimal HPR module designs for building-integrated solar systems (BASS) were determined in different Chinese climatic zones, with average annual thermal efficiencies ranging from 62.6% to 73.5%.

Finally, studies have also evaluated the performance of these sensors in different climates and environmental conditions, in order to better adapt their design to local needs. In short, research continues to advance this technology to make it more efficient, sustainable, and suitable for residential and industrial applications.

2. Problematic

The article introduces innovative solutions for devices designed to elevate air temperature for industrial drying applications or residential heating. Three types of air-heating machines were developed: a solar collector with four-tooth obstacles (SCFT), a solar collector with three-tooth obstacles (SCTT), and a solar collector with two-tooth obstacles (SCTWT).

Each heater measures 0.9 m in length, 0.5 m in width, and 0.025 m in depth, with their design illustrated in Figure 1

The boundary conditions adopted in this study are as follows:

The absorber temperature is set at $T_{ab} = 380$ K, the insulation temperature at $T_{iso} = 340$ K, and the air inlet temperature at $T_0 = 300$ K.

The flow is characterized by an initial velocity U_0 at the inlet, while the outlet pressure is assumed to be equal to atmospheric pressure ($P = P_{atm}$).

3. Formulas utilized [19]

3.1. The first formula

* The continuity equation is introduced in this section

$$\nabla \cdot (\rho \vec{v}) = 0 \quad (1)$$

$$\frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial y}(\rho v_r) + \frac{\rho v_r}{r} = 0 \quad (2)$$

3.2. The second formula

* The principle of momentum conservation

$$\nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla(\bar{\tau}) + \rho \vec{g} \quad (3)$$

$$\bar{\tau} = \mu \left[(\nabla \vec{v} + \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \quad (4)$$

$$\frac{1}{r} \frac{\partial}{\partial x} (r \rho v_x v_x) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v_r v_x) = -\frac{\partial p}{\partial x} + \frac{1}{r} \frac{\partial}{\partial x} \left[r \mu \left(2 \frac{\partial v_x}{\partial x} - \frac{2}{3} (\nabla \cdot \vec{v}) \right) \right] + \frac{1}{r} \frac{\partial}{\partial r} \left[r \mu \left(\frac{\partial v_x}{\partial r} + \frac{\partial v_r}{\partial x} \right) \right] \quad (5)$$

$$\frac{1}{r} \frac{\partial}{\partial x} (r \rho v_x v_r) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v_r v_r) = -\frac{\partial p}{\partial r} + \frac{1}{r} \frac{\partial}{\partial x} \left[r \mu \left(\frac{\partial v_r}{\partial x} + \frac{\partial v_x}{\partial r} \right) \right] + \frac{1}{r} \frac{\partial}{\partial r} \left[r \mu \left(2 \frac{\partial v_r}{\partial r} - \frac{2}{3} (\nabla \cdot \vec{v}) \right) \right] - 2 \mu \frac{v_r}{r^2} + \frac{2}{3} \frac{\mu}{r} (\nabla \cdot \vec{v}) + \rho \frac{v_z^2}{r} \quad (6)$$

where, $\nabla \cdot \vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_r}{\partial r} + \frac{v_r}{r}$ and v_z is the swirl velocity.

$$\frac{1}{r} \frac{\partial}{\partial x} (r \rho u w) + \frac{1}{r} \frac{\partial}{\partial r} (r \rho v w) = \frac{1}{r} \frac{\partial}{\partial x} \left[r \mu \frac{\partial w}{\partial x} \right] + \frac{1}{r^2} \frac{\partial}{\partial r} \left[r^3 \mu \frac{\partial}{\partial r} \left(\frac{w}{r} \right) \right] - \rho \frac{v w}{r} \quad (7)$$

* The energy equation

$$\nabla \cdot (\vec{v} (\rho E + p)) = \nabla \cdot (k_{eff} \nabla T - \sum_j h_j \vec{J}_j + (\bar{\tau}_{eff} \cdot \vec{v})) \quad (8)$$

$$k_{eff} = k' + \frac{C_p \mu_t}{Pr_t} \quad (9)$$

The turbulent Prandtl = 0.85.

$$E = h' - \frac{p}{\rho} + \frac{v^2}{2} \quad (10)$$

$$h' = \sum_j Y_j h_j + \frac{p}{\rho} \quad (11)$$

$$h_j = \int_{T_{ref}}^T c_{p,j} dT \quad (12)$$

Where $T_{ref} = 298.15k$.

*** The equations used for turbulence (Standard k-ε Model)**

The methods employed for direct simulation rely on principles of statistical physics, treating turbulence as a statistical phenomenon described solely by its temporal distribution at each point. The approach involves several steps: formulating equations for mean values and fluctuations, modeling terms associated with fluctuations as needed, and linking these terms to standard laws governing flow behavior near walls. Hybrid methods, such as large-scale simulations, can also be utilized. These methods filter the turbulence spectrum, directly computing large-scale structures while modeling smaller scales as described earlier.

An effective calculation of turbulent viscosity within the fluid circuit is based on the turbulent viscosity model. However, the K-epsilon standard model is used as the turbulence model in this case.

$$\frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon \quad (13)$$

$$\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (14)$$

$$G_k = -\rho \overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i} \quad (15)$$

(3) Modeling the Turbulent Viscosity:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (16)$$

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

*** The formulas for the Nusselt number, friction factor, and Stanton number**

$$Nu_u = 0.023 Re^{4/5} Pr^{0.4} \quad (17)$$

Where $Re \geq 10^4$

$$f = (0.79 \ln Re_D - 1.64)^{-2} \quad (18)$$

Where the Re is between 3000 and 5.10^6

$$St.Pr^{2/3} = \frac{f}{8} \quad (19)$$

*** The thermal improvement factor**

$$THEF = \frac{(Nu/Nu_B)}{(f/f_B)^{1/3}} \quad (20)$$

*** The thermohydraulic performance**

$$THPP = \frac{(St/St_B)^3}{(f/f_B)} \quad (21)$$

4. Overall Findings

4.1. Mesh

Meshing, or mesh generation, is the process of creating a two- or three-dimensional grid that breaks down complex geometries into simpler elements, facilitating the discretization of the computational domain. Ansys Mesh is widely regarded as a benchmark tool for modeling and simulating airflow in sensor systems.

Figure 2 presents various mesh configurations for the three types of sensor models, each with distinct mesh quality levels, as detailed in Table 1.

4.2. Validation of simulation results and mesh integrity

Validation of simulations from our studies is a crucial step in ensuring that the models and simulations accurately reflect real-world conditions. This validation allows us to continue our work on modeling different sensor types aimed at improving heat transfer by convection, which is essential for optimizing solutions for industrial applications and home heating. Notably, our CFD results show a strong agreement with the experimental data from Moumni [36] (see figure 3).

A discrepancy of up to 4°C can be observed between experimental and numerical results. This difference can be explained by several factors.

Firstly, numerical simulations are often based on simplifying assumptions, such as the homogeneity of the medium, ideal conditions, the absence of defects, or the use of theoretical equations and boundary conditions. These simplifications can lead to a discrepancy with experimental reality.

Secondly, experimental measurements also involve uncertainties, particularly those related to the limited precision of the instruments, measurement noise, potential sensor miscalibration, and human error.

All of these factors can explain the observed difference between the two types of results.

4.3. The in-depth results

The velocity distribution has a logarithmic profile, reaching a maximum at the center of the flow before gradually decreasing to zero at the walls, due to the friction effects of the air against them. The presence of obstacles in the flow bed influences the development of the boundary layer, which evolves to become turbulent from the first phases of the flow.

It is observed that turbulence, in the devices studied, is characterized by a random and three-dimensional nature. It occurs at relatively high Reynolds numbers, constituting an intrinsic property of the flow, unlike fluid-specific properties such as viscosity. Furthermore, turbulence enhances the transport capacity within the fluid by intensifying diffusion processes—specifically the diffusion of momentum and heat—while also increasing the dissipation of mechanical energy into heat (see Figure 4).

Figure 5 shows the scooter movement; the speed is modified to avoid certain obstacles. This parameter is characterized by a vector representation. Thanks to the reduction in section there is an accumulation of vectors close to the walls and the end of the sensor.

Turbulence in collectors refers to an air state characterized by a velocity exhibiting swirling behavior at all points, with vortices whose size, location, and orientation constantly vary. Air flows are therefore distinguished by a chaotic appearance, a behavior that is difficult to predict, and the presence of multiple spatial and temporal scales.

In our studied models, the complex behavior of turbulent flows is generally analyzed by a statistical approach, thus integrating the study of turbulence into the field of statistical physics. To reflect the dominance of inertial forces over viscosity forces in the air stream flow, the Reynolds number, properly defined, must exceed a certain threshold.

To model complex turbulence in solar channels, we adopted the K-epsilon model (see figures 6 and 7)

Thermal energy results from the movement of microscopic particles in the air, such as atoms and molecules. These particles exhibit kinetic energy and are continuously in motion. Their speed is observed to increase near walls and baffles, which leads to an increase

se in thermal energy in these areas. Heat is defined as the transfer of thermal energy between regions, consistently moving from higher to lower temperature areas (see figure 8).

The Nusselt number increases with the Reynolds number, which is qualitatively explained by the analysis of the fluid flow field. It is also influenced by the physical properties of the fluid, represented by the Prandtl number. Its evaluation is based on empirical equations derived from experimental data. However, in recent years, significant advances have been made using semi-analytical methods. In a steady turbulent regime, in a long and smooth channel, the Dittus-Boelter relation, also known as the Colburn formula.

The NU values for the SCFT model corresponding to each input speed (12, 15, 18, 21, 24, 27 et 30) m/s are: 81.97, 98.3, 114.31, 125.39, 139.75, 154.02 and 167.19. As for the SCTT and SCTWT models, they present an increase of 1.82%, 1.57%, 0.9%, 2.76%, 2.53%, 2.22% and 2.48%, respectively, as well as 4.11%, 3.72%, 3.31%, 6.49%, 6.35%, 6.07% and 5.52% compared to the SCFT model (see figure 9).

When the fluid in the sensors has a non-zero viscosity, the fluid layers in contact with the walls of the sensors experience viscous friction forces. These forces result from the interaction between the fluid and the internal surface of the walls. As with all frictional forces, viscous friction opposes the direction of fluid motion. Although it may seem difficult to consider a fluid layer as a separate object, it is possible to model it in this way to analyze the friction forces acting on it. The fluid layers near the walls experience a reduction in velocity due to these forces, as illustrated in the figure 10.

The values of the magnitude ffor the (SCFT) model are: 0.0231, 0.0219, 0.0210, 0.0204, 0.0198, 0.0193 and 0.0189. Compared to this model, the magnitudes obtained with the (SCTT) and (SCTWT) models are reduced respectively by: 0.43%, 0.45%,

0.47%, 0.49%, 0.50%, 0.51% and 1.05% for the SCTT model, and by: 1.29%, 0.91%, 0.95%, 1.47%, 1.51%, 1.55% and 2.11% for the (SCTWT) model.

The Stanton number (St) is a dimensionless parameter that evaluates the heat or mass transfer in a fluid flow, relating it to the heat or mass capacity of the fluid. It depends on (Re) and (Pr). As the Reynolds number increases, the Stanton number typically decreases, reflecting a reduction in transfer efficiency compared to the available kinetic energy.

Figure 11 shows the St values obtained with the (SCFT) model, which are respectively: 0.003669459, 0.003477076, 0.003328681, 0.003242277, 0.003145196, 0.003061841 and 0.002994016. In comparison, the St values obtained with the (SCTT) and (SCTWT) models are lower than those of the (SCFT) model by 0.55%, 0.46%, 0.28%, 0.77%, 0.7%, 0.61% and 0.67% for the (SCTT) model, and by 1.29%, 1.07%, 0.94%, 1.76%, 1.70%, 1.61% and 1.70% for the SCTWT model.

The Nusselt (Nu), friction (f), and Stanton (St) ratios were plotted as a function of Reynolds number (Re). Figure 12 illustrates the heat transfer enhancement ratio used to evaluate the thermal efficiency of various solar systems. This improvement is particularly pronounced for the SCTWT model, with values reaching 1.327. In comparison, the SCTT and SCFT models exhibit slightly lower ratios, reaching 1.298 and 1.282, respectively.

Regarding the average normalized friction factor (f/f_{wo}), the values obtained for the SCTWT model are relatively low, indicating that pressure drops are lower compared to the SCTT and SCFT models (see Figure 13).

Regarding the Stanton ratio (St/St_{wo}), also shown in Figure 14, it remains below 1 for all three models, with values around 0.9. This indicates that the new configuration transfers approximately 10% less heat per unit mass and temperature than the reference configuration.

The thermal improvement factor is a concept used to assess the effectiveness of techniques aimed at optimizing heat transfer. It essentially corresponds to the ratio between the increase in the heat transfer rate and the increase in the friction factor generated by this improvement. In other words, it measures the trade-off between thermal gain and pressure loss.

THEF are higher than 1 for the three models studied, therefore the enhancement is thermally efficient even when considering the added pressure drop. The SCTWT model shows the highest values, reaching up to 1.3670, followed by the SCTT model with a maximum value of 1.3331, and then the SCFT model, which reaches 1.3123 (see figure 15)

The expression of THPP results from an analysis based on the second law of thermodynamics as well as on energy efficiency studies applied to forced convection systems. It places more emphasis on the heat transfer gain, raising it to the power of three, and compares it directly to the penalty induced by the pressure drop.

THPP values are less than 0 for the three models studied therefore not thermodynamically efficient — too much pressure drop for the heat transfer gain (see figure 16).

5. Conclusions

Solar air collectors are devices designed to capture the sun's energy to heat air. Their operating principle is based on the absorption of solar heat by a black or specially treated surface, which then heats the air flowing through or around the collector. Recent advances in their development have included the integration of turbulence promoters, as illustrated in our study. The results of CFD give approaches a beneficial effect.

- The turbulence enhances the transport capacity within the fluid by intensifying diffusion processes—specifically the diffusion of momentum and heat—while also increasing the dissipation of mechanical energy into heat.

- The speed is observed to increase near walls and baffles, which leads to an increase in thermal energy in these areas. Heat is defined as the heat transfer across the entire surface of the collectors, always moving from high energy locations to low energy locations.

- Nu values for the SCFT model corresponding to each input speed (12, 15, 18, 21, 24, 27 and 30) m/s are: 81.97, 98.3, 114.31, 125.39, 139.75, 154.02 and 167.19. As for the SCTT and SCTWT models, they present an increase of 1.82%, 1.57%, 0.9%, 2.76%, 2.53%, 2.22% and 2.48%, respectively, as well as 4.11%, 3.72%, 3.31%, 6.49%, 6.35%, 6.07% and 5.52% compared to the SCFT model (see figure 9).

- The values of the magnitude f for the (SCFT) model are: 0.0231, 0.0219, 0.0210, 0.0204, 0.0198, 0.0193 and 0.0189. Compared to this model, the magnitudes obtained with the (SCTT) and (SCTWT) models are reduced respectively by: 0.43%, 0.45%, 0.47%, 0.49%, 0.50%, 0.51% and 1.05% for the SCTT model, and by: 1.29%, 0.91%, 0.95%, 1.47%, 1.51%, 1.55% and 2.11% for the (SCTWT) model.

- The St values obtained with the (SCFT) model, which are respectively: 0.003669459, 0.003477076, 0.003328681, 0.003242277, 0.003145196, 0.003061841 and 0.002994016. In comparison, the St values obtained with the (SCTT) and (SCTWT) models are lower than those of the (SCFT) model by 0.55%, 0.46%, 0.28%, 0.77%, 0.7%, 0.61% and 0.67% for the (SCTT) model, and by 1.29%, 1.07%, 0.94%, 1.76%, 1.70%, 1.61% and 1.70% for the SCTWT model.

- THEF are higher than 1 for the three models studied, therefore the enhancement is thermally efficient even when considering the added pressure drop. The SCTWT model shows the highest values, reaching up to 1.3670, followed by the SCTT model with a maximum value of 1.3331, and then the SCFT model, which reaches 1.3123.

- THPP values are less than 0 for the three models studied therefore not thermodynamically efficient — too much pressure drop for the heat transfer gain.

Adding obstacles to an air-based solar collector aims to increase flow turbulence, improve the convection coefficient, and reduce the thickness of the thermal boundary layer. These modifications promote more intense air mixing, thus increasing heat exchange between the absorber plate and the fluid.

However, introducing obstacles also presents drawbacks, notably increased pressure drops, which leads to higher energy consumption by the fan.

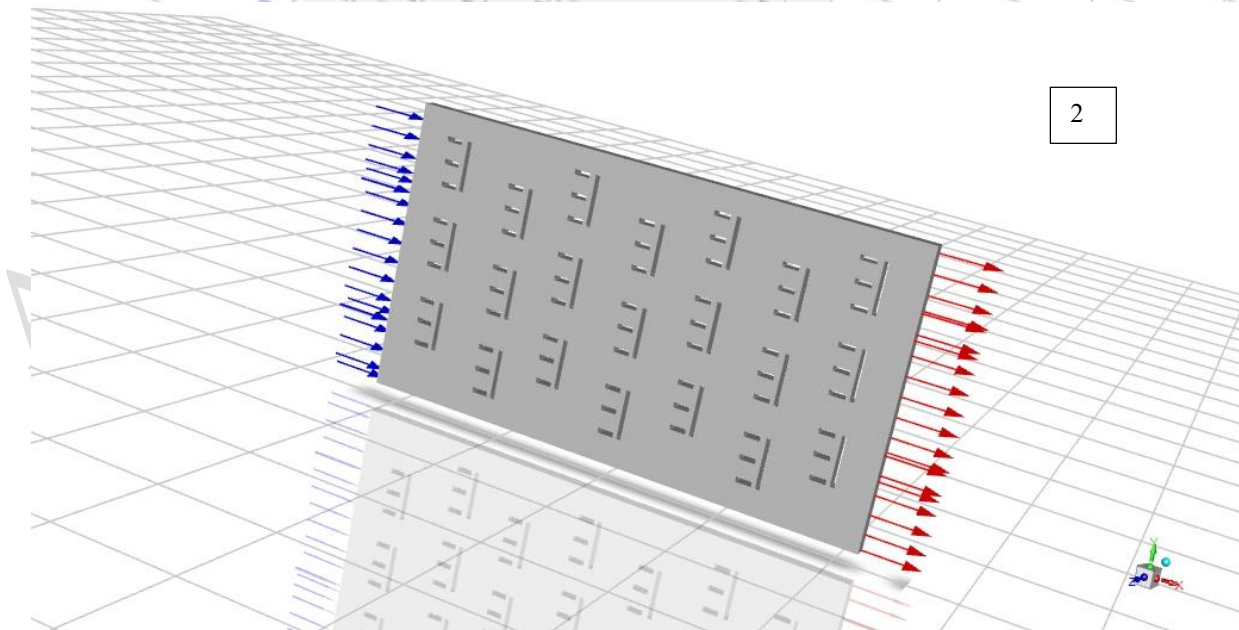
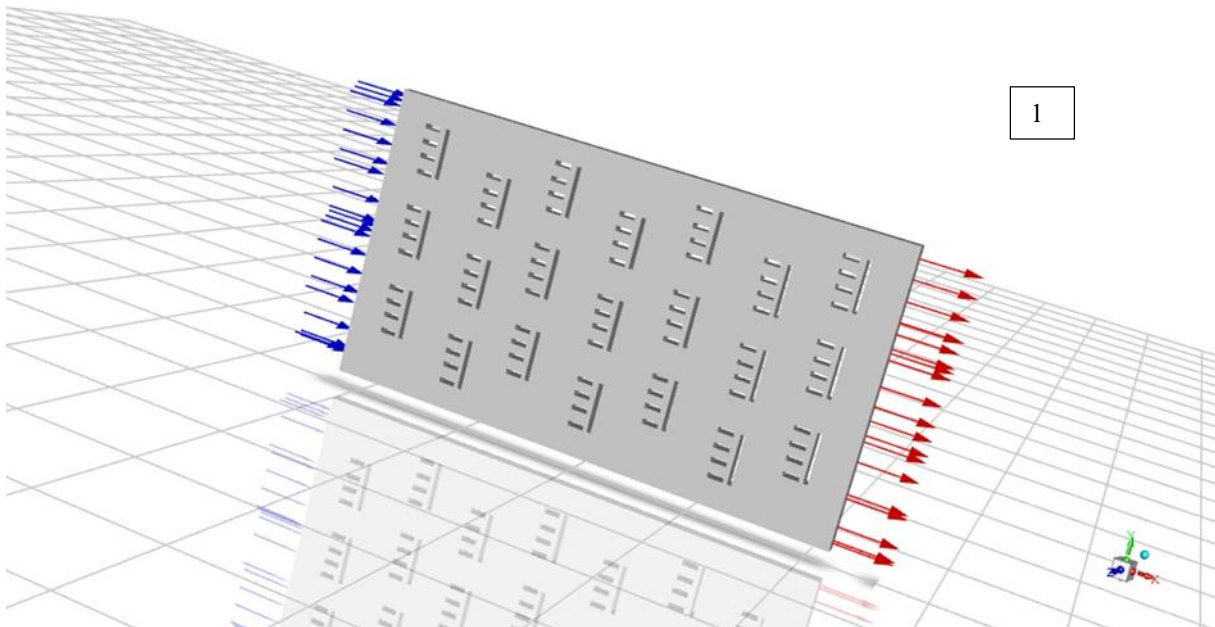
To address this trade-off between thermal performance and pressure drops, three types of air solar collectors with obstacles were studied, with the goal of proposing an optimized model that combines the best thermal efficiency with the lowest possible pressure drops.

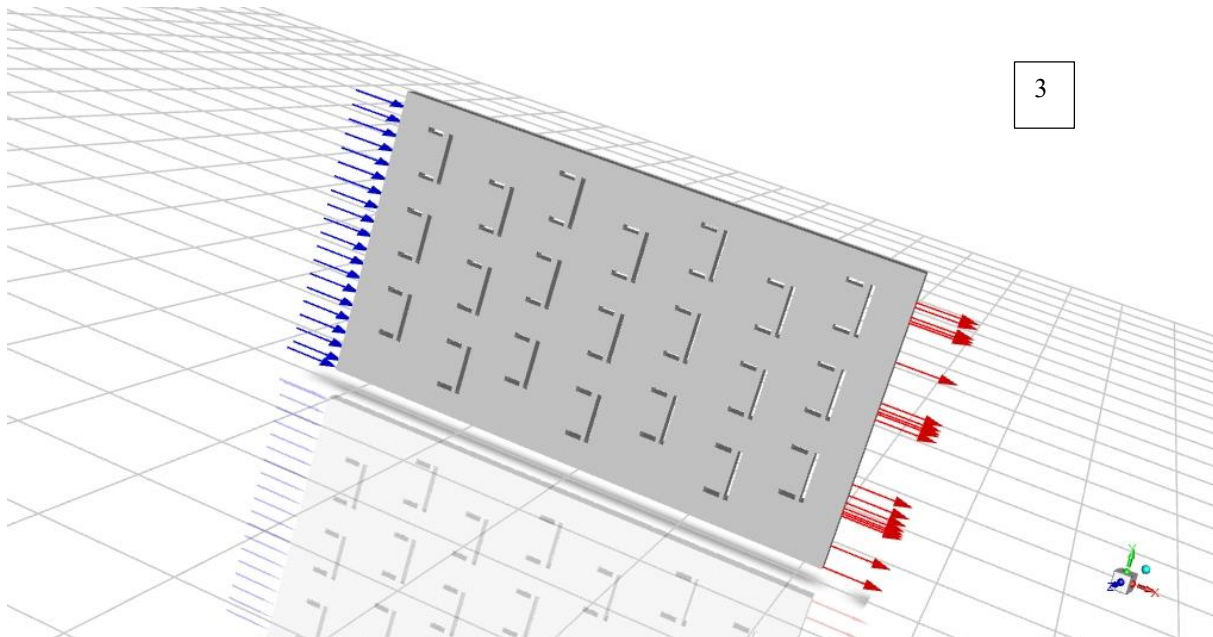
In general, air-based solar collectors with obstacles are not considered optimal under natural convection conditions. This is because the obstacles are primarily designed to increase turbulence, disrupt the boundary layer, and improve the convection coefficient. However, under natural convection, flow velocities are low, and the driving energy, generated solely by density differences, remains limited. Under these conditions, introducing obstacles can lead to a significant increase in pressure losses, potentially hindering the flow and reducing the airflow rate.

However, the results obtained in our study show that the thermal improvement factor (THEF) is greater than 1 for all three models analyzed. This means that the overall thermal improvement remains effective, even considering the increased pressure losses.

The SCTWT model exhibits the highest performance, with a maximum value of 1.3670, followed by the SCTT model (1.3331), and then the SCFT model (1.3123).

Therefore, despite the theoretical limitations related to natural convection, the use of these models can be considered acceptable from a thermo-hydraulic perspective, since the thermal gain compensates for the adverse effect of the additional pressure losses.



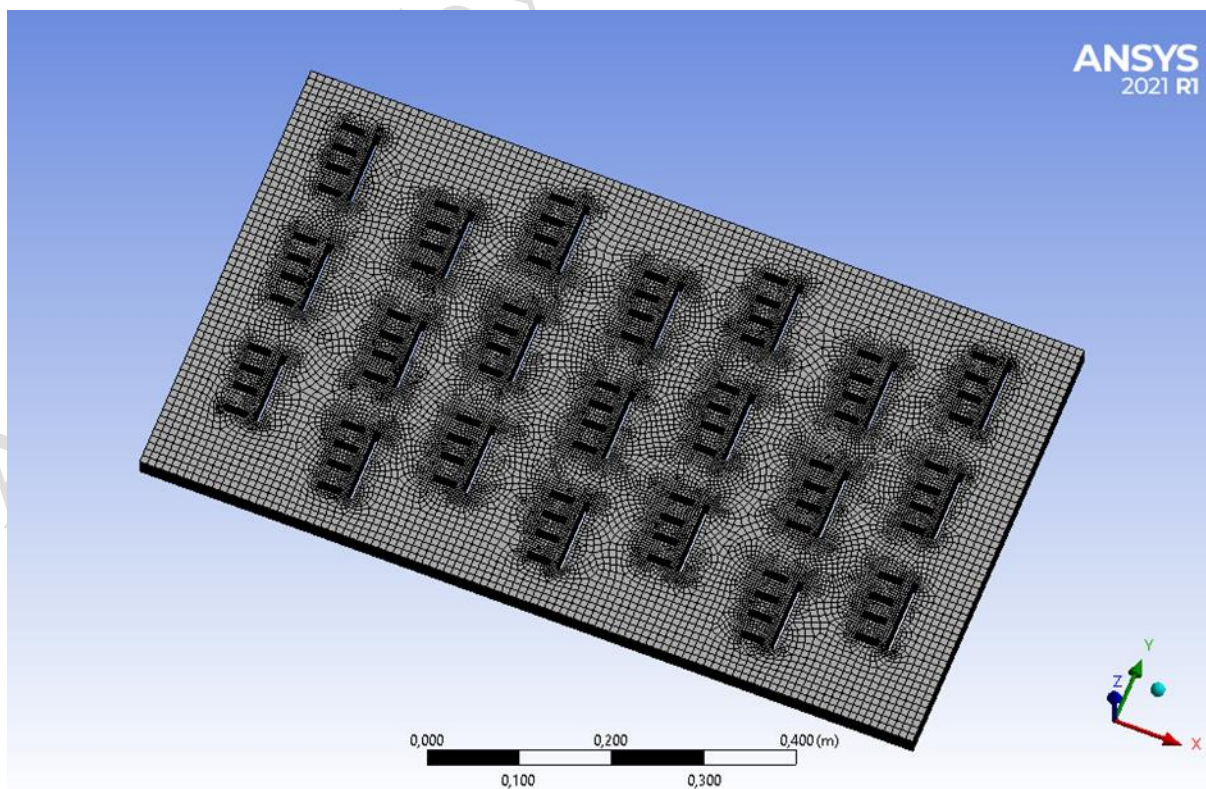


3

Figure 1. (Style name: Figure Caption) Solar collector with four-tooth obstacles (SCFT) (1), solar collector with three- tooth obstacles (SCTT) (2), and solar collector with two-tooth obstacles (SCTWT) (3).

Table 1. The mesh quality for different types of sensors.

model	Order elements	Export format	Mesh	The smoothing	Inflation option	The number of elements	the number of nodes
(SCFT)	linear	standard	simple	medium	progressive transition	182907	217770
(SCTT)	linear	standard	simple	medium	progressive transition	136881	164160
(SCTWT)	linear	standard	simple	medium	progressive transition	118521	140970



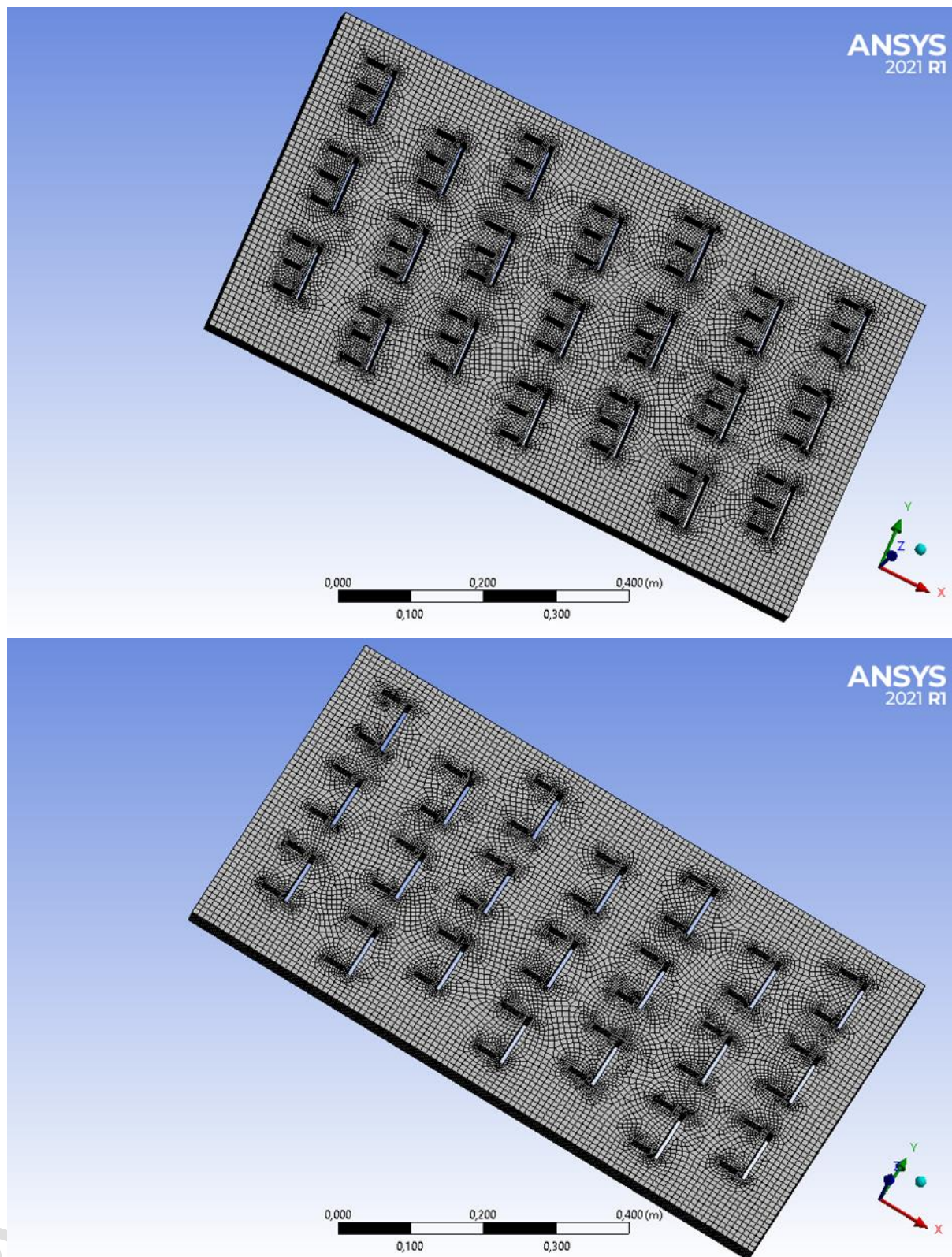


Figure 2. The mesh configurations for various sensor models

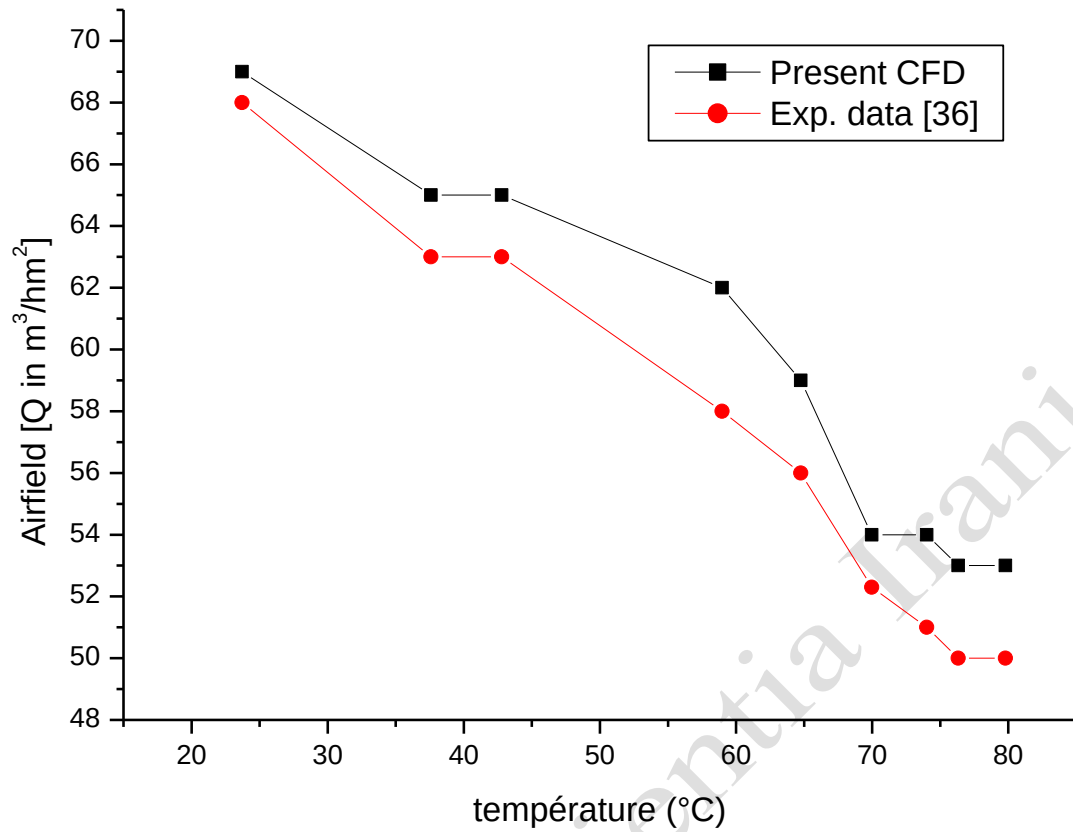
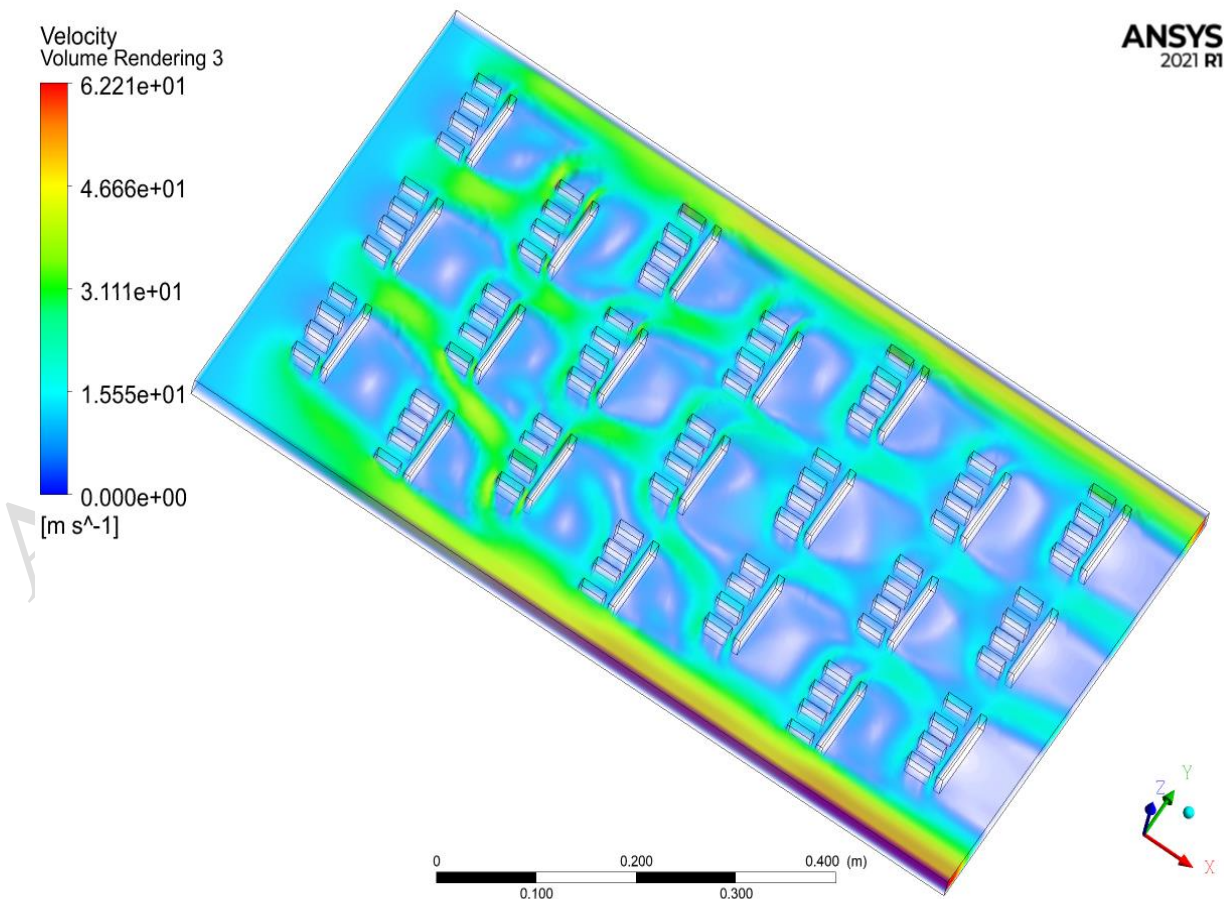


Figure 3. Validation of Moummi Abdelhafid's actual work and our simulation [20]



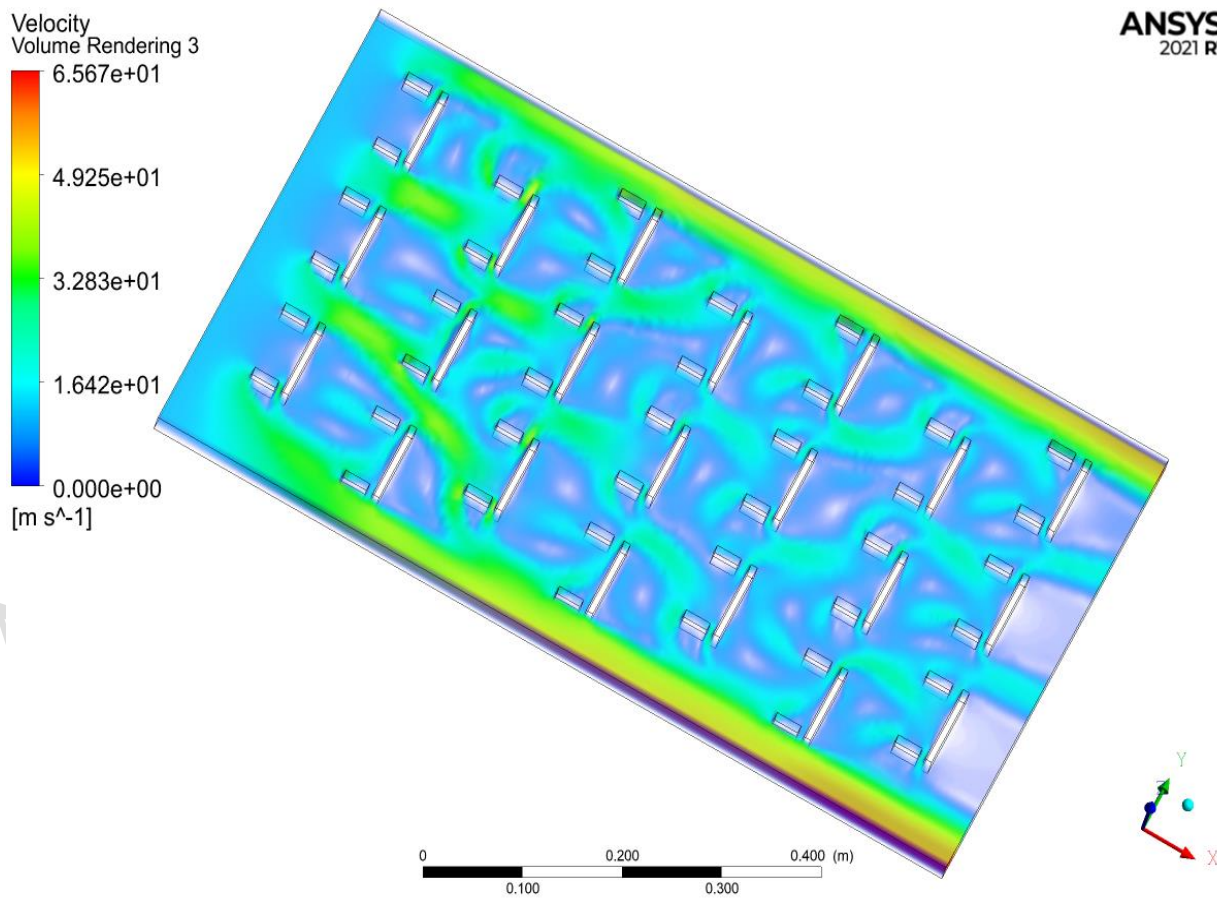
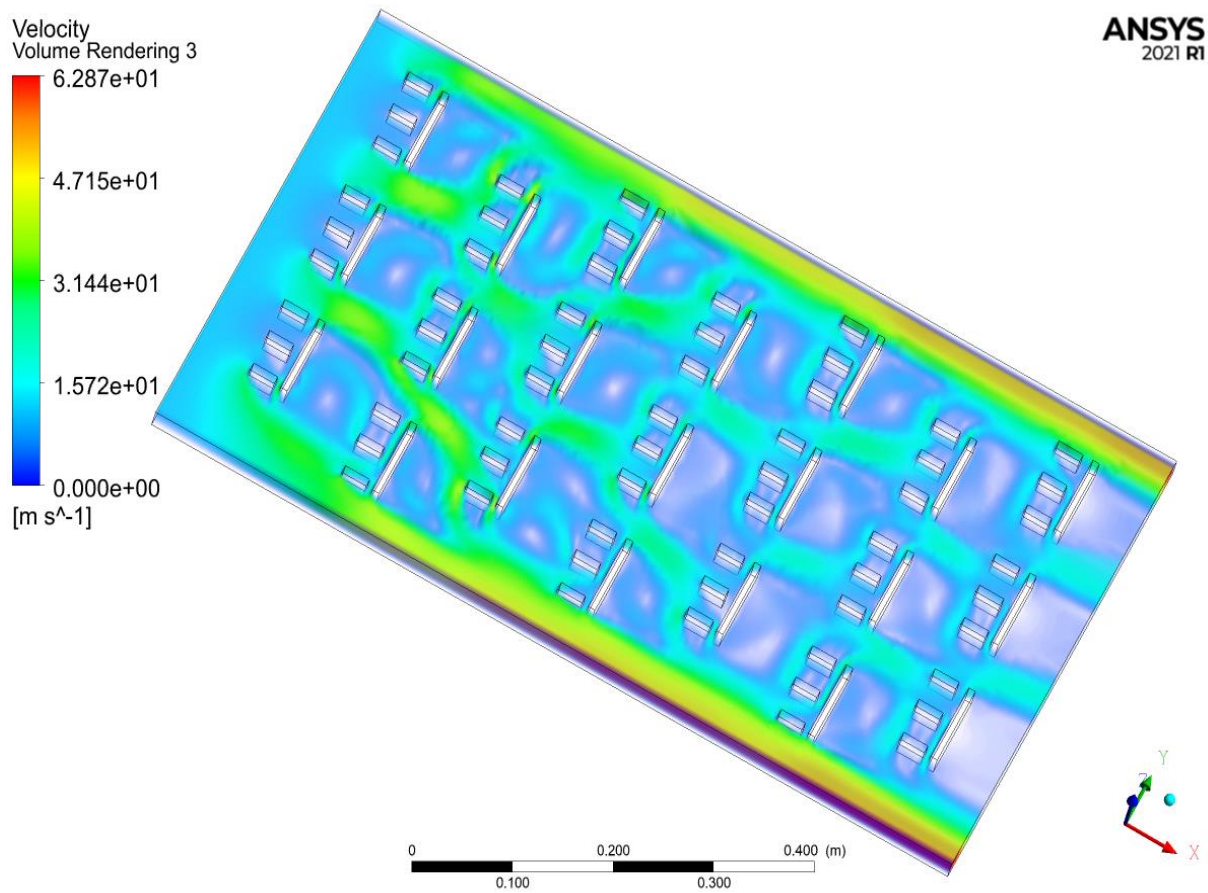
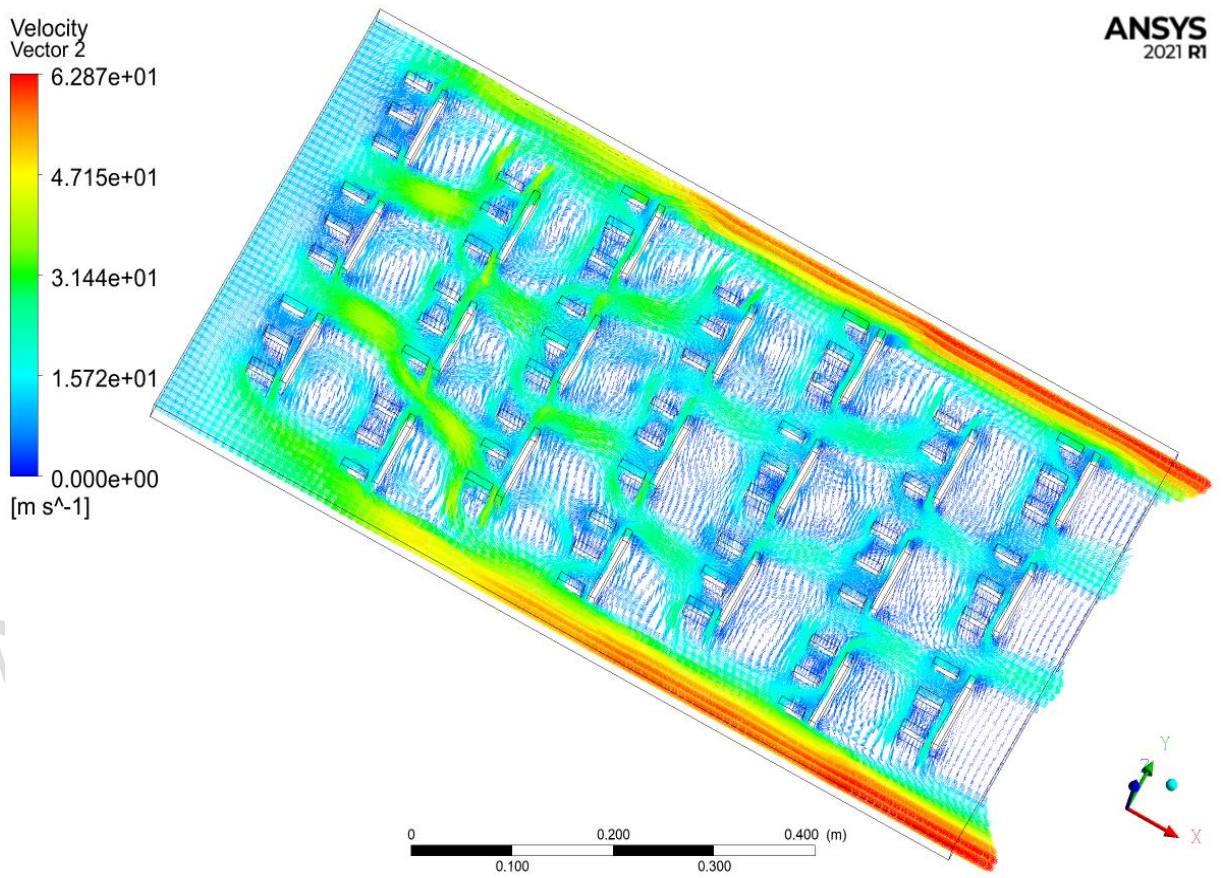
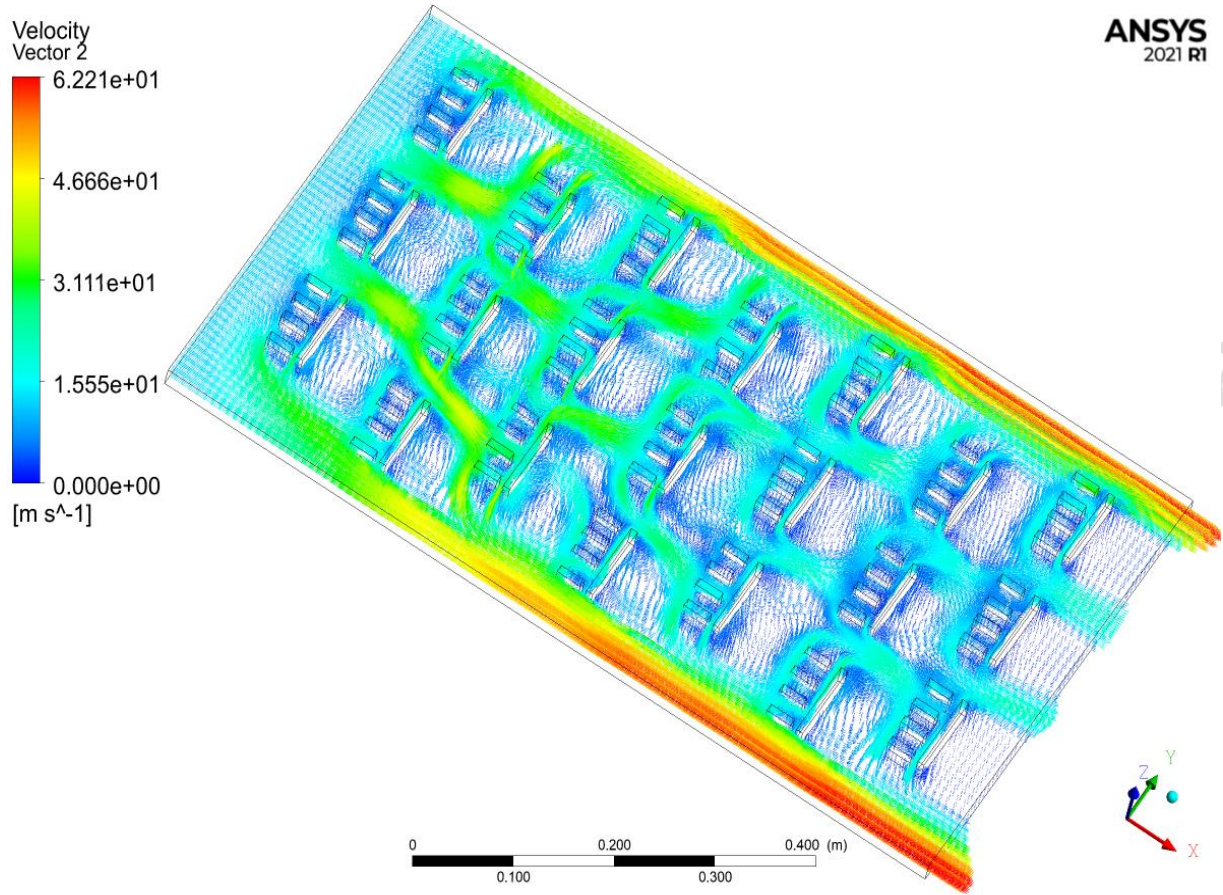


Figure 4. The distribution of speeds in the three types of sensors



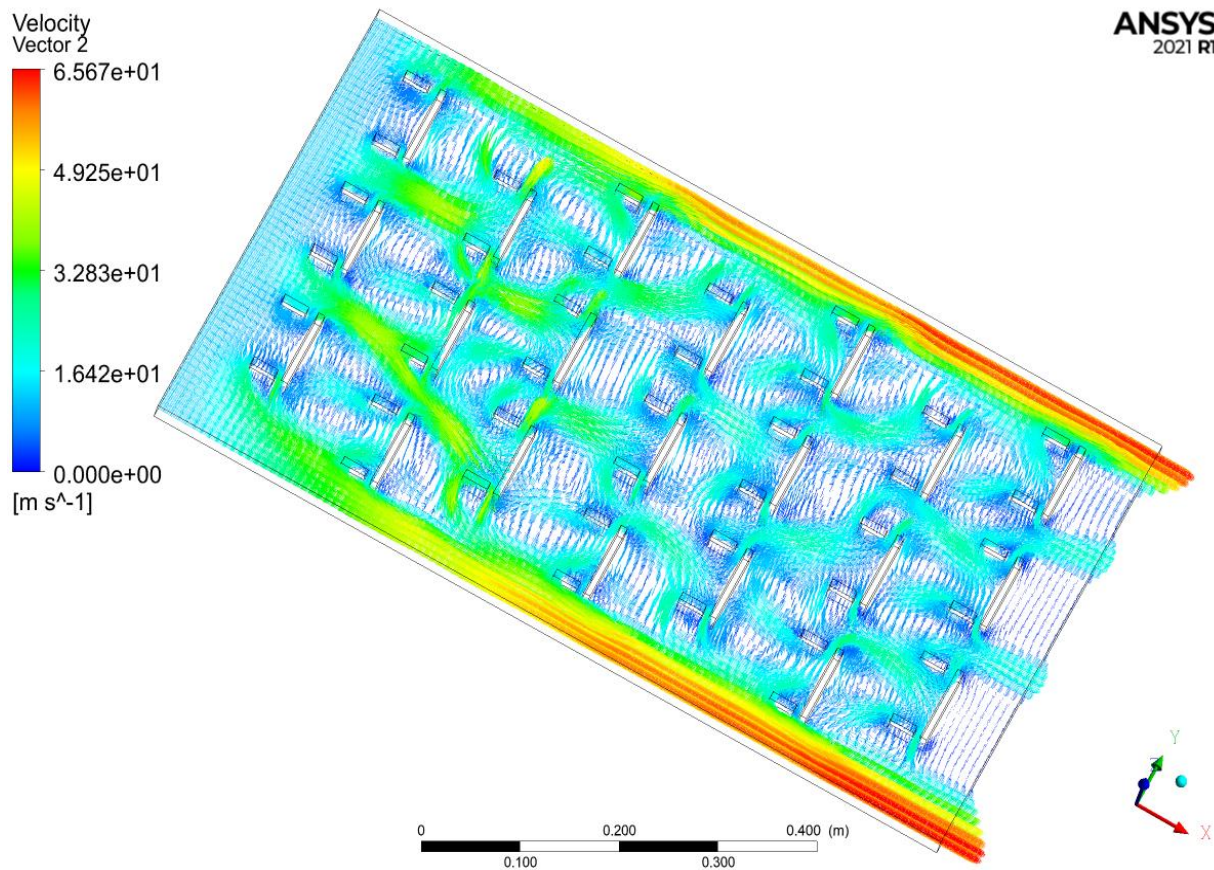
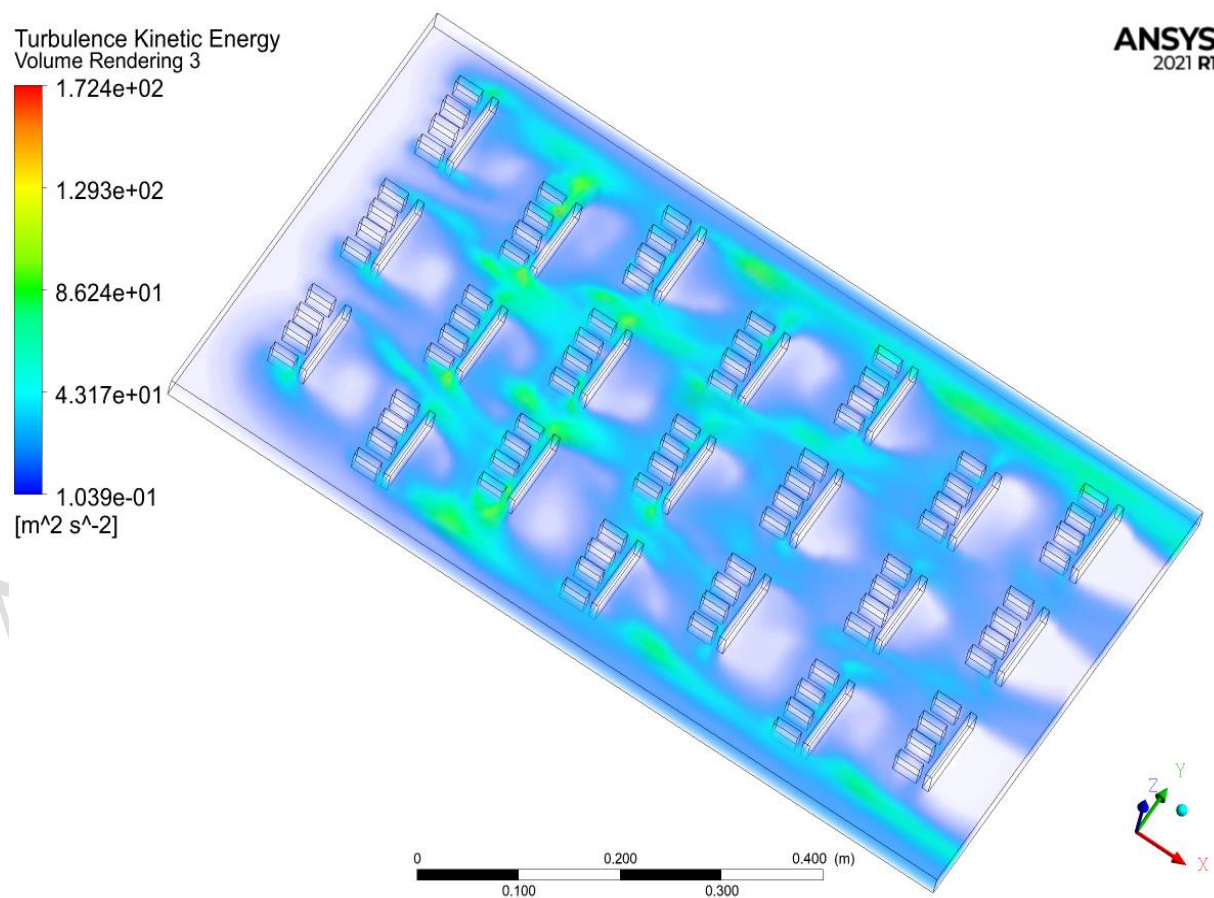


Figure 5. The velocity vectors in the air streams of the sensors



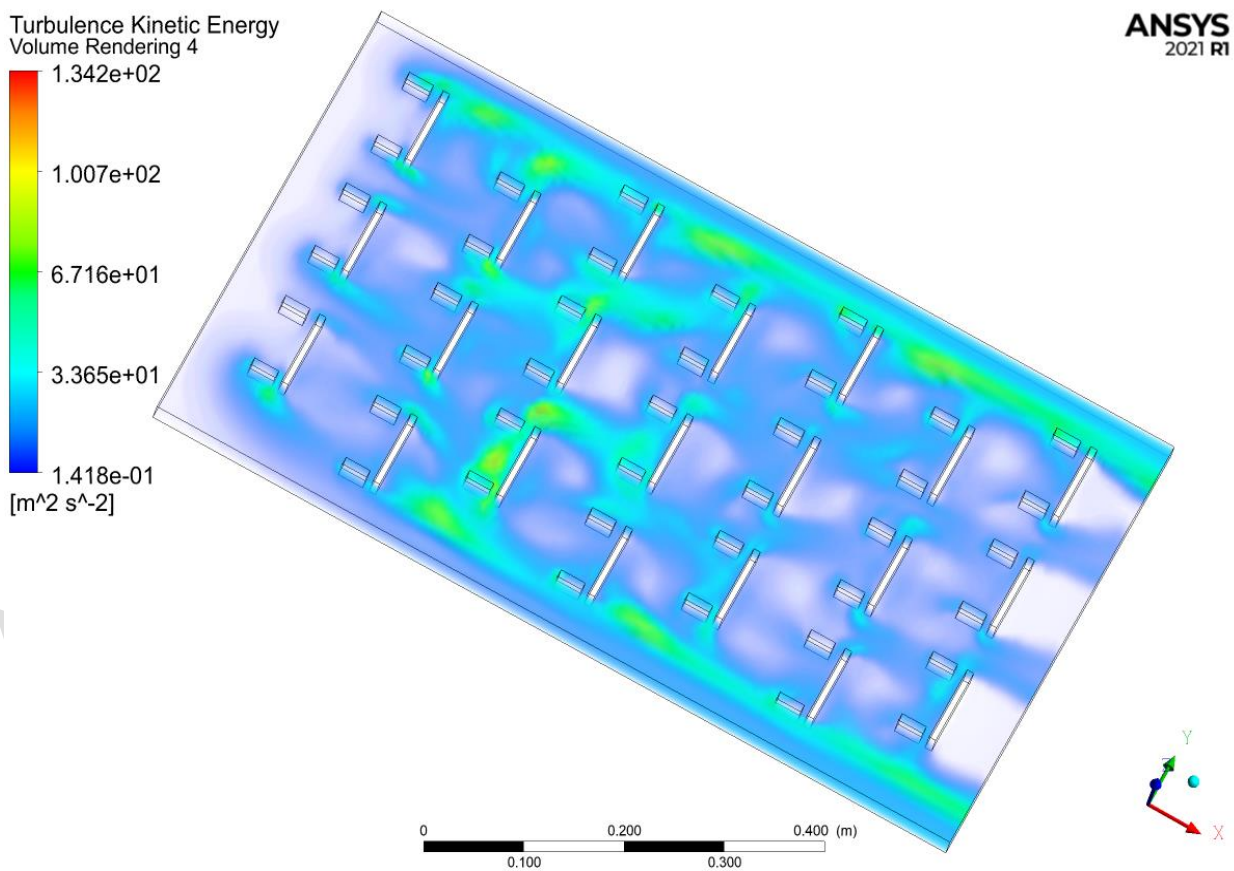
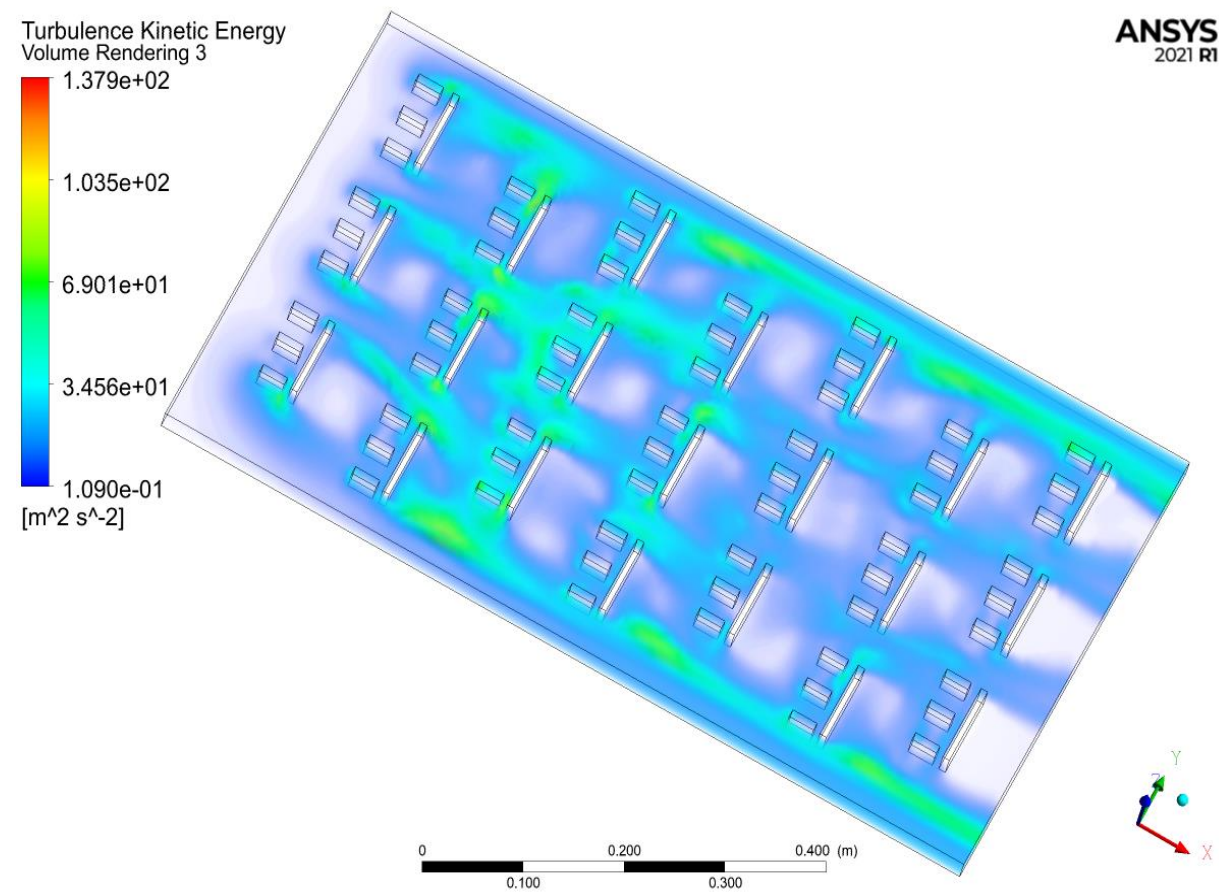
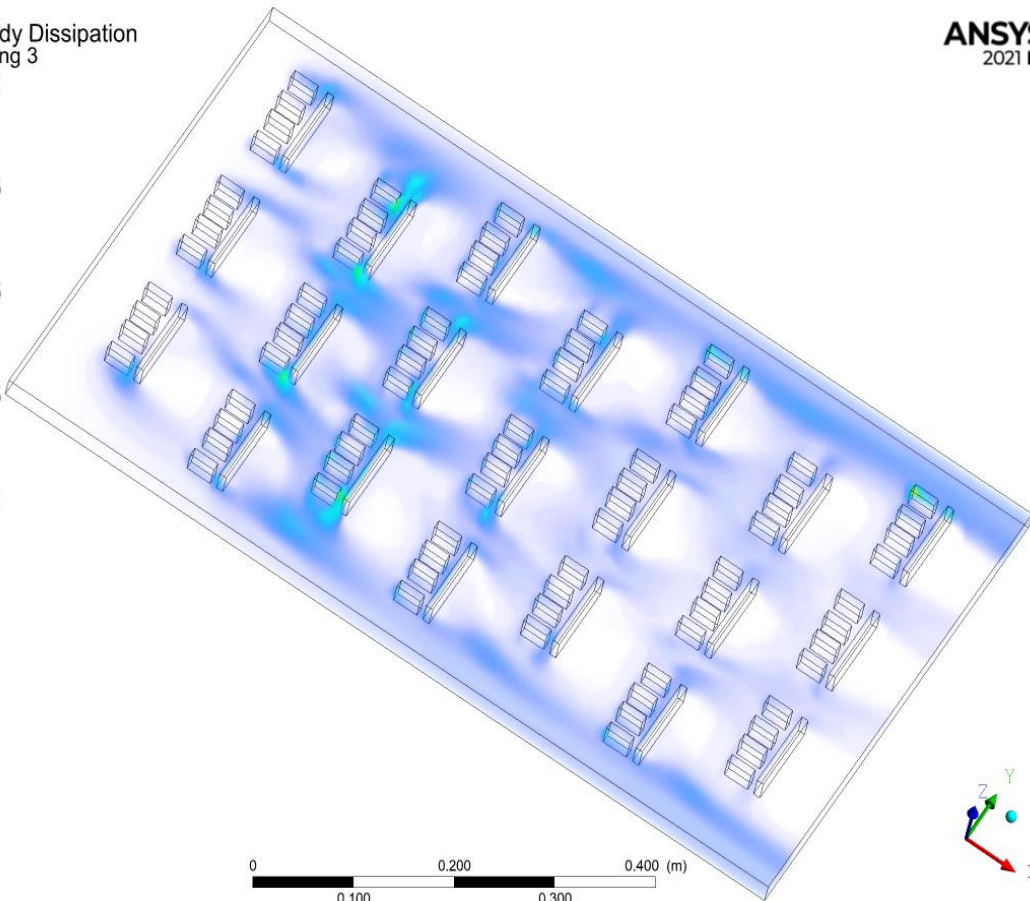
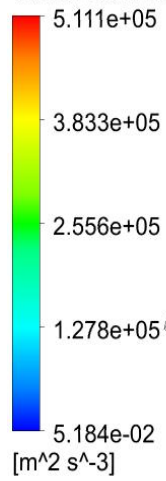
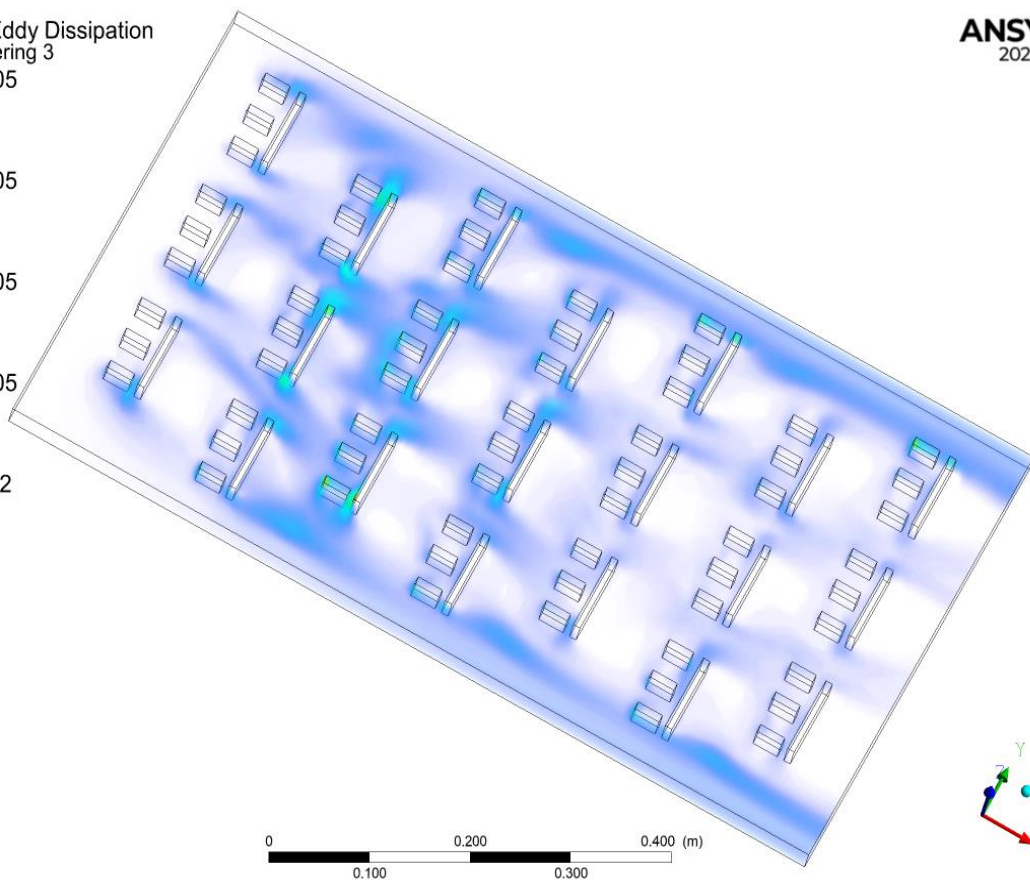
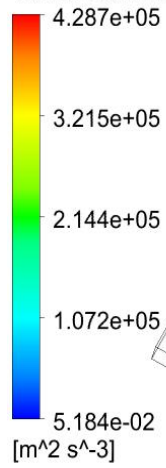


Figure 6. Turbulence exposure (k) in the air streams of the sensors

Turbulence Eddy Dissipation
Volume Rendering 3



Turbulence Eddy Dissipation
Volume Rendering 3



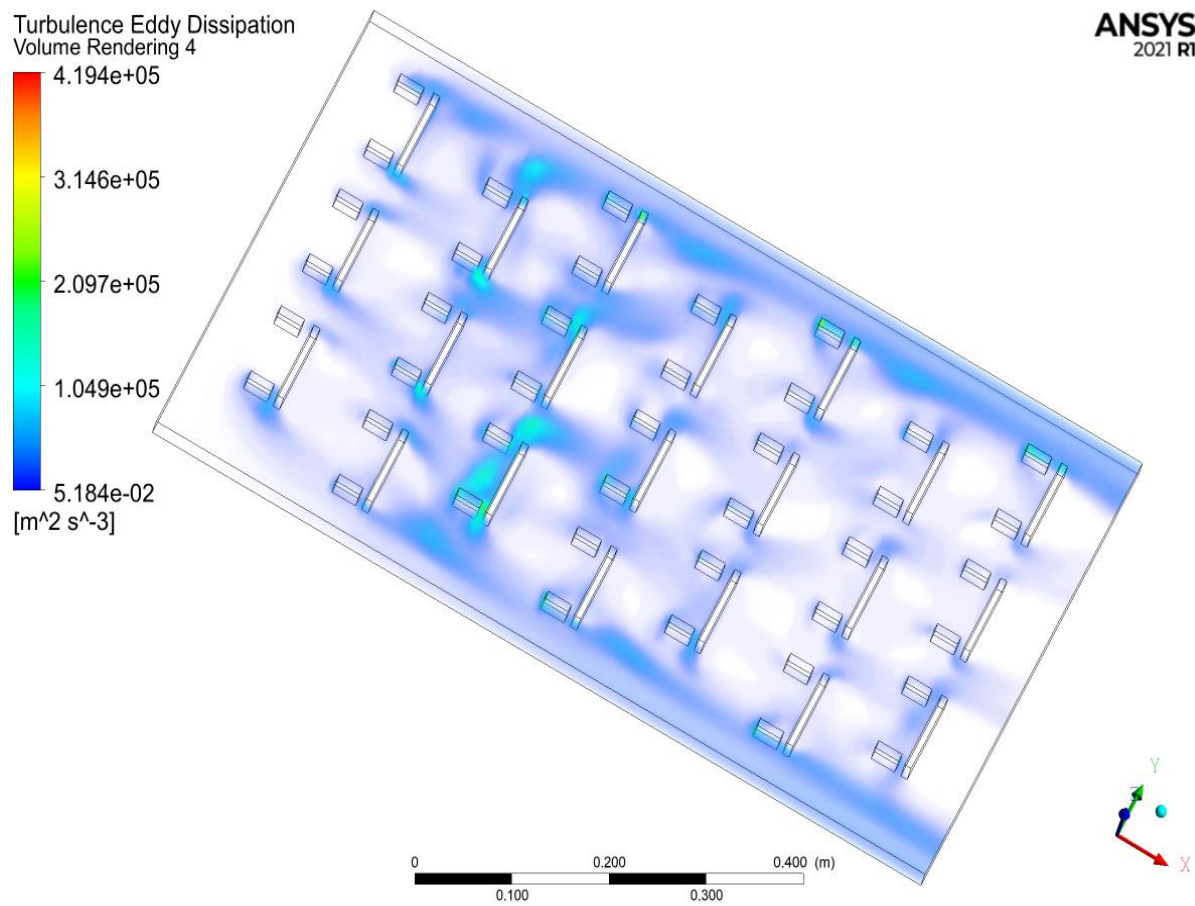
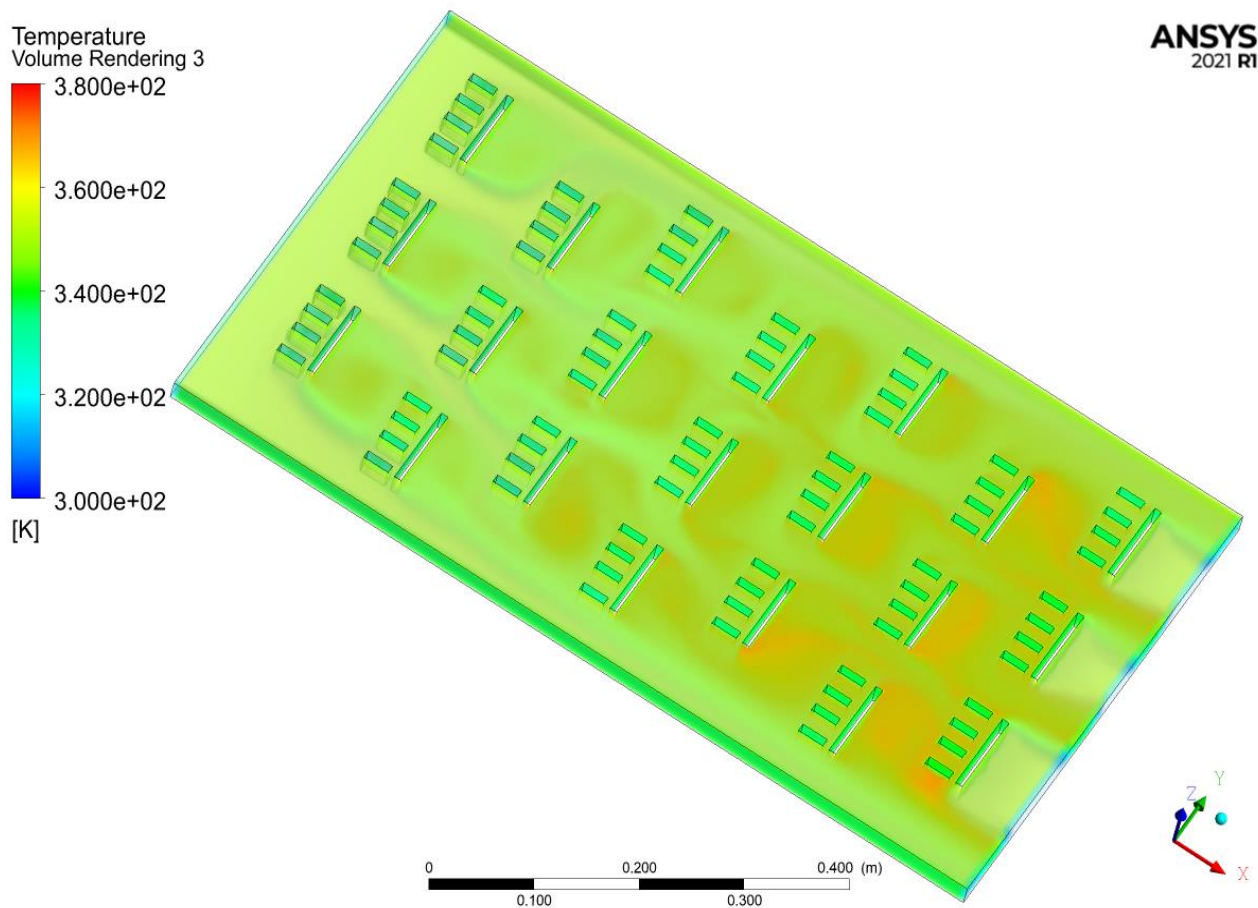


Figure 7. Turbulence exposure (ε) in the air streams of the sensors



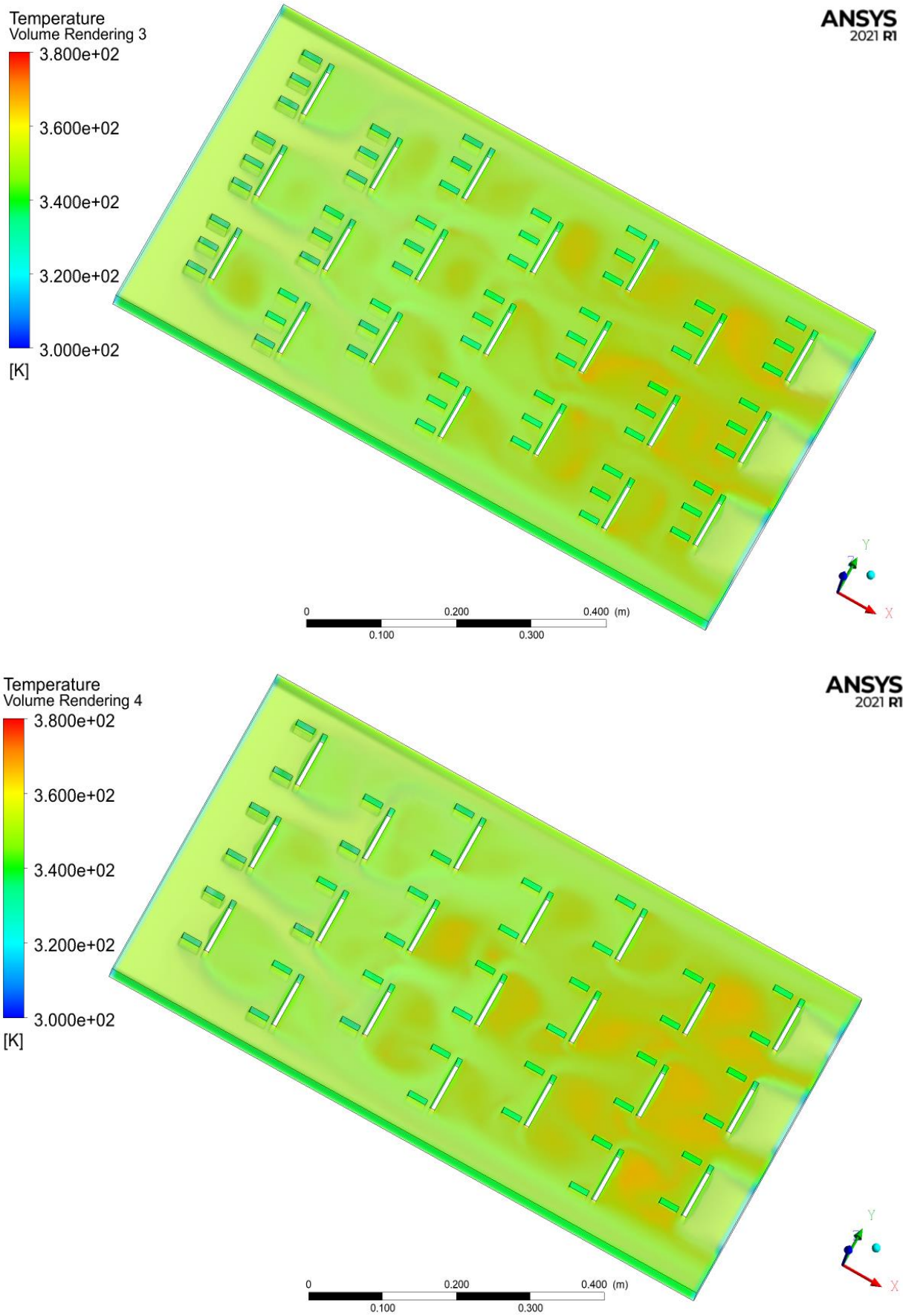


Figure 8. Temperature distribution in the air streams of the sensors

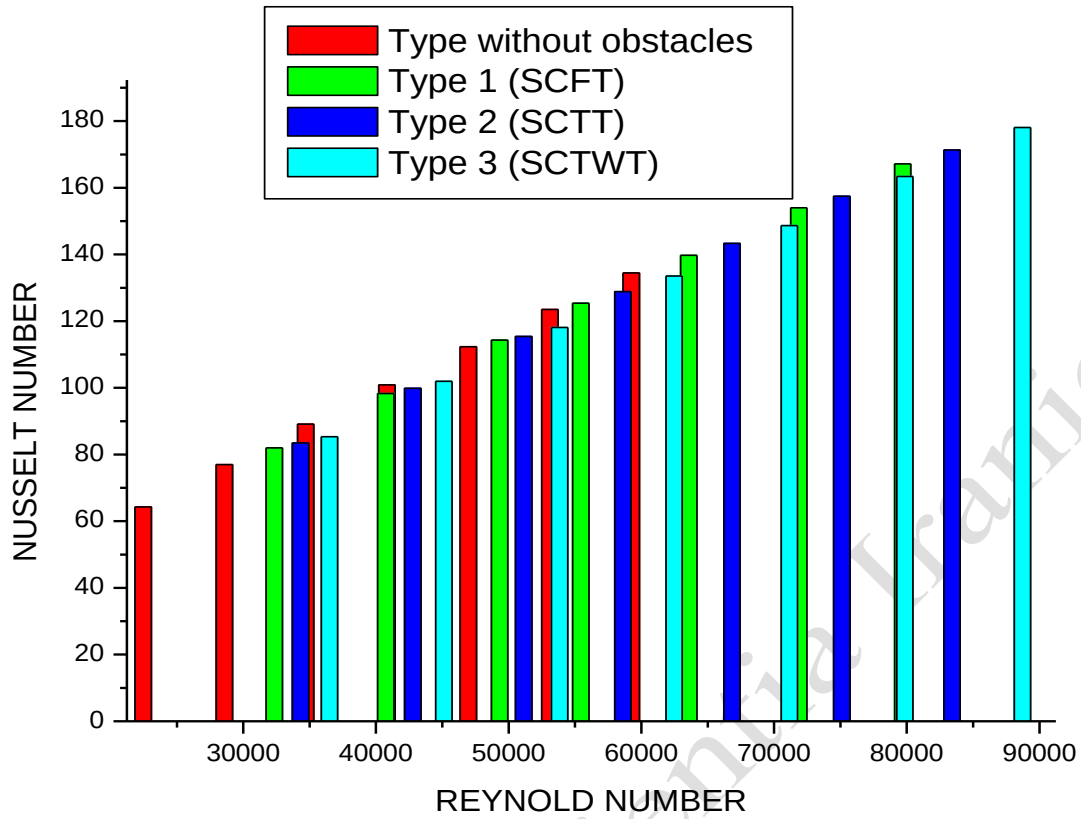


Figure 9. Nu in function Re for different sensor models

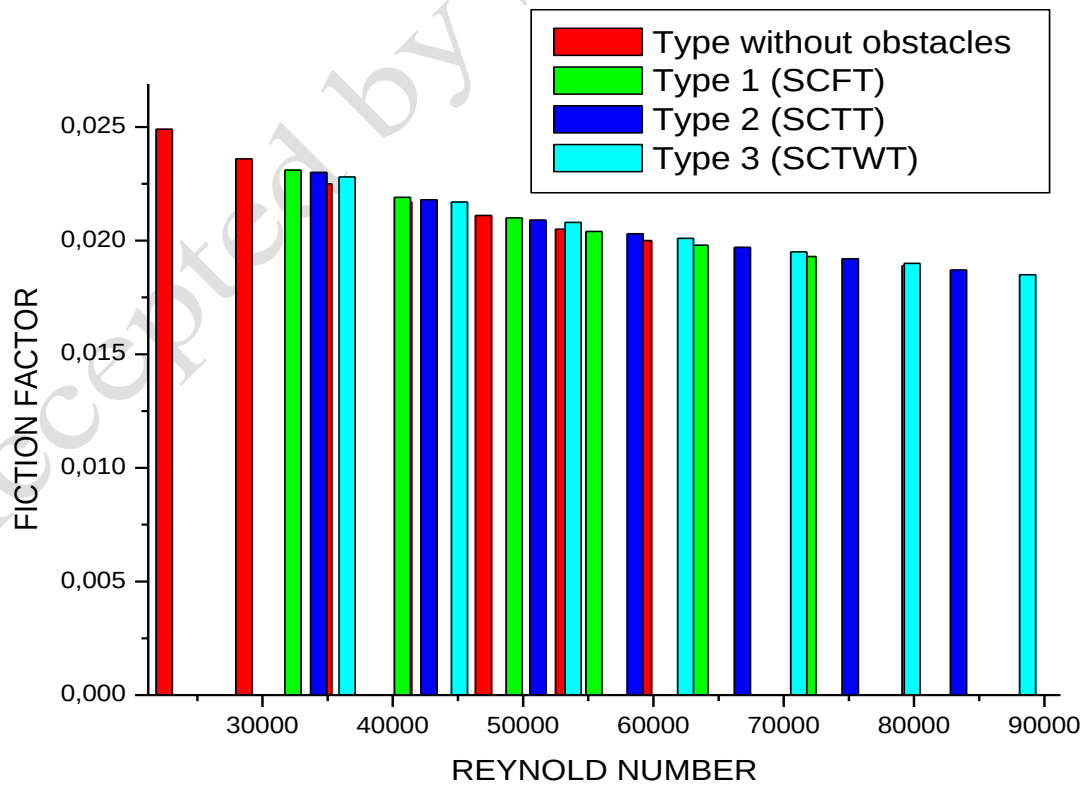


Figure 10. f in function Re for different sensor models

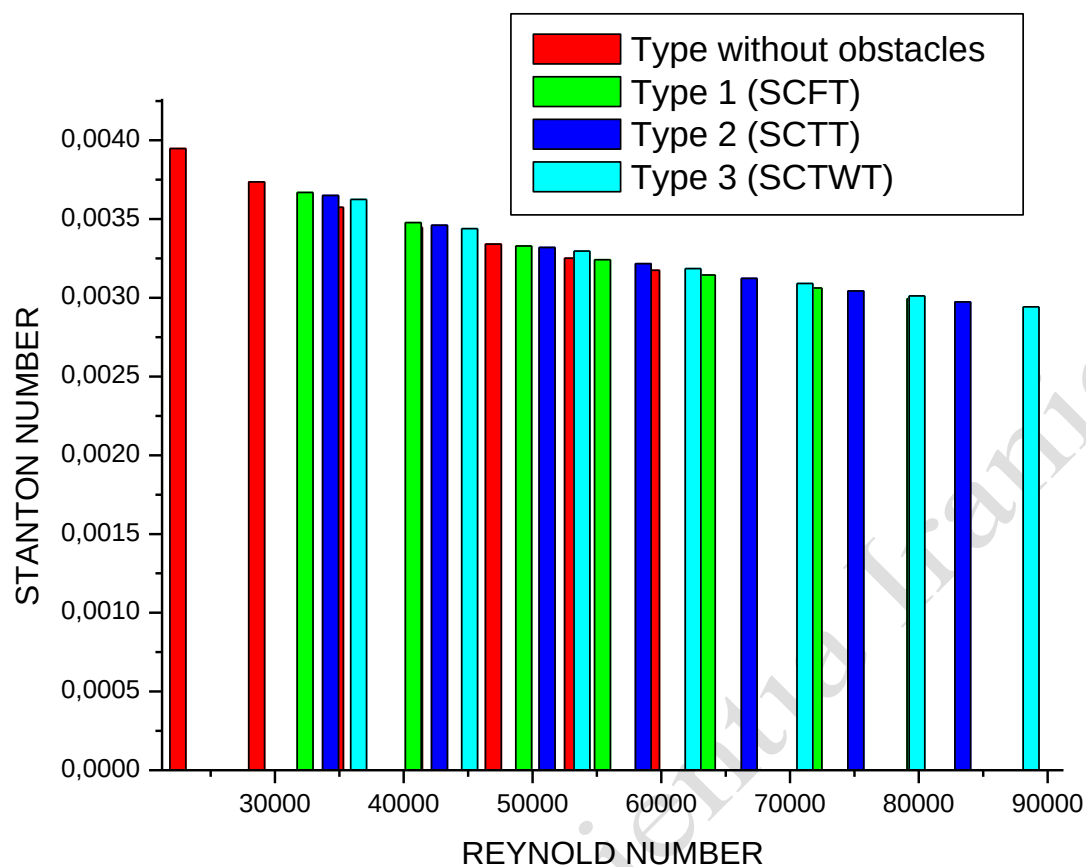


Figure 11. St in function Re for different sensor models

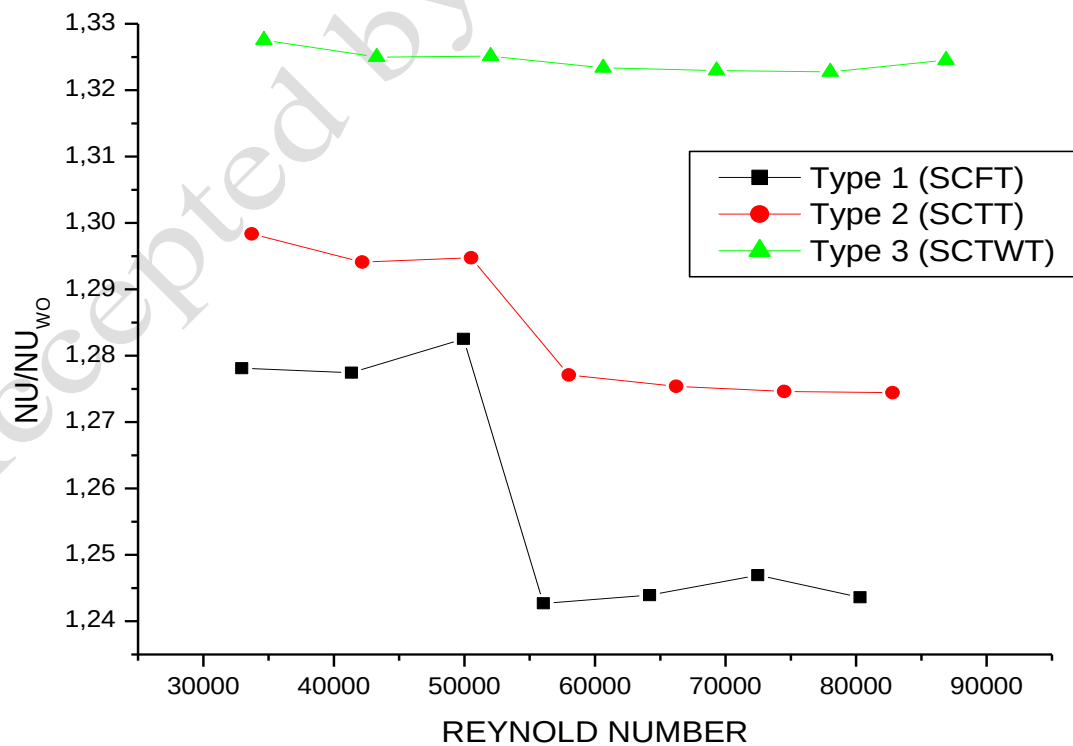


Figure 12. The ratio of Nu as a function of Re for the three sensor models

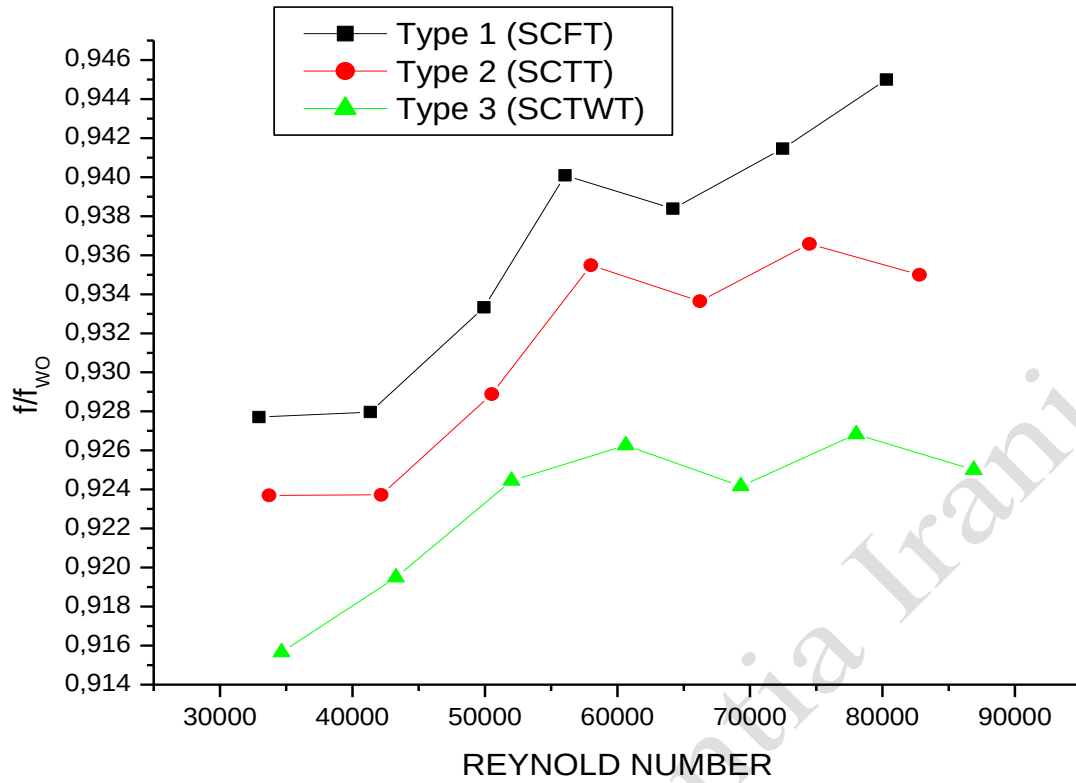


Figure 13. The ratio of f as a function of Re for the three sensor models

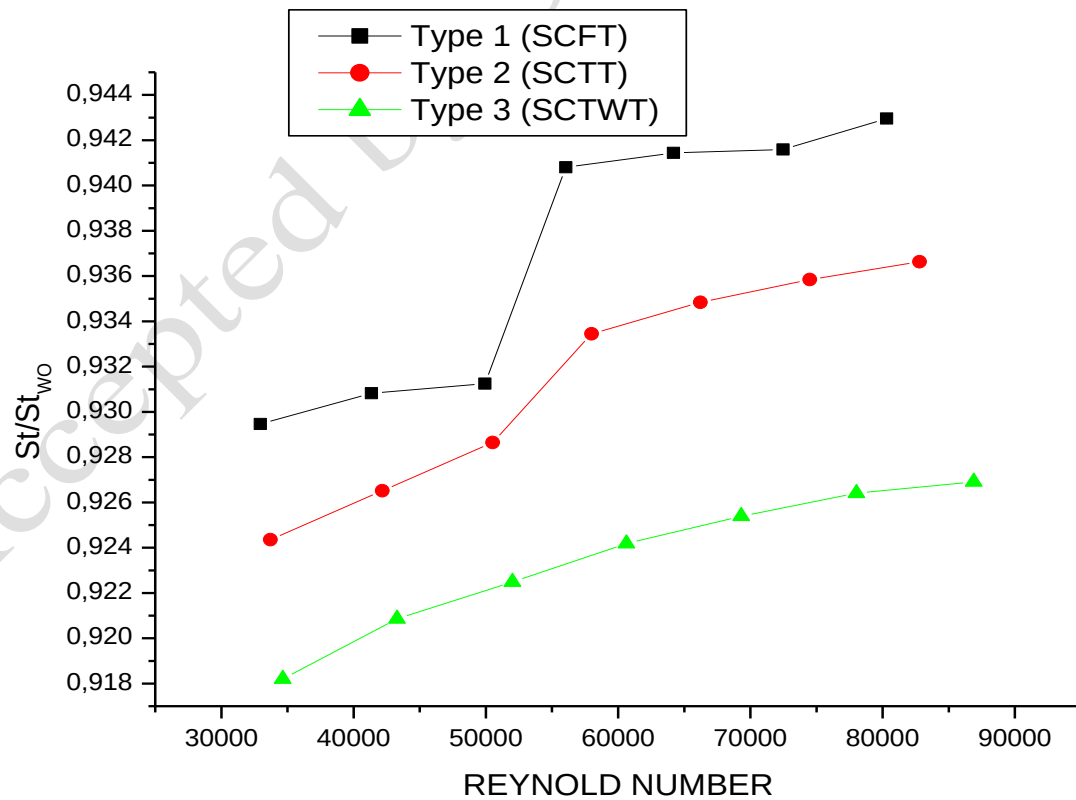


Figure 14. Re The ratio of St as a function of Re for the three sensor models

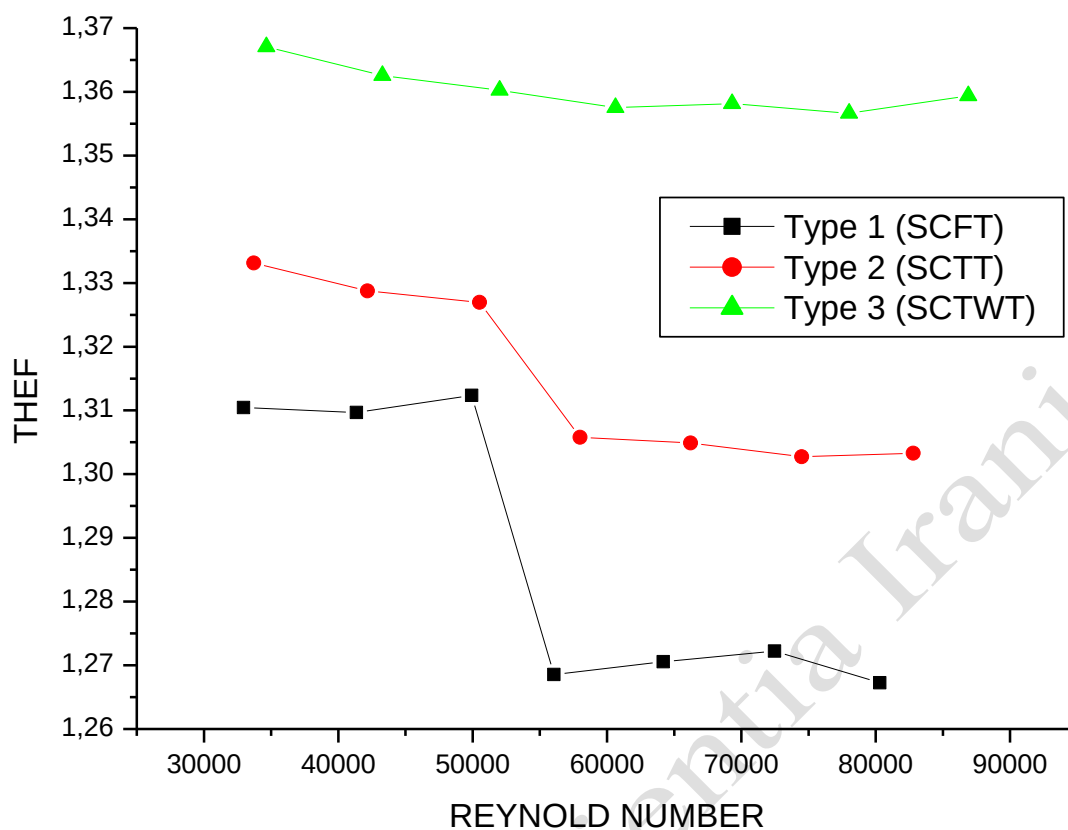


Figure 15. THEF as a function of Re for the three sensor models

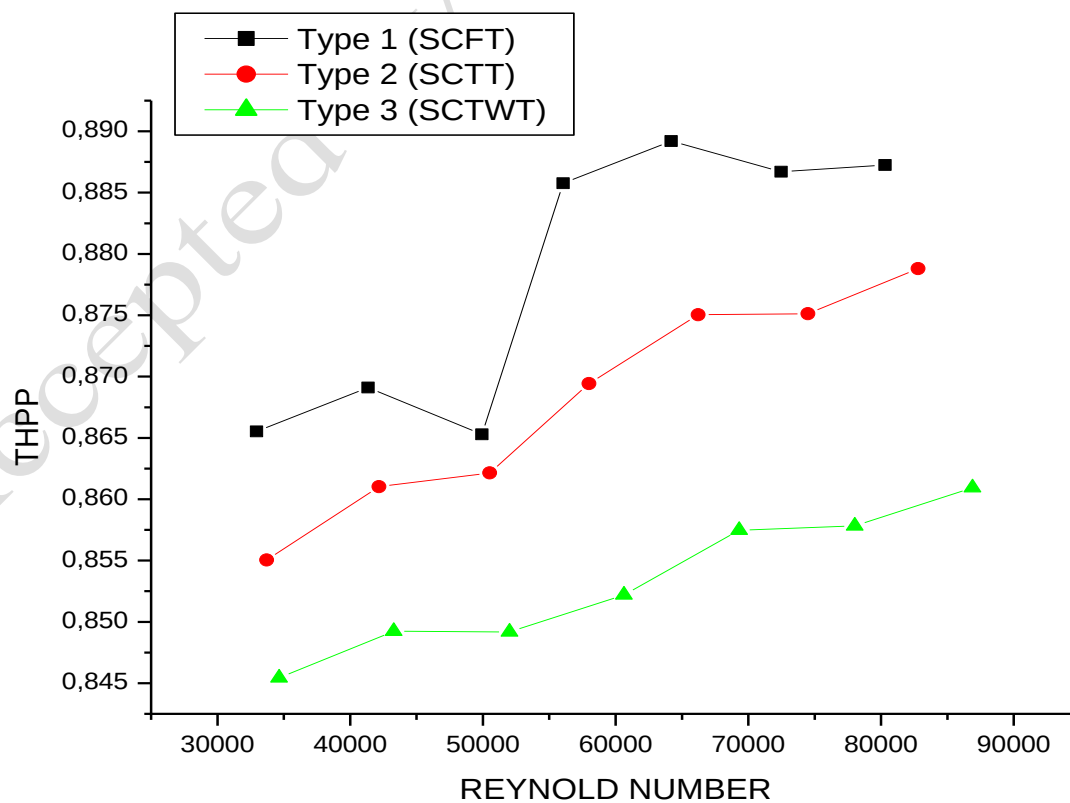


Figure 16. THPP as a function of Re for the three sensor models

Nomenclature

Mathtype	
c_p	Specific enthalpy Dimensionless heat source length Heat capacity at constant pressure, volume [J/kg.K]
$C_{1\varepsilon}, C_{2\varepsilon}$ and C_μ	Constants
dH	Hydraulic diameter of the section
E	The total energy [J]
$\vec{g}$	Gravitational acceleration [m/s ²]
G_k	Energy due to the mean velocity gradients
h'	Sensible enthalpy [energy/mass]
$\vec{j}_j$	The diffusion flux of species j
k'	The thermal conductivity [W/m.K]
k	The turbulent kinetic energy [m ² /s ²]
k_{eff}	The effective conductivity
K_P	Turbulence kinetic energy at the near-wall node P
Nu	Nusselt number of flow
p	The static pressure [pa]
Pr	Molecular Prandtl numbers
T	Temperature [K]
u_i	Flow velocity [m/s]
$\vec{v}$	Overall velocity vector [m/s]
Y_j	The mass fraction of species j
$\vec{\tau}$	The stress tensor [described below] [pa]transmittance
μ	The molecular viscosity [Pa.s]
μ_t	The turbulent (or eddy) viscosity [Pa.s]
ν	Kinematic viscosity [m ² /s]
ε	The turbulent eddy dissipation [m ² /s ³]
$\sigma_k, \sigma_\varepsilon$	The turbulent Prandtl numbers for k and ε respectively
ρ	Density [kg/m ³]

Funding

- ☐ The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Conflicts of interest

- ☐ The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

1. Struchalin, P. G, Zhao, Y., and Balakin, B. V. "Field study of adirect absorption solar collector with eco-friendly nanofluid", Applied Thermal Engineering, 243, 122652. (2024). <https://doi.org/10.1016/j.applthermaleng.2024.122652>
2. Hekal, M., El-Mghlany, W.M., Eldrainy, Y.A., et al "Hydro-thermal performance of fabric air duct (FAD): Experimental and CFD simulation assessments", Case Studies in Thermal Engineering, 47, 103107. (2023). <https://doi.org/10.1016/j.csite.2023.103107>
3. Rana, M., Nuhash, M. M., and Bhuiyan, A.A. "A CFD modelling for optimizing geometry parameters for improved performance using clean energy geothermal ground-to-air tunnel heat exchangers", Case Studies in Thermal Engineering, 53 (2024) 103867. (2019). <https://doi.org/10.1016/j.csite.2023.103867>
4. Abbas, F. A. and Alhamdo, M. H. "Thermal response of an asphalt solar collector: Numerical and experimental validation", Case Studies in Thermal Engineering. 52, 103656. (2023). <https://doi.org/10.1016/j.csite.2023.103656>
5. Ghalati, A. S., Maleki, A., Besharati, S. M. et al "Prediction and optimization of performance parameters of solar Collectors with flat and porous plates using ANN and RSM:Case Study of Shahrekord, Iran", Case Studies in Thermal Engineering, 60 (2024) 104719. <https://doi.org/10.1016/j.csite.2024.104719>

6. Hou, G., Yadav, A., Ali, E., et al. "Thermal performance improvement for parabolic trough solar collectors integrated with twisted fin and nanofluid: Modeling and validation", *Case Studies in Thermal Engineering*, <https://doi.org/10.1016/j.csite.2024.104763>
7. Peng, D., Zhang, X., Dong, H., et al. "Performance study of a novel solar air collector", *Applied Thermal Engineering*, 30, 2594-2601. (2010). <https://doi.org/10.1016/j.applthermaleng.2010.07.010>
8. Peng, W., and Sadaghiani, O. K. "Thermal performance enhancement of direct absorption solar collector using nanoparticle in sunray trap", *Applied Thermal Engineering*, 187, 116578. (2021). <https://doi.org/10.1016/j.applthermaleng.2021.116578>
9. Famiglietti, A., and Lecuona, A. "Direct solar air heating inside small-scale linear Fresnel collector assisted by a turbocharger: Experimental characterization", *Applied Thermal Engineering*, 196, 117323. (2021). <https://doi.org/10.1016/j.applthermaleng.2021.117323>
10. Simo-Tagne, M., Zoulalian, A., Rémond, R., et al "Mathematical modelling and numerical simulation of a simple solar dryer for tropical wood using a collector", *Applied Thermal Engineering*, 131, 356-369. (2018). <https://doi.org/10.1016/j.applthermaleng.2017.12.014>
11. Singh, R., Salhan, P., and Kumar, A. "CFD Modelling and Simulation of an Indirect Forced Convection Solar Dryer", *IOP Conf. Series: Earth and Environmental Science*, 795, 012008. (2021). <https://iopscience.iop.org/article/10.1088/1755-1315/795/1/012008>
12. Tong, Y., Chi, X., Kang, W., et al. "Comparative investigation of efficiency sensitivity in a flat plate solar collector according to nanofluids", *Applied Thermal Engineering*, 174, 115346. (2020). <https://doi.org/10.1016/j.applthermaleng.2020.115346>,
13. Ullah, F., and Kang, M. "Impact of air flow rate on drying of apples and performance assessment of parabolic trough solar collector", *Applied Thermal Engineering*, 127, 275-280. (2017). <https://doi.org/10.1016/j.applthermaleng.2017.07.101>
14. Ural, T., Keçebas, A., and Vahip Güler, O. "Thermodynamic performance evaluation of a heat pump system with textile based solar air heater for heating process", *Applied Thermal Engineering*, 191, 116905. (2021). <https://doi.org/10.1016/j.applthermaleng.2021.116905>
15. Zarei, A., Akhavan, S., Babaie Rabiee, M., et al "Energy, exergy and economic analysis of a novel solar driven CCHP system powered by organic Rankine cycle and photovoltaic thermal collector", *Applied Thermal Engineering*, 194, 117091. (2021). <https://doi.org/10.1016/j.applthermaleng.2021.117091>
16. Zhou, J., Yu, T., Wu, H., et al. "A dynamic model of hollow ventilated interior wall integrated with solar air collector", *Applied Thermal Engineering*, 175, 115380. (2020). <https://doi.org/10.1016/j.applthermaleng.2020.115380>
17. Zhu, T., Diao, Y., Zhao, Y., et al "Thermal performance of a new CPC solar air collector with flat micro-heat pipe arrays", *Applied Thermal Engineering*, 16, S1359-4311. (2016). <http://dx.doi.org/doi:10.1016/j.applthermaleng.2016.01.033>
18. Zhu, T., Diao, Y., Zhao, Y., et al "Performance evaluation of a novel flat-plate solar air collector with micro-heat pipe arrays (MHPA)", *Applied Thermal Engineering*. (2017). <http://dx.doi.org/10.1016/j.applthermaleng.2017.02.076>
19. Amraoui, M., A., and Benosman, F. "Computational 3D Simulations of Fluid Dynamics and Comparisons of Heat Transfer Characteristics of Various Flat-Plate Solar Collectors", *Iranian Journal of Science and Technology, Transactions of Mechanical Engineering*, 48, 2113-2129. (2024). <https://doi.org/10.1007/s40997-024-00766-x>
20. Amraoui, M., A., and Aliane, K. "Three-dimensional Analysis of Air Flow in a Flat Plate Solar Collector", *Periodica Polytechnica Mechanical Engineering*, 62(2), 126-135. (2018). <https://doi.org/10.3311/PPme.11255>
21. Amraoui, M., A., and Aliane, K. "Numerical analysis of a three dimensional fluid flow in a flat plate solar collector", *International Journal of Renewable and Sustainable Energy*, 3(3), 68-75. (2014). <https://sciencepg.com/article/10.11648/j.ijrse.20140303.14>
22. Amraoui, M., A., and Aliane, K. "Dynamic and Thermal Study of the Three-Dimensional Flow in a Flat Plate Solar Collector with Transversal Baffles", *International Review of Mechanical Engineering (I.R.E.M.E.)*. (2014). <https://doi.org/10.15866/ireme.v8i6.1623>
23. Amraoui, M., A. "Numerical Study of an Air Flow in a Flat Plate Air Solar Collector with Circular Obstacles", *Artificial Intelligence and Renewables Towards an Energy Transition*, 174, 1-8, (2021). https://doi.org/10.1007/978-3-030-63846-7_81
24. Amraoui, M., A., and Benosman, F. "Numerical modeling of a flat air solar collector fitted with obstacles", *E3S Web of Conferences*, 321, 04016. (2021). <https://doi.org/10.1051/e3sconf/202132104016>
25. Amraoui, M., A., Alkhafaji, M., A., Abdullaev, S., et al. "CFD Study of a dual-passage solar collector with longitudinal and transverse baffles for enhanced thermal performance", *Thermal Science*, 27, 3133-3142. (2023). <https://thermalscience.rs/2023/4/100>
26. Amraoui, M., A. "Three-Dimensional Numerical Simulation of a Flat Plate Solar Collector with Double Paths, *International Journal of Heat and Technology*, 39 No. 4, 1087-1096, (2021). <https://doi.org/10.18280/ijht.390406>
27. Arkhazloo, N., B., Bouissa, Y., Bazdidi-Tehran, F., et al. "Experimental and unsteady CFD analyses of the heating process of large size forgings in a gas-fired furnace", *Case Studies in Thermal Engineering*, 14, 100428. (2019). <https://doi.org/10.1016/j.csite.2019.100428>
28. Zheng, X., Hu, F., Wang, et al "Optimization airflow distribution for anti-condensation of high-voltage switchgear using CFD method", *Case Studies in Thermal Engineering*, 28, 101479. (2021). <https://doi.org/10.1016/j.csite.2021.101479>
29. Boonloi, A., and Jedsadaratanachai, W. "CFD analysis on heat transfer characteristics and fluid flow structure in a square duct with modified wavy baffles", *Case Studies in Thermal Engineering*, 29, 101660. (2022). <https://doi.org/10.1016/j.csite.2021.101660>
30. Xu, T., Zhao, D., Tao, H., et al. "Extended CFD models for numerical simulation of tunnel fire under natural ventilation: Comparative analysis and experimental verification", *Case Studies in Thermal Engineering*, 31, 101815. (2022). <https://doi.org/10.1016/j.csite.2022.101815>

31. Razzagh, M., J. P., ghassabian, M., Daemiashkezar, M., et al. "Thermo-hydraulic performance evaluation of turbulent flow and heat transfer in a twisted flat tube: A CFD approach", *Case Studies in Thermal Engineering*, 35, 102107. (2022). <https://doi.org/10.1016/j.csite.2022.102107>
32. Shafieian, A., Bahrani, H., R., Roostae, A., et al. "Effects of different inlet configurations on the performance of solar storage tanks: A three-dimensional unsteady CFD simulation", *Case Studies in Thermal Engineering*, 45, 103019. (2023). <https://doi.org/10.1016/j.csite.2023.103019>
33. Wei, D., Basem, A., Alizadeh A., et al. "Optimum tilt and azimuth angles of heat pipe solar collector, an experimental approach", *Case Studies in Thermal Engineering*, 55, 104083, (2024). <https://doi.org/10.1016/j.csite.2024.104083>
34. Goudarzi K., Ramazani M., and Pournaderi S.P. "Numerical and experimental investigation of the effect of radiant reflector on the conical solar collector for heat transfer enhancement", *Case Studies in Thermal Engineering*, 56, 104292. (2024). <https://doi.org/10.1016/j.csite.2024.104292>
35. Liu W., Liu X., Chow T., et al. "Thermal characteristics of heat-pipe-ring embedded building façades as both building-integrated and building-attached solar collectors", *Case Studies in Thermal Engineering*, 60, 104621. (2024). <https://doi.org/10.1016/j.csite.2024.104621>
36. Abdelhafid, M., Global and Local Study of the role of geometry in optimizing solar air flat plate collectors : Phd Thesis, University of Valenciennes and Hainaut Cambresis France. (1994).

Figure 1. (Style name: Figure Caption) Solar collector with four-tooth obstacles (SCFT) (1), solar collector with three-tooth obstacles (SCTT) (2), and solar collector with two-tooth obstacles (SCTWT) (3).

Table 1. The mesh quality for different types of sensors.

Figure 2. The mesh configurations for various sensor models

Figure 3. Validation of Moumami Abdelhafid's actual work and our simulation [20]

Figure 4. The distribution of speeds in the three types of sensors

Figure 5. The velocity vectors in the air streams of the sensors

Figure 6. Turbulence exposure (k) in the air streams of the sensors

Figure 7. Turbulence exposure (ε) in the air streams of the sensors

Figure 8. Temperature distribution in the air streams of the sensors

Figure 9. Nu in function Re for different sensor models

Figure 10. f in function Re for different sensor models

Figure 11. St in function Re for different sensor models

Figure 12. The ratio of Nu as a function of Re for the three sensor models

Figure 13. The ratio of f as a function of Re for the three sensor models

Figure 14. Re The ratio of St as a function of Re for the three sensor models

Figure 15. THEF as a function of Re for the three sensor models

Figure 16. THPP as a function of Re for the three sensor models

Biographies

- Mr. A. Amraoui was born on January 22, 1983, in Tlemcen, Algeria. He is currently a professor at Djillali Liabes University in Sidi-Bel-Abbès. He earned a doctorate in mechanics from Abou Bekr Belkaid University in Tlemcen in 2017 and, two years later, received a Ph.D. in mechanics from Djillali Liabes University. He is a recognized authority in the field of thermal engineering, with a notable specialization in solar energy systems. His research focuses on the mechanisms of heat and momentum transfer in complex geometries.