

# Numerical Analysis of a Solar-Powered Tube Heater

Hadi Tannous<sup>1</sup>, Valentina Stojceska<sup>1</sup>, Savvas Tassou<sup>1</sup>

<sup>1</sup>Brunel University London

Kingston Lane Uxbridge, Middlesex, London, UK

Hadi.Tannous@brunel.ac.uk; Valentina.Stojceska@brunel.ac.uk; Savvas.Tassou@brunel.ac.uk

**Abstract** – This paper evaluates the performance of a solar-powered tube heater that uses air impingement jets to heat steel tubes in the powder-based coating process. To evaluate the thermal performance of the tube heater, two numerical models, ANSYS *FLUENT Dynamic Mesh (FDM)* and ANSYS *FLUENT Transient Thermal (FTT)*, were developed and their accuracy and computational efficiency were compared. The FDM model analyzed the heat transfer in the tube heater by simulating a moving steel tube with a steady heat source while the FTT model by simulating a steady steel tube with a moving heat source. Results showed the FDM model to be computationally more time and cost-efficient, requiring 4 processors and 4 days to run compared to the FTT model which required 40 processors and 31 days. On the other hand, the FDM model showed a more detailed temperature contour of the tube with higher temperatures on the edges due to air crossflow. However, this did not have any significant effect on the final average temperature of the tube which was found to reach 76°C by both models, consequently reducing the required load of the induction heater by 22% and the Greenhouse gas (GHG) emissions of the induction heater by 2.15 gCO<sub>2</sub>e/m.

**Keywords:** Air Impingement Jets - Dynamic Mesh – CFD – Heat Transfer – Solar Heating in Industrial Processes – Steel Tube Industry

## 1. Introduction

Industries are responsible for 37% of the world's greenhouse gas (GHG) emissions due to their use of fossil fuels to meet the heat and electricity demand of their processes [1]. Governments have set targets to achieve net-zero emissions by 2050 [2] which encouraged industries to increase the use of renewable energy technologies, particularly solar thermal energy which has the ability to reduce GHG emissions [3]. Currently, 635 solar thermal energy collectors that operate at a temperature of up to 400 °C are integrated into industrial processes producing approximately 441 MWh of heat annually [4]. However, only 10% of these collectors are employed in high-consuming industries like cement, ceramics, chemicals, iron and steel while the rest are integrated into lower energy-consuming processes [5]. The iron and steel industry was the second-highest energy consumer in EU exceeding 550 TWh in 2015 [6]. Its processes require 92% of their heat at high temperatures above 400 °C and 4.5% (16.6 TWh in EU) at medium temperatures of 150-400 °C [3]. To the author's best knowledge, no solar thermal energy systems have been integrated to this industry despite the available solar thermal technology for the processes of medium temperature [7], [8].

One of the processes in the iron and steel industry that can be considered for the application of solar thermal energy is the powder-based coating process of steel tubes. It uses electric-based induction heaters to heat the tubes to a high temperature of 240 °C as they move axially before they are powder coated [3]. This leads to high electricity consumption and increases the environmental impact of the industry [9]. A novel solar-powered tube heater was developed to reduce the use of the current induction heater in a powder-based coating process in Romania [10]. Figure 1 presents the design of the tube heater that was designed and optimised numerically to employ multiple air impingement jets to effectively heat the moving tubes without interrupting the coating process [10]. These simulations were carried out using ANSYS FLUENT to evaluate the thermal performance of the tube heater with a steady-state tube [10]. To evaluate the performance of the tube heater with a moving tube, this paper develops two ANSYS models and compares them against each other for their computational efficiency and accuracy. The first model includes FLUENT Dynamic Mesh (FDM) while the second model includes FLUENT-Transient Thermal (FTT). The heat transfer in the tube heater was analysed in the FDM model by simulating a moving steel tube with a steady heat source while in the FTT model by simulating a steady steel tube with a moving heat source. Moreover, the results of the numerical models are used to evaluate the environmental benefits of the presented tube heater.

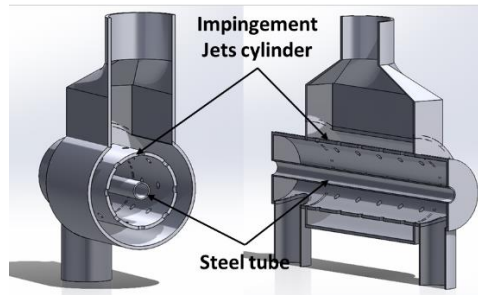


Fig. 1: Concept design of the solar powered tube heater [10]

## 2. Methodology

This section presents the methodology followed to evaluate the performance of the tube heater as follows: Firstly, the development of each of the two ANSYS models; FDM and FTT, is discussed. Moreover, the equation used to calculate the environmental benefits is presented.

### 2.1. ANSYS FLUENT Dynamic Mesh (FDM) Model

The first method followed to simulate the tube heater with a moving tube uses ANSYS FLUENT's dynamic mesh model (FDM). Dynamic mesh is a function that is mainly used to model flow in a domain of moving boundaries and has been extensively used in the automotive, aerospace, chemical and environmental industries [11]. This method was used to simulate the movement of the steel tube through the heater. The modelling process was set up by developing the geometry, meshing the geometry, setting up the ANSYS FLUENT model, and analysing the results. In addition, a User Defined Function (UDF) was developed to describe the motion of the tube in the dynamic mesh.

### Geometry

The geometry of the dynamic mesh model was designed based on the concept design of the tube heater (Figure 1) and presented in Figure 2. It includes two parts; the steel tube and the airflow part. The steel tube (Tube - Figure 2a) is designed with a length of 1000 mm to move in and out of the heater axially. The airflow region includes a pathway for the steel tube (Figure 2 b and c) to ensure that the tube always has full contact with the air to prevent errors in the simulation of the dynamic mesh.

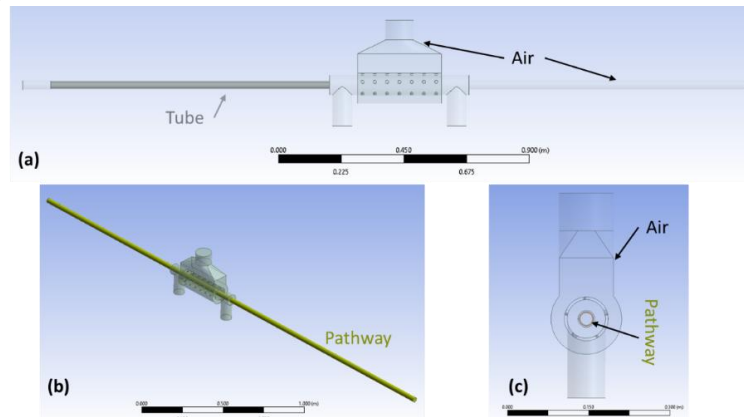


Fig. 2: Geometry of tube heater for dynamic mesh model – XZ view (a) – far view (b) – YZ view (c)

### Meshing

ANSYS Meshing was used to mesh the geometry of the tube heater, as presented in Figure 3. The *method* function with mesh type of *tetrahedral* and meshing algorithm of *patch-independent* was used as it is a prerequisite for the dynamic mesh model. The obtained mesh showed 1,253,699 nodes and 6,867,533 elements with an average skewness

of 0.25, maximum skewness of 0.62 and a minimum orthogonal quality of 0.38 which is considered a good mesh quality [11].

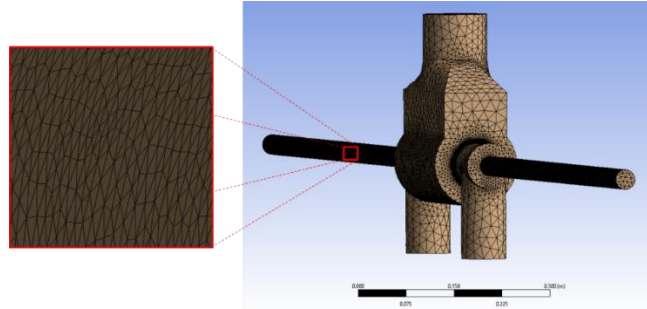


Fig. 3: Meshing of the air flow region and the steel tube for the dynamic mesh model

### Fluent set-up

The next step includes setting up the solution for the system using ANSYS FLUENT. The turbulence model was set to *SST k-w*, the material for each cell zone was air for the *Air* part and Stainless Steel 235j for the *Steel Tube* part with an initial temperature of 30 °C, and the boundary conditions for the air inlet were set similarly to the steady-state model [10] to a temperature of 240 °C and a flowrate of 0.28 kg/s. The motion of the steel tube was set up in three steps: (i) setting up the motion of the solid zones, (ii) developing the UDF to define the motion of the steel tube wall and (iii) setting up the dynamic mesh function for the deformation of the *Air* part and the motion of the steel tube wall, as defined by the UDF. Firstly, the motion of the solid zone was set up by enabling the *Moving Solid* characteristic for the *Steel Tube* part and identifying its speed on the x-axis to be 0.1 m/s. Secondly, the motion of the walls of the steel tubes was set up using a UDF. It used the *Define Grid Motion* function to identify the displacement of the walls on the x-axis in terms of time and the *Set Deformation Thread Flag* function to prevent the generation of low-quality meshes [12]. This UDF was compiled and loaded to the ANSYS FLUENT environment to simulate the motion of the steel tube walls. Thirdly, the dynamic mesh function set-up for the moving walls included *Remeshing* and *Deformation*. On one hand, *Remeshing* allows ANSYS FLUENT to re-generate the mesh every timestep when the steel tube and its walls move. This is carried out using the Laplacian Smoothing Method which adjusts the location of each cell to the centre of the neighbouring mesh by calculating the spring constant which represents the stiffness between the two cells. This method is the most commonly used method and is considered efficient in terms of computational cost and time [11]. The Laplace method set-up for the current simulation was kept as default except for the spring constant which was set to 0.001, the maximum number of iterations to 250 and the node relaxation to 0. On the other hand, *Deformation* allows ANSYS FLUENT to recognise the change in the boundaries of the *Air* part [11]. It was enabled for local cell and local face with the length scale adjusted to meet that of the current mesh and size re-meshing interval of 1.

These settings were used to generate the dynamic mesh zones as follows:

(i) Deformation has been set to the *Air* part since its boundaries will be changing as the tube moves. In this case, the length scale range was also adjusted to meet that of the *Air* part.

(ii) the UDF has been assigned to the walls of the steel tube and their shadows to identify their motion through the system.

### Solution set-up

The solution was set up for a specific number and size of the timesteps which determines the total time of the simulation. Based on the speed of the steel tube of 0.1 m/s and the distance of the tube to enter and exit the heater which is 1.5 m, the total time required for the simulation was found to be 15 s. To determine the timestep size, the dynamic mesh motion preview setting was used to test the largest possible size to avoid errors during the simulation. Table 1 presents four different tests that were carried out to identify the timestep size. It can be seen that Test D with 0.0025 seconds showed no error therefore was used to set up the solution of the simulation using 40 processors for 600 timesteps with 10 iterations per timestep.

Table 1: Identifying the timestep size for the dynamic mesh model

	Timestep Size [s]	Number of Timesteps	Status
Test A	0.05	300	Error
Test B	0.01	1500	Error
Test C	0.005	3000	Error
Test D	0.0025	6000	No Error

## 2.2. Method II - ANSYS FLUENT-Transient Thermal Model

ANSYS FLUENT- Transient Thermal (FTT) model was used to simulate the tube heater with a moving tube. ANSYS Transient Thermal is part of ANSYS Mechanical which includes a group of software that deals with finite element analysis (FEA) and can be used to analyse heat transfer to an object in terms of time [11]. This software was used to import the heat transfer characteristics of the airflow in the tube heater as simulated by ANSYS FLUENT. The ANSYS Transient Thermal allows importing the same heat source more than once and applying them at different timesteps and locations to the same target, resembling a moving heat source. This approach was used to analyse the heat transfer of a moving tube in the heater. It is required to hypothetically divide the target (steel tube) into smaller subsections and apply the different heat sources imported from different FLUENT simulations to them subsequently which will represent the motion of the tube. To implement this method three main factors were identified: (i) the length of the hypothetical subsections of the tube, (ii) the number of required FLUENT simulations and (iii) the timestep at which each FLUENT simulation will be applied to the steel tube in ANSYS Transient Thermal.

(i) **The length of the hypothetical subsections of the tube,  $L_{HS}$ ,** in this method, is equivalent to the timestep size in the Dynamic mesh method (Section 2.1). Similar to the timestep size, the size of the subsection was determined to be the smallest subsection of the temperature contour with uniform temperature around the axis of motion. As can be seen in Figure 4 which presents the temperature contour of the steel tube from a steady state test, the smallest subsection with a uniform temperature gradient around the axis of motion (axial axis) is the area between the stagnation points of adjacent jets. The length of this area is approximately 20 mm which will be considered as the size of the 50 hypothetical subsections of the 1000 mm tube. The length of the subsections can also hypothetically subdivide the imported heat load of the 500 mm wide tube heater into 25 subsections which helped to calculate the required number of the ANSYS FLUENT simulations.

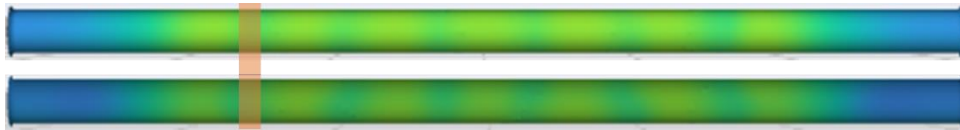


Fig. 4: Temperature gradient of the steel tube in the powder-based tube heater

(ii) **The number of ANSYS FLUENT simulations,  $N_{FS}$ ,** required for this method is related to the number of hypothetical subsections of the steel tube and imported heat loads. To obtain accurate results, heat was applied from all subsections of the tube heater to every single subsection of the tube. This will require 74 Fluent simulations as calculated using the following equation:

$$N_{FS} = \text{No. of target subsections} + \text{No. of heat source subsections} - 1 \quad (1)$$

(iii) **The timestep size** of each heat load of the 74 Fluent simulations applied to the steel tube is based on the speed at which the tube should be moving, the subsection size and the number of subsections in the steel tube. Considering the tube heater velocity,  $V_{ST}$ , of 0.1 m/s and a subsection size of 0.02 m, it was obtained that each FLUENT simulation was applied subsequently for 0.2 s. Considering that there are 74 fluent simulations, the total time of the simulation will count to 14.8 s. This does not match the required time of 15 s for the tube to travel the required distance of 1.5 m (Section 2.1). This is due to the initial simulation setup where the tube is placed 20 mm inside the tube heater. To overcome this difference and mimic the real-life flow of the tube in the tube heater, a 75th timestep with no heat load is added to the

simulation. Thus, the start time of the timesteps,  $t_{TS}$ , for each imported ANSYS FLUENT simulation,  $N_{FS}$ , was calculated using the following equation:

$$t_{TS}(N_{FS}) = \frac{V_T}{L_{HS}}(N_{FS}) \quad (2)$$

These factors are considered for the modelling of the tube heater that is carried out in three folds: (i) developing the geometry, (ii) carrying out the ANSYS Fluent simulations and (iii) carrying out the ANSYS Transient Thermal simulation.

### Geometry

The geometry for the ANSYS FLUENT – Transient Thermal model included the geometries for the 74 Fluent simulations and the steel tube to be analysed in Transient Thermal. These geometries were designed in a single geometry file (Figure 5) and then transported to the FLUENT and Transient Thermal simulations to ensure that all simulations carried have a common coordinate system. Hence, a single geometry file was connected to all the FLUENT and Transient Thermal simulations where ANSYS Meshing was used to suppress the irrelevant parts.

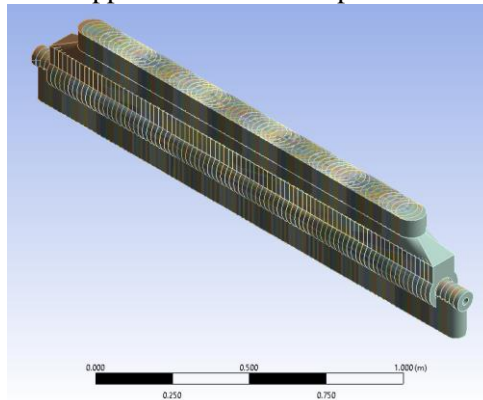


Fig. 5: Geometry for the ANSYS FLUENT Transient Thermal with 74 Air parts and 1 steel tube part

### FLUENT

The 74 FLUENT simulations were simulated similarly. Firstly, the respective *Air* part for each simulation was meshed in ANSYS Meshing. This produced a mesh of 459,106 elements and 89,207 nodes with average skewness of 0.23, maximum skewness of 0.68 and minimum orthogonal quality of 0.32 which is considered a good mesh quality [11]. Secondly, a steady-state fluent simulation was carried out with the boundary conditions identified as presented in Figure 6. Later, the wall that contacts the steel tube (Green) was identified as a moving wall at a speed of 0.1 m/s to mimic the real-life conditions of the process. Furthermore, the airflow inlet was set to 0.28 kg/s at a temperature of 240 °C. Moreover, the simulations were finally initialized and run for 1000 iterations. Finally, the heat transfer characteristics at the moving wall (Green – Figure 6) were obtained to be exported to the ANSYS Transient Thermal environment.

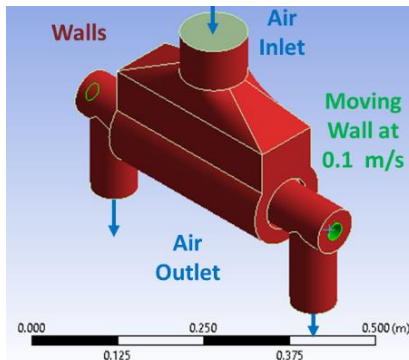


Fig. 6: FLUENT airflow simulation for the FLUENT – Transient Thermal method

### Transient Thermal

The FLUENT simulations results are imported to ANSYS Transient Thermal where the solution for the heat transfer to the steel tube is setup as follows: Firstly, the steel tube was meshed with hexahedron mesh type. Secondly, the initial condition of the tube was set to a temperature of 30 °C. Then the timestep for each imported FLUENT simulation was identified following Equation 2. Finally, the temperature of the tube is set up to be monitored throughout the simulation.

### 2.3 Environmental benefits of the solar-powered tube heater

The environmental benefits of the tube heater were calculated based on its heat rate to the tube which is directly proportional to the reduction in the use of the induction heater leading to lower GHG emissions. Without the solar-powered tube heater, the induction heater is expected to heat the steel tube from 30 to 240 °C, whereas, with the tube heater, the induction heater will have to the heater from the final average temperature of the tube as obtained from the two ANSYS models,  $T_{ST-f}$  to 240 °C. Hence, Equation 3 is used to calculate the reduction in GHG emissions per 1 meter of steel tube heated,  $E_{savings}$  based on  $T_{ST-f}$ , the mass of a 1-meter tube,  $m_{tube}$ , heat capacity of steel,  $c_{p-steel}=0.49$  kJ/kg.K [13], the efficiency of the induction heater,  $\eta_{IH}$ , and the CO<sub>2</sub> emission rate for electricity in Romania,  $E_{rate}=252$  gCO<sub>2</sub>/kWh [14].

$$E_{savings} \left[ \frac{gCO_2e}{1 - meter\ tube} \right] = m_{tube} \times c_{p-tube} \times \frac{T_{ST-f} - 30}{\eta_{IH}} \times E_{rate} \quad (3)$$

## 3. Results

### 3.1 Comparison of the two ANSYS models

This section evaluates and compares the performance of the tube heater with a moving tube using two models; FDM and FTT. The average, minimum and maximum temperatures of the tube as obtained by the two methods are presented in Figure 7. The average tube temperature obtained by both methods was very close in value throughout the process, reaching 76 and 76.8 °C, respectively. Contrarily, the minimum temperature showed a slight difference from t=13 to t=15s and the maximum temperature showed a higher temperature increase at the beginning and end of the process in FDM model due to the interaction of the air crossflow with the edges of the tube (Figure 8). This phenomenon was not observed in the FTT model. Hence it can be deduced that the FDM model reflects the thermal behaviour of the tube heater more accurately compared to the FTT model. Nevertheless, considering the real-life process at ArcelorMittal where tubes enter subsequently after each other, this leaves no space for the air crossflow to hit the edges which reduces the difference in the results. Since the phenomenon specifically observed in the FDM model did not have any impact on the final results, it is hence more justifiable to use the FTT model to understand the thermal behaviour of the heater and analyse its benefits due to its lower computational time and cost.

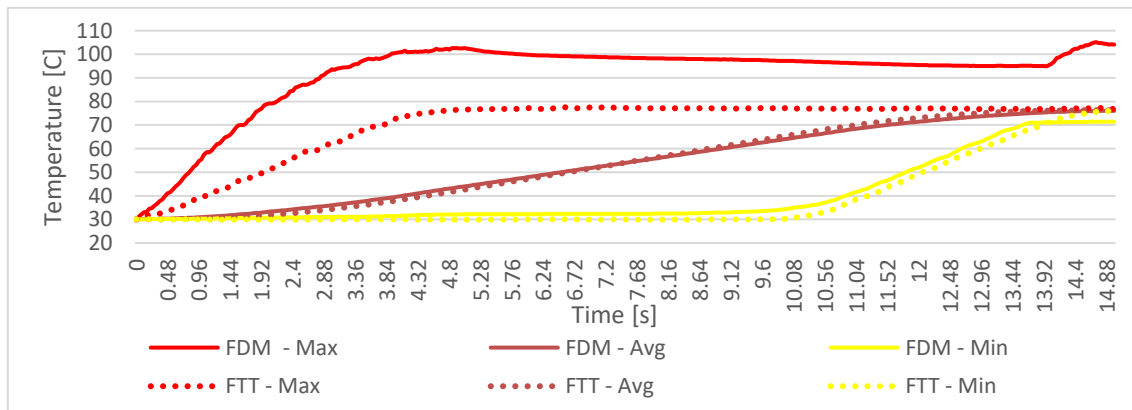


Fig. 7: Tube temperature as obtained from the FDM model and the FTT model



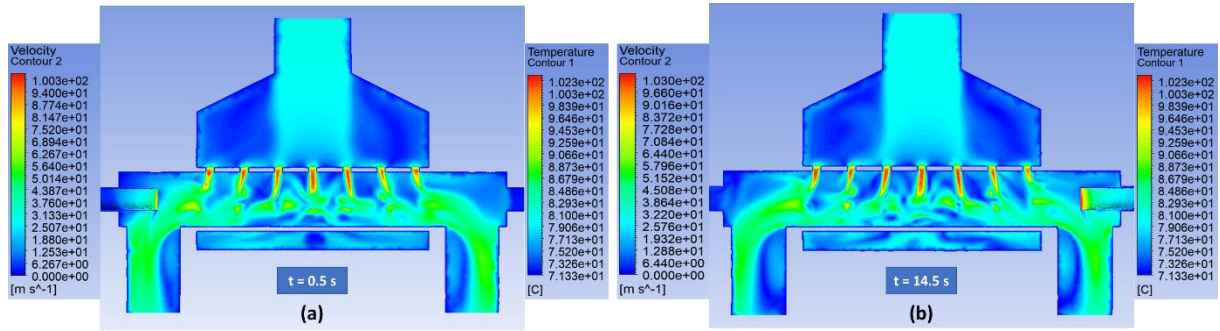


Fig. 8: Air velocity contour and steel tube temperature contour as obtained in FLUENT at (a)  $t = 0.5s$  and (b)  $t = 14.5s$

Table 2 summarises the main differences between the two methods in terms of computational cost and time, thermal performance analysis accuracy, real-life application and results accuracy compared to experimental results. It can be seen that the FDM model required 40 processors, 12 hours to set up and 31 days to run whereas the FTT model required only 4 processors, 4 days to set up and 2 days to run. This shows that the FTT model is computationally more time and cost-efficient compared to FDM model. Nevertheless, the FDM model simulated all aspects of the thermal performance of the system and obtained more reasonable results. In addition, when considering the real-life process at ArcelorMittal of producing tubes in batches, FDM model is more suitable to analyse the heat transfer to all the tubes of the batch, whereas the FTT model does not allow the simulation of the first and last tubes. In conclusion, both simulations produced reasonable results where the FDM model is preferable to produce high accuracy results whereas the FTT model produces sufficiently adequate results in a shorter time period and lower computational cost.

Table 2: Summary of differences between the two models

Model	ANSYS FLUENT Dynamic Mesh	ANSYS FLUENT - Transient Thermal
Computational Cost [No. of Processors]	Required 40 processors to run the simulation	Required 4 processors to run the simulation
Computational Time [days]	Required 12 hours to set up + 31 days to run	Required 4 days to set up + 2 hours to run
Thermal performance analysis accuracy	simulates all aspects of the performance with high accuracy	simulates the heat transfer to the outer wall of the tube
Real-life application	Suitable to analyse the heat transfer to a whole batch	Suitable to analyse the heat transfer to most tubes except for the first and the last of a batch

### 3.2 Environmental benefits of the tube heater

The two ANSYS Models showed the steel tube to reach a final temperature,  $T_{ST-f}$ , of approximately 76 °C (Figure 7). This shows that the system can cover 22% of the required heat load of the tubes to reach 240 °C. Based on Equation 3, it is calculated that the use of the tube heater will lead to emissions savings of 2.15 gCO<sub>2</sub>e per each 1-meter tube processed in the powder-based coating process. This will bring the steel tube industry closer to achieving their 2050 net-zero emissions target.

### 4. Conclusion

Two ANSYS models were developed, FDM and FTT, to evaluate the thermal performance of a novel solar-power tube heater to reduce the GHG emissions in the powder-based coating process. Both models showed similar results in terms of the temperature rise of the steel tube to 76 °C. This resulted in a reduction of GHG emissions by of 2.15 gCO<sub>2</sub>e per each 1-meter tube processed. Although the FDM model showed to produce a more detailed temperature contour of the heated tube, the FTT still showed sufficiently accurate results with lower computational time and cost. Hence, the FTT model is advised

to be used for numerical simulations that investigate heat transfer to a moving solid. Future work will include developing an experimental test rig to validate the numerical models.

## Acknowledgements

This project is funded from the EU Horizon 2020 research and innovation programme, Application of Solar Energy in Industrial processes (ASTEP), under grant agreement No 884411.

## References

- [1] L. Kumar, M. Hasanuzzaman, and N. A. Rahim, 'Global advancement of solar thermal energy technologies for industrial process heat and its future prospects: A review', *Energy Conversion and Management*, vol. 195. Elsevier Ltd, pp. 885–908, Sep. 01, 2019. doi: 10.1016/j.enconman.2019.05.081.
- [2] European Commission, 'Climate Strategies and Targets', 2007.
- [3] H. Tannous, V. Stojceska, and S. Tassou, 'The Use of Solar Thermal Heating in SPIRE and Non-SPIRE Industrial Processes', *Sustainability*, vol. 15, no. 10, p. 7807, May 2023, doi: 10.3390/su15107807.
- [4] B. Muster-Slawitsch, S. Meitz, and E. Frank, 'Solar Heat Integrations in Industrial Processes', 2020. [Online]. Available: <http://ship-plants.info/>
- [5] AEE INTEC, 'Solar Thermal Plants Database', 2020. <http://ship-plants.info/> (accessed Aug. 07, 2020).
- [6] European Commission, 'Energy Balance Sheet – 2015 edition', 2017. Accessed: Dec. 26, 2021. [Online]. Available: <https://ec.europa.eu/eurostat/web/products-statistical-books/-/KS-EN-17-001>
- [7] R. Abbas, M. J. Montes, J. Cano, L. F. González-Portill, A. Sebastián; J. Muñoz-Antón, A. Rovira, and J. M. Martínez-Val, 'Enhancement of SunDial optical performance handling cosine and end losses', in *AIP Conference Proceedings*, American Institute of Physics Inc., May 2022. doi: 10.1063/5.0085667.
- [8] S. Suman, M. K. Khan, and M. Pathak, 'Performance enhancement of solar collectors - A review', *Renewable and Sustainable Energy Reviews*, vol. 49. Elsevier Ltd, pp. 192–210, Sep. 01, 2015. doi: 10.1016/j.rser.2015.04.087.
- [9] H. M. Tahar, M. T. Hannachi, B. Dahache, and E. Guelloudje, 'Effect of Heating of Steel Tubes by Induction at High Frequency', *Microwave Review*, vol. 24, no. 1, pp. 23–27, 2018, [Online]. Available: <https://www.researchgate.net/publication/333668727>
- [10] Hadi Tannous, Valentina Stojceska, and Savvas Tassou, 'The Use of Multiple Impingement Jets in a Solar-Powered Tube Heater for Decarbonization of the Steel Industry', in *Posters*, Boston, MA, USA, 2023.
- [11] ANSYS Inc., 'ANSYS FLUENT 12.0 User's Guide', 2009.
- [12] ANSYS Inc., 'ANSYS FLUENT 12.0 UDF Manual', 2009.
- [13] The Engineering Toolbox, 'Metals - Specific Heats', 2023.
- [14] Ember, 'Carbon intensity of the power sector in Romania from 2000 to 2021', 2023.