Peer-reviewed version available at *Energies* **2016**, *9*, 833; doi:10.3390/en9100833

Article

Thermal Analysis of an Industrial Furnace

Mirko Filipponi [1,*], Federico Rossi [1], Andrea Presciutti [1], Stefania De Ciantis [1], Beatrice Castellani [1] and Ambro Carpinelli [2]

1 CIRIAF - UniversitàdegliStudi di Perugia, Via G. Duranti 67, 06125 - Perugia, Italy
2 DivisioneFucine di AcciaiSpeciali Terni, V.leB.Brin 218, 05100 - Terni, Italy
* Correspondence: mirko.filipponi@unipg.it

Abstract: Industries, which are mainly responsible for high energy consumptions, need to invest in research projects in order to develop new managing systems for rational energy use and to tackle the devastating effects of climate change caused by human behavior. The study reported in this paper concerns the forging industry, where the production processes generally start with the heating of the steel in furnaces and continue with other processes, such as heat treatments and mechanical machining. One of the most critical operations, in terms of energy loss, is the opening of the furnace doors for the insertion and extraction operations. During this time, the temperature of the furnaces decreases by hundreds of degrees in a few minutes. Because the dispersed heat needs to be supplied again through the combustion of fuel, increasing the consumption of energy and the pollutant emissions, the evaluation of the amount of the lost energy is crucial for the development of operating or mechanical systems able to contain this dispersion. To perform this study, CFD simulation software was used. Results show that at the door opening, because of temperature and pressure differences between the furnace and the ambient, turbulences are generated. Results also show that the amount of energy lost for an opening of 10 minutes for radiation, convection and conduction is equal to 5606 MJ where convection is the main contributor with 5020 MJ. The model created, after being validated, has been applied to perform other simulations in order to improve the energy performance of the furnace. Results show that a reduction of the opening time of the door allows energy savings and limits pollutant emissions.

Keywords: CFD simulation; industrial furnace; heat flux; forging industry; thermal analysis

---

1. Introduction

Global carbon dioxide emissions reduction is becoming an explicit target of governmental policies, linked to the decrease in fossil fuel use [1]. Global greenhouse gases are produced by several economic activities, mainly electricity and heat production and industry [2], as well as agriculture and breeding [3], transportation [4], buildings [5] and fuel extraction, processing and transportation [6].

Greenhouse gas emissions from industry primarily involve fossil fuels burned on-site at facilities for energy and the iron and steel industry is among the top five most energy-intensive industry sectors as reported in the Energy Technology Perspectives 2014 [7]. The main challenge for this sector is lowering energy consumption, greenhouse gas emissions and pollutant emissions [8].

Iron and steel industries need to invest resources to limit their environmental footprint. The most used methods to determine energy consumption and efficiency of the involved production processes are numerical thermal simulations and LCA analysis [9-10].

Most of the works reported in the literature refer to walking beam furnaces and have the objective to predict the temperature distribution in the slabs, which plays a major role in the quality of the final product, and the heating efficiency of the furnace [11-18]. Most of these works have relied on mathematical models, since experiments are quite difficult to perform due to the large size of real furnaces, limited physical access and harsh environmental conditions in the furnace [19].

Instead, this paper focuses on the thermal analysis of a forging furnace and discusses the results of joint research between the University of Perugia and Divisione Fucine di AcciaiSpeciali Terni (SdF), a worldwide leader in the forging industry of the steel.

In SdF the forged products are obtained through a production process which involves furnaces and presses. Generally several cycles of heating and machining at the press are needed to reach the right geometry. However these processes generate internal stresses and defects that compromise the products' quality. For this reason, the forged steel is subjected to other thermal and finishing treatments to release stresses, modify the molecular structure, improve quality and performance of the materials.

From several analyses made in SdF, it emerged that the massive use of the furnaces along all the production processes constitutes one of the main factors responsible for the high consumption of energy. An improvement of their energy efficiency allows the company to reduce the pollutant emissions, save energy and decrease the cost of the final product.

All the furnaces in SdF are fueled with natural gas but differ from each other by dimension, number of burners, refractory properties and so on. It is, however, possible to classify them into two main categories. The heat treatment furnaces that are generally turned off when the processing is finished, and the forging furnaces which are always turned on at maximum temperature, and are used for the heating of the ingots. The continuous operation of the forging furnaces is the main contributory reason for the high energy consumption, since a continuous energy supply is required to compensate the energy loss during the furnace opening and maintain the operating temperature conditions.

A typical insertion/extraction operation starts with the opening of the furnace door, continue with the movement of an extractable bogie hearth, the positioning or the picking up of the products by means of an overhead travelling crane, and then finish with the insertion of the bogie hearth and the door closing. The time necessary to perform these operations depends on many factors, such as the dimensions of the ingots, the maneuver time, the presence of faults and so on. From the analysis of the data disposed by SdF, it emerged that the time varies between about 10 minutes up to 1 hour. In this time interval the furnace temperature decreases by hundreds of degrees, from 300°C for brief openings up to 700°C for longer openings.

To the best of our knowledge, in literature there are no other studies on the assessment of the thermal behavior of forging furnaces during transient insertion and extraction periods, even though the evaluation of energy losses and other parameters, such as the velocity and the directions of the gases, represents a valid decisional instrument to reduce energy consumption. The complexity of the phenomena that occur in real conditions makes difficult the analytical study where conduction, convection and radiation occur concurrently [20]. For this reason, the present paper discusses the results of a three dimensional transient simulations, done with a CFD Software, of a forging furnace, considering a total opening time of 10 minutes. The first simulation was done with a lifting time of the door of 157 s, which represents the real operating condition of the furnace in SdF. The model created was then validated through an experimental test done in SdF with the same operating condition applied to the model. Successively, in order to improve the energy efficiency of the furnace, two other simulations with different opening speed of the door were done and in this paper discussed. In particular, an opening time of 40 s and 60 s were considered. The results obtained were also used to determine the carbon footprint of the process.

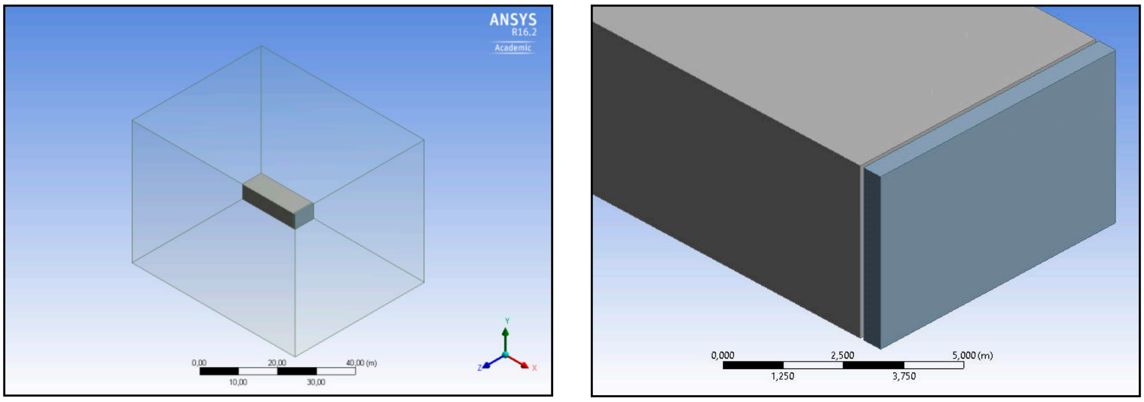
2. Model settings

The model creation started with the drawing of the geometry and its discretization through a meshing process. Continued with the definition of the simulation setup, and ended with the analyses of the results. [21].

2.1. Geometry creation

Peer-reviewed version available at *Energies* **2016**, *9*, 833; doi:10.3390/en9100833

The geometry of the case of study is composed only by the fluid domains involved in the heat fluxes and by the furnace door. Since the furnace is at the operating condition when the door is lifted up, and all the burners are turned off, the model can be created with a very clean geometry without drawing components that do not influence the motion of the fluid and the heat fluxes. In Figure 1 two particulars of the geometry created are shown, specifically the position of the furnace geometry is considered from one corner of the entire domain to facilitate the complete view of the model.



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

Figure 1. Geometry used for the simulation: (**a**) The entire domain considered for the analysis, consisting of a fluid domain of the furnace enclosed in another fluid domain which represents the ambient air; (**b**) Particular of the geometry where the fluid domain of the furnace is showed in grey, while the door is showed in blue.

The furnace is 4.7 m height x 6.9 m width x 18.55 m depth. These dimensions are relative to the internal space of the furnace, which is the space occupied by the fluid. The door has been drawn at a distance of 10 cm from the furnace border since at the moment of the opening, it is slid in a horizontal direction before being lifted up. This movement allows the reduction of the frictions between the door and the furnace borders. The width and the height of the door are the same as the furnace, while the depth is of 50 cm. Finally the enclosure, which represents the ambient air outside the furnace, has been drawn with a distance of 20 m from the upper surface of the furnace, 1 m from the lower and back surfaces, and 5 m from the lateral surfaces. In the following Table 1 are reported the dimensions of the model components.

Table 1. Model components dimensions.

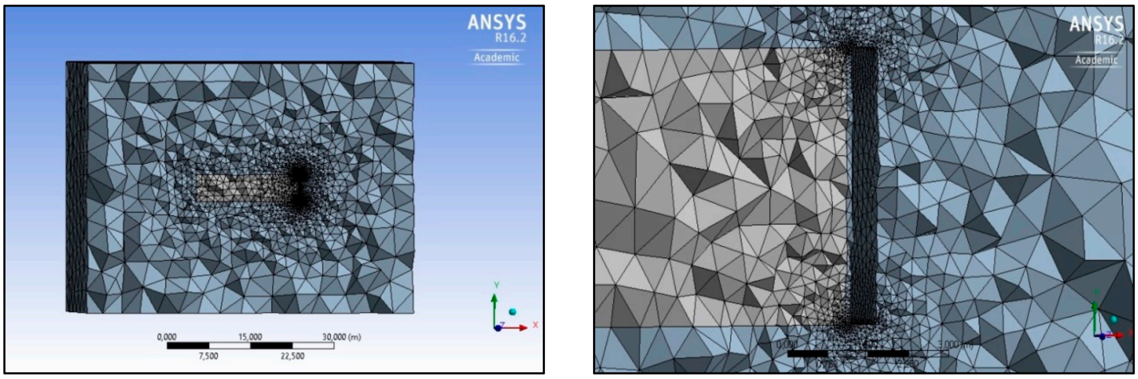
	Height [m]	**Width [m]**	**Dept [m]**
Furnace	6.9	4.7	18.55
Door	6.9	4.7	0.5
Enclosure	25.7	16.9	30.06

The representation of the furnace contained in an enclosure, allows us to characterize the motion of the fluids between the furnace and the environment.

2.2. Meshing process

To guarantee the continuity between the two domains: furnace and the environment, a conformal mesh has been used (Figure 2 (a)). Because the door does not influence significantly the heat fluxes, it has not been subject to the meshing process, as it possible to see in the Figure 2 (b). In addition, being a transient simulation with a dynamic mesh, the entire domain has been meshed with a tetrahedral mesh that supports the operation of remeshing at every time step.

Peer-reviewed version available at Energies 2016, 9, 833; doi:10.3390/en9100833



(a) (b)

Figure 2. Drawings of a meshed domain: (**a**) Representation of the entire domain meshed with a conformal method; (**b**) Particular of the meshed domain near the door which is not included in the meshing process.

2.3. Models for convection and radiation

To model the turbulent flows which occur during the door opening, the Reynolds-Averaged Navier - Stokes (RANS) approach was used. Recent studies demonstrate that it represents the most commonly accepted model for turbulence modeling, especially for industrial applications [19]. This model is derived from the formulation of the standard k-ε model proposed by Launder and Spalding which is based on two transport equations which includes the two variables of the turbulent kinetic energy (k), and the turbulent dissipation rate (ε) [22]. With the standard k-ε model it is possible to predict the behavior of a flow in presence of turbulences with high accuracy. However, the realizable k-ε model, developed after the standard k-ε model, proposes a different formulation giving more accurate results, than the first one, for turbulent flows [23]. With the use of "enhanced wall treatments" option, simplified formulas for turbulence quantities in the cells near the wall are used.

For the modeling of the radiative heat flux a preliminary analysis about the concentration of the flue gases inside the chamber of the furnace at the door opening has been done. Results showed that the burners work with an excess of air avoiding soot formation. For this reason, the assumption of an optical thickness equal to zero was done and the model Surface to Surface has been used [24]. In this model, the energy exchange between surfaces depends on their size and position with respect to the others. The model considers these geometric factors through the computation of the view factors [25].

The S2S model also assumes that the surfaces are gray and diffuse, and the emissivity and the absorptivity do not depend on the wavelength. It is well known that for the gray bodies the total energy incident (E) on a surface is partly reflected (ϱE), partly absorbed (αE) and partly transmitted (τE). Since this model is often used with opaque surfaces to thermal radiation in the infrared spectrum, the fraction of energy transmitted can be not considered. With these assumptions, from the Kirchoff's law [26] results that $\alpha+\varrho+\varepsilon=1$, but $\alpha=\varepsilon$, therefore $\varrho=1-\varepsilon$ [25].

2.4. Boundary conditions

The boundary condition at wall boundaries was given using Fluent's "enhanced wall treatment" which allows us to predict the flows in the near-wall region where laminar and turbulent flows are blended [27]. The boundary and cell conditions given for the initial time were provided by SdF, in particular the temperature of the furnace is at the operating condition of 1500 K , while the ambient temperature is at 300 K. The pressure is set at the 101325 Pa for the ambient and an over pressure of 40 Pa respect to the ambient, for the furnace.

 doi:10.20944/preprints201608.0029.v1

Peer-reviewed version available at *Energies* **2016**, *9*, 833; doi:10.3390/en9100833

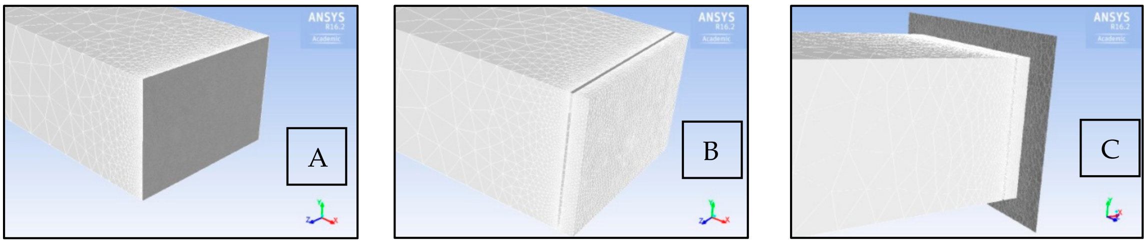
2.5. Dynamic mesh

The use of a dynamic mesh allowed us to do a transient simulation with the movement of the door. Through the use of the first-order backward difference formula it was possible to calculate the conservative energy equation for each time step, considering two steps concurrently [28].

For the motion of the door, a profile has been defined and applied. In particular, the door has been set as a "rigid body" using the profile for the motion, while the adjacent cells have been set as "deforming" with the application of a remeshing and smoothing methods.

2.6. Control surfaces

In order to monitor and evaluate the fluid motion and the heat fluxes that occur during the opening of the door, three control surfaces were created. As shown in Figure 3(a) the first surface A, having dimensions equal to 6.9m x 4.7m, has been positioned in correspondence of the exit of the gases from the furnace. This surface allows us to quantify the amount of the total convective flow that leaves the furnace during all the transient time considered. As it is intuitive to expect that the hot gases leaving the furnace tend to move upwards passing through the space between the door and the edge of the furnace formed because the scrolling of the door, a new surface B was created. This surface with dimensions of 6.9m x 0.1m is shown in Figure 3(b). To capture the flows deflected in the other directions the last control surface C was positioned behind the door with a larger dimension than the others of 6.3m x 9.2m, (Figure 3(c)). The orientation of the geometry in the figures has been chosen to show a clear view of the surfaces created.



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

Figure 3. The three control surfaces created for the evaluation of the convective flow in different zones: (**a**) Control surface A positioned in correspondence of the exit of the gases from the furnace.; (**b**) Control surface B created in the upper section between the door and the furnace edge; (**c**) Control surface C created outside the door.

3. Results

The period of 10 minutes from the opening of the door with a time step of 0.1 s was done. In particular, the first simulation, related to the real operating condition of the furnace in the plant with a complete opening in 157 s, was done and validated through an experimental test done in the real furnace in SdF.

3.1. Fluids behavior

The first parameter monitored with the simulation was the maximum temperature reached on the two control surfaces B and C during the opening. This trend is shown in Figure 4 where it is possible to see that the highest temperature is registered on surface B which reaches values of about 1200 K after a few seconds from opening and then decreases, reaching values of 800 K. Instead, the maximum temperature registered on surface C starts from the ambient value up to values around 700K.

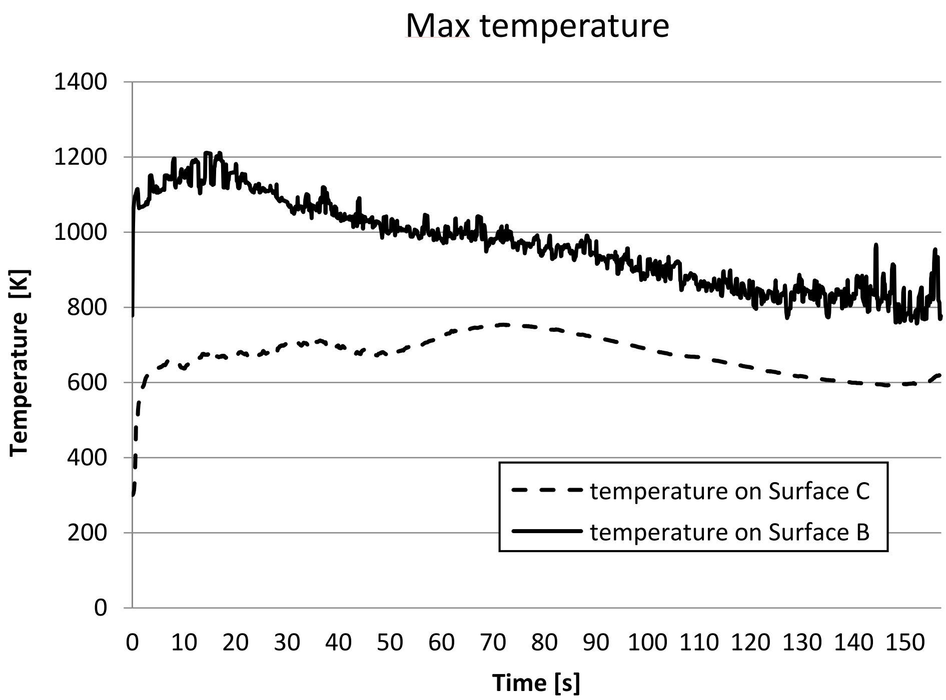
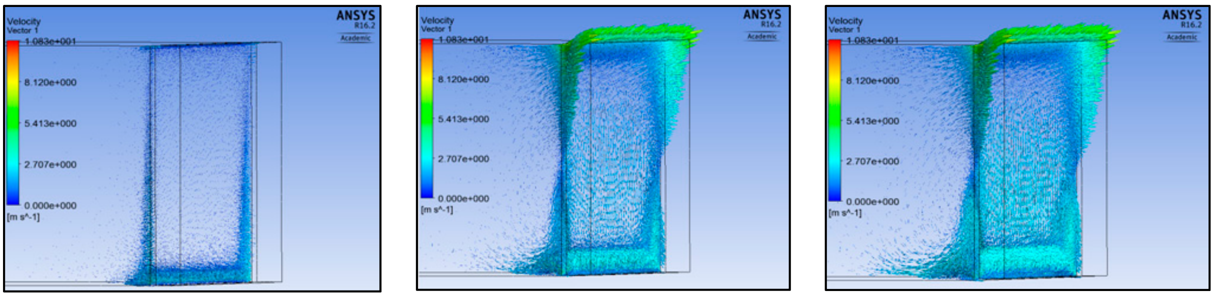


Figure 4. The trend of the maximum temperature reached in the surface B and C during the 157 s of the door opening.

The values of the max temperature registered in the two surfaces depend on the motion of the fluid leaving the furnace. In particular, in Figure 5 a sequence of velocity vectors from the initial time up to 1s is shown. This sequence of frames demonstrates that the hot gases inside the chamber at high temperature move upwards and leave the furnace through the upper section with a velocity that reaches 12 m/s, meanwhile the external air, at ambient temperature, moves towards the furnace through the bottom section.



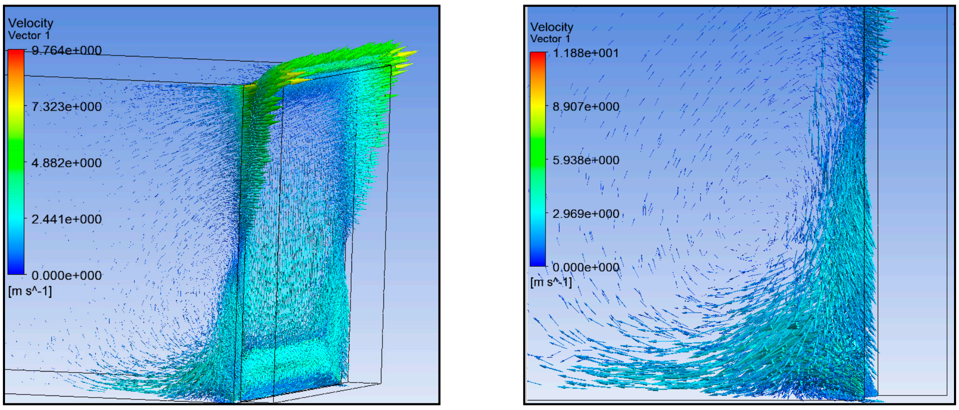
(**a**) (**b**) (**c**)

Figure 5. Sequence of the velocity vectors at different time at the exit of the furnace: (**a**) Velocity vectors at 0 s; (**b**) Velocity vectors at 0,5s; (**c**) Velocity vectors at 1 s.

Continuing to analyze the fluid dynamics of the model, in Figure 6 can be observed that the hot gases which cannot get out from the upper section, are recalled downwards forming a convective motion when they meet the cold air coming from the outside.

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 3 August 2016 doi:10.20944/preprints201608.0029.v1

Peer-reviewed version available at Energies 2016, 9, 833; doi:10.3390/en9100833



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

Figure 6. Velocity vectors of the gases for an opening of 3 cm: (**a**) Velocity vectors and formation of convective motion; (**b**) Particular of convective motion at the bottom of the furnace.

As the door continues to open, the vortexes disappear, the velocity vectors decrease, and the hot gases leaving the furnace start to pass under the door (Figure 7). The complete motion of the convective fluxes for the transient time of 157 s, relative to the furnace domain, is shown in the attached video.

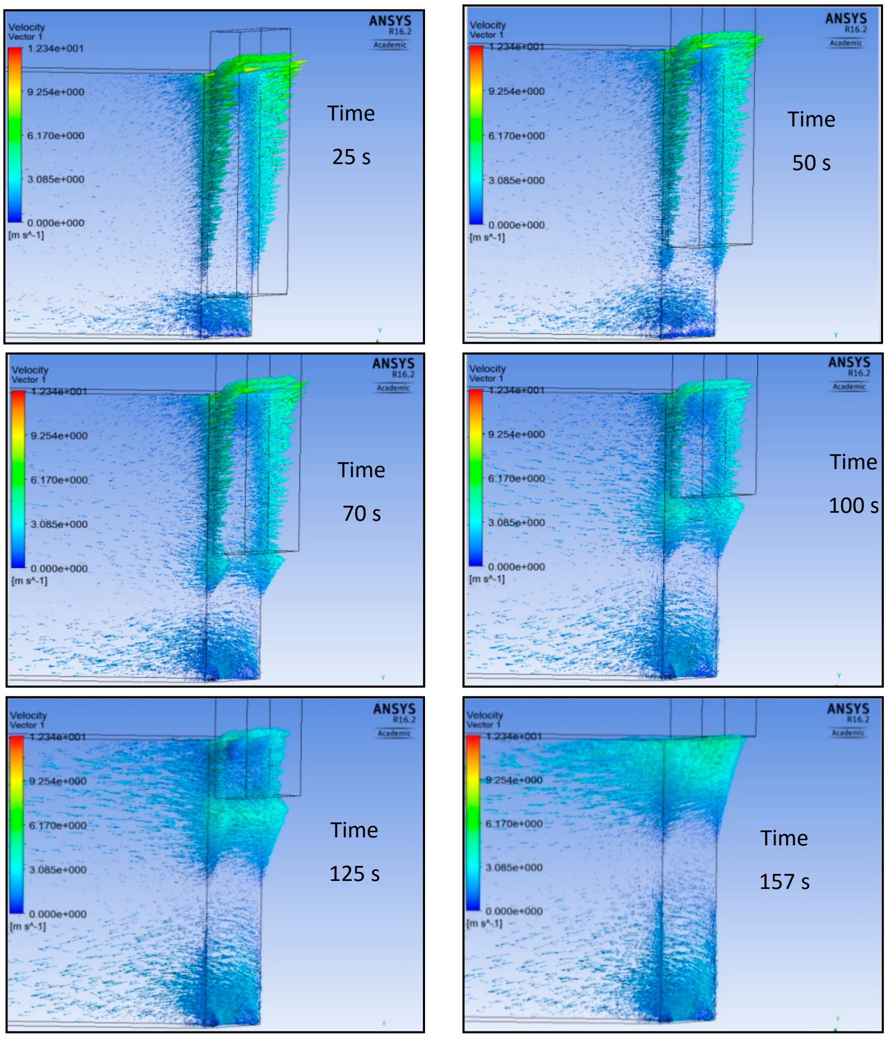


Figure 7. Fluid motion during the transient time from the opening of the door, up to the complete opening of 4.7m at 157 s.

It is important to underline that the vectors reported in the previous images are referred to the gases that move in the furnace domain. For example, observing the picture related to 50 s, it might seem that the vectors pass through the door. In reality this does not occur, in so far as when the gases pass to the air domain, the presence of the door deflects the motion of the vectors in other directions, as it possible to see in Figure 8.

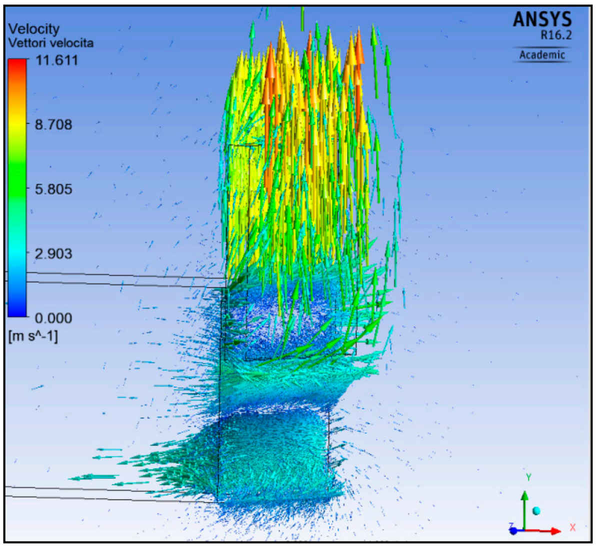


Figure 8. Velocity vectors directions in the air domain that move in vertical direction upwards between the door and the furnace and immediately outside the door.

3.2. Heat flux evaluation

For the evaluation of the energy fluxes, the contributions were identified of the three forms of heat exchange: convection, radiation and conduction. With the aid of the CFD Software, the amount of the heat flux during the opening for convection and radiation has been evaluated, while a spreadsheet for the conduction has been used.

3.1.1. Radiative heat flux

The values of the radiative energy, which passes through the opening of the door evaluated for the first 157 s and related to the complete opening of the door of 4.7 m, are reported in Figure 9.

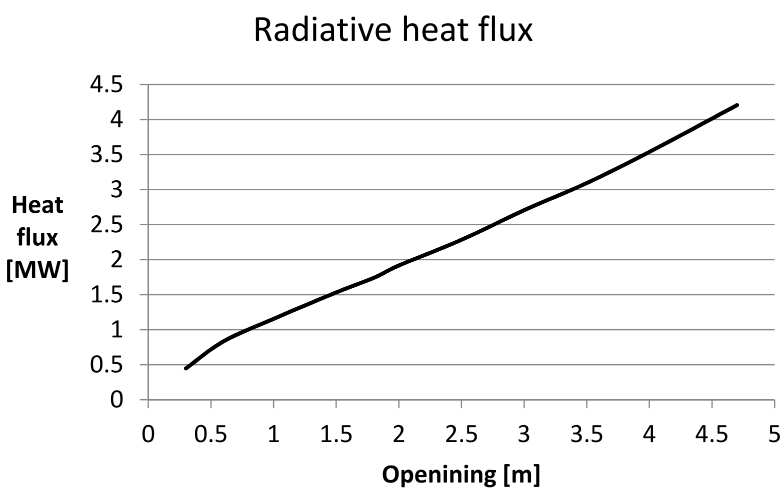


Figure 9. The graph reports the trend of the radiative heat flux through the door during the 157 s.

The values obtained show that the flux grows linearly with time and so with the opening. Even if during the opening the walls start to cool down, the radiative heat flux per unit of area remains about the same because the difference of temperature changes slightly. These values have been verified through the analytical calculation for several time steps using the view factors given by fluent.

3.1.2. Convective heat flux

The evaluation of the convective heat flux is carried out through the calculation of the mass flow rate of enthalpy as the following Equation 1:

$$Q = \int H\rho\vec{v} \cdot d\vec{A} \tag{1}$$

where H is the Enthalpy, and $\rho\vec{v} \cdot d\vec{A}$ is the mass flow.

The knowledge of the values calculated for every time step in the simulation allows us to obtain the total convective heat flux. The trend of the fluxes for the complete opening of the door of 4,7 m in 157 s is shown in Figure 10. It is possible to see that the heat flux on the door is given from the sum of the others. Comparing the two types of heat flux for the first 157 s, it is possible to see that the convective heat flux is higher than the radiative, even if the last one depends on the fourth power of the temperature. This behavior is due to the fact that the turbulence generate from the opening create a motion of air with a high Reynolds number.

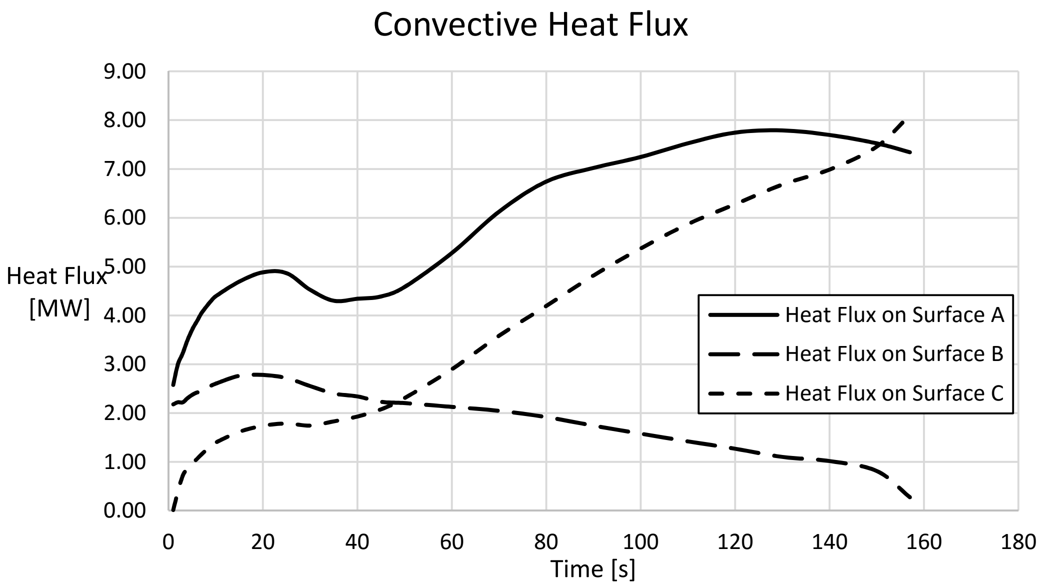


Figure 10. The convective heat flux evaluated in the three control surfaces created for a complete opening of the door of 4.7 m.

3.1.3. Conductive heat flux

The amount of the conductive heat flux through the walls of the furnace was calculated from the knowledge of the stratification of the walls, shown in Figure 11, and from the trend of the temperature obtained from the simulation.

Peer-reviewed version available at Energies 2016, 9, 833; doi:10.3390/en9100833

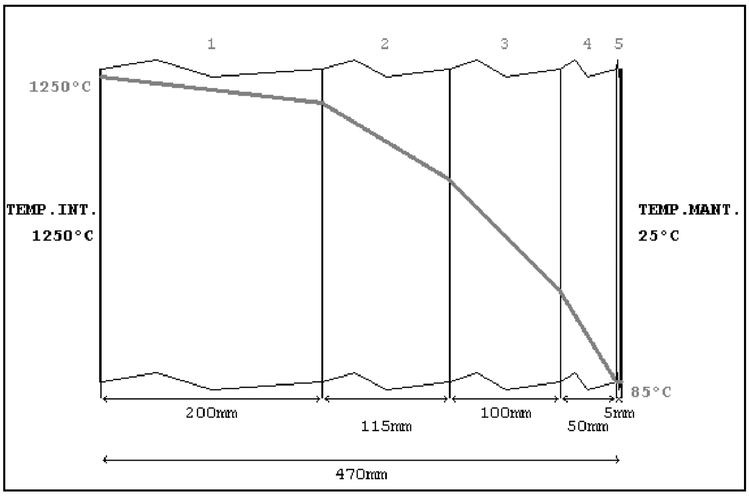


Figure 11. Drawing shows the stratigraphy of the walls of the furnace with the decrement of the temperature when the furnace is at the operating condition.

The application of the following gives the values of the heat flux in function of the time.

$$Q = U \cdot \Delta T \cdot A \tag{2}$$

Where U is the equivalent transmittance of the wall, ΔT is the difference of temperature and A is the area of the surface. However, with this formulation the thermal bridges were not considered.

4. Total energy loss and validation of the model

The knowledge of the trend of the heat flux in function of the time allowed the calculation of the energy loss. In particular Table 1 gives the energy for the three heat flux.

Table 1. Amount of energy lost for an opening of 600 s for radiation, convection e conduction.

Type ofheatflux	Energy [MJ]
Radiation	313
Convection	5252
Conduction	41
TOTAL	**5606**

To evaluate the accuracy of these values an indirect analysis through the knowledge of the methane consumption of the furnace was made. Assuming a lower calorific value of the natural gas equal to 34 MJ/m3, the flow rate is equal to 165 Nm3 of methane. However the methane consumption calculated is relative to the model created with the ideal operating condition of the furnace. The real behavior instead is conditioned by many parameters such as, for example, the presence of thermal bridges and damaged refractories which increase the heat flux through the walls. Another aspect is the presence of an interstice between the bogie hearth and the lateral walls of the furnace, which is necessary to avoid the frictions during the movement, but it allows part of the heat to exit from the furnace increasing the losses. These and other phenomena influence the real consumption of natural gas increasing the total amount of fuel necessary to maintain the furnace chamber at a certain temperature.

The validation of the model was carried out through an experimental test made in SdF. The test started with the opening of the door of an empty forging furnace at the operating temperature of 1230°C. The furnace remained open for 10 minutes with the bogie hearth inside and then closed again.

 doi:10.20944/preprints201608.0029.v1

Peer-reviewed version available at Energies 2016, 9, 833; doi:10.3390/en9100833

When the door was completely closed the burners started to call up to reach again the operating temperature. Two instants of this experimental test are reported in Figure 12 where it is possible to see the variation of the intensity of the furnace colour which corresponds to a sudden fall in temperature.

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

Figure 12. Pictures taken during the experimental test made in SdF: (**a**) The picture is related to a partial opening at 60 s; (**b**) The picture is related to a complete opening at 157 s .

The consumption of methane was monitored between the closing time, after the 10 minutes, and the time to return at the operating conditions. Taking into account the simplification done in the model, the two values of methane consumption were compared and resulted that the values are of the same order of magnitude. This validation guarantees the accuracy of the model and allows us to use it as a decisional instrument for future developments which will allow us to obtain more efficient processes.

In order to characterize the footprint of this phenomenon, the emission of CO_2 caused by the combustion of the 165 Nm^3 of methane is evaluated. From the stoichiometric equation of the methane combustion results that 1 mol of CH_4 produces 1 mol of CO_2 (equation 3):

$$CH_4 + 2O_2 \rightarrow CO_2 + 2H_2O \quad (3)$$

Being the molar mass of the methane equal to 16 g/mol and the molar mass of the CO_2 equal to 44 g/mol, results that the combustion of 1kg of CH_4 produces 2,75 kg of CO_2. For a density of the methane equal to 0,73 kg/m^3 results that 1 kg of CH_4 corresponds to 1,4 m^3 of CH_4. So the kg of CO_2 produced by the combustion of 165 Nm^3 of methane are 331 kg, where 309 kg are due to the combustion of natural gas for the heat lost for convection.

5. Application of the model

As shown in Table 1, the major contribution to the total heat loss is due to convection. For this reason the model created has been applied to evaluate how an increase of the door speed can influence the convective fluxes of the furnace and the consequent global energy performances. For this reason two other simulations of a complete opening of 40 s and 60 s were done. In the following Figure 13 the trend of the convective heat flux on the Surface A for the three different openings is shown.

Peer-reviewed version available at Energies 2016, 9, 833; doi:10.3390/en9100833

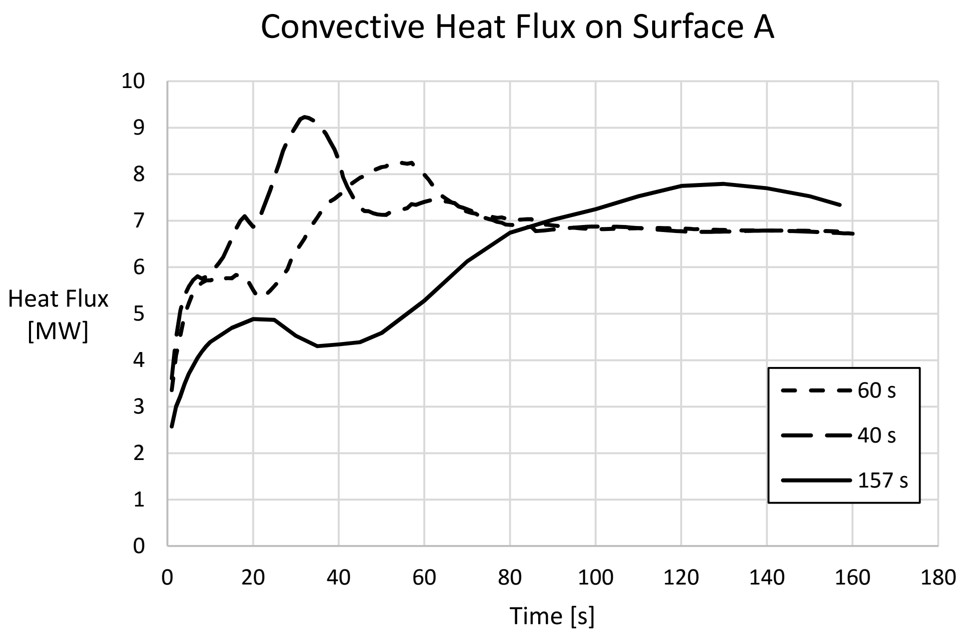


Figure 13. Convective heat flux evaluated on the control surface A for the three different opening of 40 s, 60 s and 157 s.

As it is possible to see, a reduction of the opening time entails an increase of the maximum value of convective heat flux calculated, which exceeds 9 MW for a 40 s opening. From the evaluation of the amount of the energy lost during the lifting time of the door and in the next 10 minutes for the three different cases of study, results that the opening of 40 s is the one which minimizes the heat loss with a total of 4324 MJ for a total time of 640 s. In the following Table 2 the values of the convective heat flux for the different openings are shown.

Table 2. Energy fluxes evaluated on the control Surface A in the three different cases of study for the time of the door opening and the total time.

Time [s]	Energyflux [MJ]	Time [s]	Energyflux [MJ]
40 s	499	640 s	4586
60 s	607	660 s	4664
157 s	1304	757 s	5252

Thanks to this study the company made a more efficient production process, saving fuel and limiting pollutant emissions, simply by reducing the opening time of the door, from 157 s to 40 s. An amount of about 60 kg of CO_2 is saved with the implementation of the new opening. A further reduction of this time is not allowed due to the mechanical limits of the door lifting system. The data obtained from this study is also being used for other studies concerning the design of an air knife system to install close to the door.

6. Conclusion

In this study the behavior of a real industrial furnace used in the forging industry was analyzed. The CFD simulation of a typical insertion/extraction operation done in a forging furnace showed that the amount of the energy lost for a total time of 10 minutes is equal to 5606 MJ, where the main contributory factor is the convective heat flux with a value of 5252 MJ. In order to characterize the heat flows during all the transient time, other variables were also monitored. In particular, results showed that at the opening, because the difference of temperature and pressure internally and externally, turbulences were generated constituting the main reason for the heat loss. The validation of the model, carried out through an experimental test done in SdF, has allowed the use of the model as a decisional instrument. For this reason two other simulations were also done. Results showed that a reduction of the door opening time from 157 s to 40 s, reduces the energy loss, limiting the fuel consumption and pollutant emissions. The importance of this study is due also to the fact that the model created is applicable to a very large variety of furnaces, as it depends only on the geometry and on the operating conditions. The model can be used for furnaces of every dimension and powered by any kind of energy.

References

1. Nastasi B., Lo Basso G. Hydrogen to link heat and electricity in the transition towards future Smart Energy Systems. Energy, 2016.
2. Edenhofer O. et al. Climate Change 2014: Mitigation of Climate Change. Contribution of Working Group III to the Fifth Assessment Report of the Intergovernmental Panel on Climate Change IPCC (2014). Cambridge University Press, Cambridge, United Kingdom and New York, NY, USA.
3. Castellani, B., Rossi, F., Filipponi, M., Nicolini, A. Hydrate-based removal of carbon dioxide and hydrogen sulphide from biogas mixtures: Experimental investigation and energy evaluations, Biomass and Bioenergy, **2014**, 70, 330-338.
4. Cotana, F., Rossi, F., Nicolini, A., Rinaldi, S., Merico, M.C. Electric vehicles for postal service equipped with a kinetic energy recovery system, International Journal of Green Energy, **2015**, 12 (5), 485-492.
5. Rossi, F., Castellani, B., Presciutti, A., Morini, E., Anderini, E., Filipponi, M., Nicolini, A. Experimental evaluation of urban heat island mitigation potential of retro-reflective pavement in urban canyons, Energy and Buildings, **2016**, 126, 340-352.
6. Brinchi, L., Castellani, B., Rossi, F., Cotana, F., Morini, E., Nicolini, A., Filipponi, M. Experimental investigations on scaled-up methane hydrate production with surfactant promotion: Energy considerations. Journal of Petroleum Science and Engineering, **2014**, 120, 187-193.
7. IEA. (2015), Energy Technology Perspectives 2015, OECD Publishing, Paris DOI, Available online: http://dx.doi.org/10.1787/energy_tech-2015-en.
8. Castellani B.; Morini E.; Filipponi M.; Nicolini A.; Palombo M.; Cotana F.; Rossi F. *Comparative analysis of monitoring devices for particulate content in exhaust gases*; Sustainability **2014**, 6, 4287-4307
9. Sang Heon Han, Daejun Chang, Cheol Huh. Efficiency analysis of radiative slab heating in a walking-beam-type reheating furnace, Energy, **2011**, 36, 1265-1272.
10. Lei Zhang, Haihong Huang, Di Hu, Bingbing Li, Cheng Zhang. Greenhouse gases (GHG) emissions analysis of manufacturing of the hydraulic press slider within forging machine in China, Journal of Cleaner Production, **2016**, 113, 565-576.
11. J.G. Kim, K.Y. Huh, I.T. Kim, Three-dimensional analysis of the walking-beam type slab reheating furnace in hot strip mills, Numer. Heat. Transfer Part A Appl. 38 (2000) 589-609.
12. Jaklic, T. Kolenko, B. Zupancic, The influence of the space between the billets on the productivity of a continuous walking-beam furnace, Appl. Therm. Eng. 25 (2005) 783-795.
13. M.Y. Kim, A heat transfer model for the analysis of transient heating of the slab in a direct-fired walking beam type reheating furnace, Int. J. Heat Mass Transfer 50 (2007) 3740-3748.
14. J.H. Jang, D.E. Lee, M.Y. Kim, H.-G. Kim, Investigation of the slab heating characteristics in a reheating furnace with the formation and growth of scale on the slab surface, Int. J. Heat Mass Transfer 53 (2010) 4326-4332.
15. S.H. Han, D. Chang, C.Y. Kim, A numerical analysis of slab heating characteristics in a walking beam type reheating furnace, Int. J. Heat Mass Transfer 53 (2010) 3855-3861.
16. S.H. Han, D. Chang, C. Huh, Efficiency analysis of radiative slab heating in a walking-beam-type reheating furnace, Energy 36 (2011) 1265-1272.
17. S.H. Han, D. Chang, Radiative slab heating analysis for various fuel gas compositions in an axial-fired reheating furnace, Int. J. Heat Mass Transfer 55 (2012) 4029-4036.
18. S.H. Han, D. Chang, Optimum residence time analysis for a walking beam type reheating furnace, Int. J. Heat Mass Transfer 55 (2012) 4079-4087.

Preprints (www.preprints.org) | NOT PEER-REVIEWED | Posted: 3 August 2016 doi:10.20944/preprints201608.0029.v1

Peer-reviewed version available at *Energies* **2016**, *9*, 833; doi:10.3390/en9100833

19. Modeling of Solid and Bio-Fuel Combustion Technologies. In*Thermofluid Modeling for Energy Efficiency Applications 1*;Elsevier Publications; pp. 259-309.
20. Song G.; BjorgeT.; Holen J.; Magnussen B.F. Simulation of fluid flow and gaseous radiation heat transfer in a natural gas-fired furnace. *International Journal of Numerical Methods for Heat and Fluid Flow***1997**, 7, pp. 169–182.
21. JiyuanTu;Yeoh G.; Liu C. *Computational Fluid Dynamics*, 2nd ed.; Butterworth-Heinemann: Waltham, USA, 2013.
22. LaunderB.E.; SpaldingD.B. *Lectures in Mathematical models of turbulence*; Academic Press: London, England, 1972.
23. ShihT. H.; LiouW. W.; ShabbirA.; YangZ.; ZhuJ.A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows – Model Development and Validation. *Computers Fluids* 1995, pp. 227-238.
24. SiegelR.; HowellJ.R.*Thermal Radiation Heat Transfer*; Hemisphere Publishing Corporation: Washington DC, 1992.
25. Surface-to-Surface (S2S) Radiation Model Theory. *Ansys Fluent Theory Guide*, Release 16.2; AnsysInc: Canonsburg, PA 2015, 162-164.
26. M.F.Modest. *Radiative Heat Transfer*. Series in Mechanical Engineering McGraw Hill, 1993.
27. Near Walls treatments for Wall-Bounded turbulent flows. *Ansys Fluent Theory Guide*, Release 16.2; AnsysInc: Canonsburg, PA 2015, 107.
28. Dynamic Mesh Theory. *Ansys Fluent Theory Guide*, Release 16.2; AnsysInc: Canonsburg, PA 2015, 34-36.