ENVIRONMENTAL concerns and rising prices of fossil fuels have focused the steel industry’s attention on minimising emissions and improving heating efficiency in reheating furnaces. Additionally, with the rise in competitiveness within the world steel market product quality can be a key strategic factor to gain a market, a factor which arises in the reheating furnace by ensuring the correct heating rates, ultimate temperature, uniformity of temperature and furnace atmosphere.

All those targets need to be fulfilled without increasing capital expenditure and operating costs beyond acceptable limits. As a consequence, a successful project for a reheating furnace must accomplish all these targets to minimise energy consumption, limit emissions and ensure product quality as far as is practically possible.

This scenario poses various questions when designing or modifying a reheating furnace, many of which can provide valuable answers through integration of advanced design and verification methodologies into the common engineering workflow.

Various attempts to evaluate the heating process inside a furnace have been made in past years with different levels of detail and accuracy. As it is well understood that radiation is the main mechanism of heat transfer involved in heating the charge, simplified models are commonly used to analyse performance and efficiency of reheating furnaces. The well-established zonal methods range from single or two dimensional models for preliminary estimation of charge temperature to three dimensional models able to account for edge effects in slab and square bloom or billet passing through a walking beam furnace as well as the cooling effect of the skids that carry the charge through the furnace[1,2].

Nevertheless, such models cannot give an indication about the local combustion processes which develop inside the furnace, nor can they give an indication of expected emissions levels. From the experimental side, existing methods of measuring cannot provide continuous measurements of temperature inside the furnace.

Trailng thermocouples or data loggers can be used only with test charges to give an indication of the heating process at a few points measured inside the charge, while optical pyrometers or thermal cameras give information only about the surface temperature of the charge and their accuracy is greatly affected by the presence of the surface oxide layer (scale) on the charge. Therefore in recent years two main developments are on-going thanks to continuously increasing computational power: These are:

- Online three dimensional control tools based on existing zonal models extended to simulate transient radiation heat transfer[3].
- Three dimensional Computational Fluid Dynamic (CFD) simulations of the complete furnace, coupling the reacting flow combustion models developed for single burners, the heat radiation model and conjugated heat transfer models of charge heating[4,5].

This article is focused on the definition of a CFD model able to simulate the large rotary hearth furnace at the TenarisDalmine pipe mill and thus able to evaluate the fluid dynamics of the furnace gases, quantify the thermal and chemical species inside the furnace and provide a fine scale representation of the heating process of the charge.

This rotary hearth is equipped with TRGX flameless regenerative burners and TRX flameless roof burners with the aim of minimising polluting emissions and maximizing furnace efficiency through pre-heating the combustion air to a high temperature.

The established CFD methodology is integrated with the computational fluid dynamics approach that Tenova currently applies for the design, verification and continuous improvement of combustion systems. Using this successful combination combined with different levels of analysis, a complete multiscale methodology for furnace design is presented and the excellent results on an industrial installation are presented.

**CFD methodology**

The physical and numerical model currently adopted by Tenova for verification of burner design, and continuous improvement[6,7,11,12] have been extensively validated against experimental data from CSM test rigs. A complete database of CFD simulations for all the range of Tenova’s ‘FlexyTech’ burners of various sizes and at discrete turndown levels is continuously populated with newer data from the CFD activities undergoing in the R&D projects between CSM and Tenova. This creates the fundamental knowledge base that is required for the successful application of different combustion technologies such as flameless and regenerative combustion.

In particular, regenerative burners pose technological issues for rotary hearth furnaces and also generally for any reheating furnace with narrow chambers. To have the characteristic cycling behaviour between the regeneration and firing phase for each burner pair, it is necessary to install burners with double the design output size[13,14] as one of each burner pair is inoperative during the regeneration cycle. As a consequence, the flame length increases and serious questions arise about the possible ingestion of partially reacted fuel/air mixture by the burner on the opposite wall, with resulting detrimental effects on both combustion efficiency and regenerative bed maintenance. This effects is of particular concern in relatively narrow chamber furnaces such as those of a rotary hearth furnace.

To evaluate this effect and design flameless regenerative burners able to cope with these
Reheat furnace

issues, Tenova started a CFD design project. The results of this study and those for the other burners equipping the rotary hearth furnace is briefly reviewed because they are at the very basis of the entire CFD approach used for the simulation of the complete furnace.

Short flameless regenerative burner

The TRGX burner design was first realized in 2006 when Tenova decided to integrate flameless technology with regenerative combustion to obtain a burner able to guarantee both low levels of pollutants and low fuel consumption. TRGX FlexyTech Flameless Regenerative burners represent the latest generation of regenerative burners; they can work both in flame mode (for cold ignition) and flameless mode (to reach the best performance in terms of NOx emissions). Thanks to coupled gas and air staging when working in the flameless mode, NOx emissions are brought down to 35-40ppm.

The first development stages of the TRGX burner were carried out by Tenova in cooperation with the Italian research centre, CSM by means of an iterative procedure consisting of the design of the burner prototypes based on the experience gained in the engineering of the previous TSX (flameless) and TRG (regenerative) burners. Extensive CFD modelling to optimise burner design, laboratory furnace tests and industrial furnace tests were carried out. The selection of the physical models has been based on previous extensive validation work dedicated to evaluate the performance of the different representations of turbulence to simulate high velocity round jets and combustion schemes for natural gas[17].

Flameless combustion is achieved through intensive recirculation of the hot exhaust gases inside the furnace chamber promoted by high momentum air jets emanating from the burner diffuser. As a consequence, the flame length, which is intended to be an extension of the reactive zone inside the furnace chamber in the case of flameless combustion, is particularly high with respect to that realised in traditional regenerative burners. Moreover, this effect is exaggerated as regenerative burners are installed at double the size of non-regenerative to account for their cycling between their regeneration firing phases. These were the reasons for embarking on the R&D project to develop a short flame version of the TRGX flameless regenerative burner.

The main aim in this R&D project was to maintain the high levels of air dilution while limiting the reaction zone inside a more compact volume than the original long flame of the TRGX Flameless Regenerative Burner. The technical solution was found by imparting the correct amount of swirl to the combustion air flow while maintaining the air jet perimeter at a fixed value in order to dilute it with hot exhaust gases at approximately the same level as for the long flame TRGX burners. Additionally, the fuel injection lances were modified to provide greater uniformity of fuel distribution around the swirling air jet. The entire design and verification was carried out relying on the CFD models. The modelling set-up for simulating of a single natural gas burner (Table 1)[12,13] was adopted. In particular, the ‘Two step reduced mechanism’ used by Westbrook & Dryer[11] was selected as the best compromise between accuracy and computing time to simulate the process of combustion of natural gas[12,13].

The resulting performance in terms of expected flame length inside the 6 metre long experimental chamber at the CSM test rig are shown in Fig 1 where the flame temperature iso-surface is coloured in terms of CO₂ mole fraction [kmol/kmol]. The top illustration is for the traditional long flame FlexyTech TRGX burner and the lower for the short compact flame version. It should be highlighted how the reaction zone, marked by the iso-temperature surface at 1500°C (red), is confined within the length of the test chamber for the short flame version whereas for the traditional long flame TRGX flameless regenerative burner it extends beyond the chamber.

Following this first development phase, a large number of experimental tests were carried out at CSM test facilities in Dalmine to verify thermal load profiles on the test furnace walls and also the emission levels of gases[16]. The development programme resulted in a successful conclusion and the Tenova FlexyTech TRGX burner family was further extended to include the short flame version for narrow chamber furnaces. Short flame TRGX 16 burners were successfully installed in zones 1 and 2 of the rotary hearth furnace at TenarisDalmine, while long flame TRGX-14 burners were installed in zone 3 where there was no concern about the extension of the flame length because of the burner’s lower power output.
Reheat furnace

CFD model setup

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Ideal gas</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence</td>
<td>k-ω with Shear Stress correction (Wilcox) Reynolds Stress model</td>
</tr>
<tr>
<td>Chemistry</td>
<td>Finite Rate/Eddy Dissipation with A = 4 and B = 0.5 Eddy Dissipation/Arrhenius rate of reaction Westbrook &amp; Dryer reduced kinetic mechanism</td>
</tr>
<tr>
<td>Species</td>
<td>Specific heat: corrected for dissociation (Pelders) Species Specific heat, viscosity &amp; thermal conductivity: ideal gas mixing law</td>
</tr>
<tr>
<td>Mixture</td>
<td>Density: ideal gas Specific heat, viscosity &amp; thermal conductivity: ideal gas mixing law</td>
</tr>
<tr>
<td>Thermophysical properties</td>
<td>Density: ideal gas Specific heat, viscosity &amp; thermal conductivity: ideal gas mixing law</td>
</tr>
<tr>
<td>Radiation</td>
<td>Discrete Ordinate model Absorption coefficient: WSGGM</td>
</tr>
</tbody>
</table>

Table 1 Reference operating conditions for the CFD simulation.

CFD analysis of flameless roof burner

Zones 4, 5, 6 and 7 of the rotary hearth furnace at TenarisDalmine are equipped with TRX Flameless roof burners (Fig 2). Development of this burner family was carried out in cooperation between CSM and Tenova and resulted in a large database of numerical simulation results. Different turndown levels, air excess values, air preheating temperatures and fuel staging ratios were simulated for nearly all the available roof burner sizes. Zones 4, 5, 6 and 7 are entirely equipped with TRX-4 flameless roof burners.

Multiscale CFD simulation of complete reheating furnace

The physical and numerical model used for single burner design, improvement and verification phases[8] were the starting point to extend these CFD models to the complete furnace to analyse possible interaction effects between the burners. The main problems in conducting this analysis using CFD technique are:

- the large number of individual computational cells required to sufficiently resolve the various geometric scales (eg burner air and fuel ports, furnace dimensions and charge dimensions);
- the modelling of the passage of the charge through the furnace.

To represent the burner elements with sufficient detail and to obtain sufficient spatial resolution in the region of the flame interactions, a computational mesh with tens of millions of cells is required. Consequently, to reduce the computational time required, a procedure has been developed by CSM to reduce the number of cells needed.

This takes into account that over half the cells are required to represent the sharp temperature gradients as well as gas velocity, pressure, species concentration and turbulence, around the burners. It is evident that the multiscale approach to simulate the whole furnace is required (Fig 3).

The method developed and validated for every furnace simulation extracts the profiles of the relevant variables at the burner outlet from the large database of CFD simulations available and applies those values as fixed boundary condition patches to the complete furnace model. To facilitate the application of this approach, in the case of multiple burners, dedicated software has been developed[17] that enables the rotation/translation of the profile to the required position using the complete furnace grid domain and enables interpolation between possible profiles previously simulated and stored in the database.

Moreover, a representation of the heating of the charge requires a time dependent calculation due to its passage through the furnace and consequently requires a very large amount of computation time. To overcome this limitation the problem has been resolved using a multiscale approach involving iteration to refine the result (Fig 3). A coupling is achieved between the CFD calculation that is iteratively performed on the charge taking a stationary point as it passes through the furnace and the simulation of the flow of the hot gases inside the furnace (furnace fluid model) along with the unsteady state derived from the simulation of the conduction of heat within the charge during its passage through the furnace.

Reduction of the computation time is achieved by simulating the fluid pattern of the gases within the furnace in a steady state while the pattern of heat conduction through the solid charge is simulated in an unsteady state. It is necessary to iterate each stage in order to obtain convergence with respect to the energy balance. If the fluid pattern of the furnace gases was also to be simulated in an unsteady state the computation time will be unrealistically long. This approach is possible due to the different characteristic time scales of the two processes: the fluid dynamic time scale is of the order of tenths of seconds while the time scale in heating the solid bloom is of the order of seconds. Hence, the surface temperature of the charged bloom can be assumed constant with time during the fluid-dynamic calculation while the resulting heat flux on the blooms can be assumed as a fixed boundary condition during the simulation of the movement of the charge inside the furnace.

The simulation of heat transfer in the solid bloom reports the updated surface temperature of the charged bloom to the fluid simulation in an iterative process that stops when the difference between the surface temperature in two successive iterations is less than the tolerance value set.

The charge model, developed in the User-Defined-Function (UDF) and the macros[15] framework of the Ansys FLUENT® software used for the simulations has been presented in detail and successfully compared with experimental heating curves[10] and with a simplified model in which the passage of the steel charged to the furnace is represented as a continuous solid travelling at a constant velocity (equivalent strip concept)[17].

200/t HRF

In the case of the rotary hearth furnace the charge is a round solid bloom. The good predictions achieved and the shorter CPU time required indicates that the CFD coupled model
Reheat furnace

![Heat flux](image)

**Table 2 Reference operating conditions for the CFD simulation**

<table>
<thead>
<tr>
<th>Zone</th>
<th>Fuel thermal input [kW]</th>
<th>Air thermal input [kW]</th>
<th>Power split [%]</th>
<th>Zone turn-down [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>6612</td>
<td>3108</td>
<td>10</td>
<td>32</td>
</tr>
<tr>
<td>2</td>
<td>20532</td>
<td>9671</td>
<td>32</td>
<td>62</td>
</tr>
<tr>
<td>3</td>
<td>15900</td>
<td>7982</td>
<td>25</td>
<td>71</td>
</tr>
<tr>
<td>4</td>
<td>8940</td>
<td>2084</td>
<td>12</td>
<td>46</td>
</tr>
<tr>
<td>5</td>
<td>7210</td>
<td>1678</td>
<td>9</td>
<td>46</td>
</tr>
<tr>
<td>6</td>
<td>4155</td>
<td>965</td>
<td>5</td>
<td>46</td>
</tr>
<tr>
<td>7</td>
<td>4176</td>
<td>965</td>
<td>5</td>
<td>22</td>
</tr>
</tbody>
</table>

**Table 3 Reference operating conditions for CFD simulation for each furnace zone**

<table>
<thead>
<tr>
<th>Zone</th>
<th>Burner model</th>
<th>Burner type</th>
<th>Number of burners</th>
<th>Fuel flow rate [Nm³/h]</th>
<th>Fuel flow rate [Nm³/h]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>TRX416</td>
<td>Regenerative flameless</td>
<td>13</td>
<td>95</td>
<td>1154</td>
</tr>
<tr>
<td>2</td>
<td>TRX416</td>
<td>Regenerative flameless</td>
<td>22</td>
<td>188</td>
<td>2285</td>
</tr>
<tr>
<td>3</td>
<td>TRX414</td>
<td>Regenerative flameless</td>
<td>22</td>
<td>200</td>
<td>2593</td>
</tr>
<tr>
<td>4</td>
<td>TRX4</td>
<td>Flameless</td>
<td>48</td>
<td>19</td>
<td>225</td>
</tr>
<tr>
<td>5</td>
<td>TRX4</td>
<td>Flameless</td>
<td>52</td>
<td>14</td>
<td>167</td>
</tr>
<tr>
<td>6</td>
<td>TRX4</td>
<td>Flameless</td>
<td>52</td>
<td>8</td>
<td>96</td>
</tr>
<tr>
<td>7</td>
<td>TRX4</td>
<td>Flameless</td>
<td>44</td>
<td>8</td>
<td>101</td>
</tr>
</tbody>
</table>

**Table 4 Working conditions for each burner type**

The velocity components of the fuel gas and the combustion air emanating from the burner along with the kinetics of the gas turbulence and dissipation of temperature and concentration of the compounds obtained at a defined section at the burner outlet have all been applied as boundary condition to represent the 253 burners installed in the complete furnace. The boundary conditions of the complete furnace CFD model are the following:

- Furnace walls: mixed (radiation and convective heat transfer) wall boundary condition with material conductivity and thickness of lateral wall, roof, bottom and tunnel;
- Burners: velocity inlets with profile of velocity components, turbulence, temperature and species concentrations extracted from single burner simulations;
- Waste gases outlet: pressure at outlet;
- Charging door: transparent window exchanging radiation heat flux with outside ambient at 25°C;
- Furnace wall ports: transparent windows exchanging radiation heat flux with outside ambient at 25°C;
- Discharging door: transparent window exchanging radiation heat flux with outside ambient at 25°C.

The representation of this very large reheating furnace is based on a grid analysis performed for each individual furnace zone[17,15] and also on previous validation tests for a walking hearth furnace. It required 100 million computational tasks for the algorithm to be implemented using the latest version of Ansys Fluent. This enables clustering of the tetrahedral elements in the polyhedral grid.

An example of the result of the polyhedral transformation of the grid near the burners is taken from the work of M Stazi[15], together with the comparison of temperature profiles at two axial locations (Fig 5). In this way, the grid size in the case of the rotary hearth furnace is reduced by 75%, to a total of about 25 million cells.

**Fig 6** shows the furnace temperature and furnaces gases, velocity fields projected on a plane at the level of the lateral burners inlets. The interaction of the burner jets with the flue gas stream producing bending of the flame in zones 1 to 3 is evident.

The heating curve (Fig 7) confirms that for the revamped furnace it is necessary to maintain an extended soaking zone to homogenize the temperature of the charged blooms and
comply with the target for temperature uniformity of the bloom at the furnace exit both in the radial direction from the surface to the centre of the bloom (\(\Delta T_{\text{max}} = 18^\circ\text{C}\)) and in the longitudinal direction (\(\Delta T_{\text{max}} = 6^\circ\text{C}\)). The simulation shows \(\Delta T_{\text{max}} = 433^\circ\text{C}\). This temperature is the maximum temperature difference in the bloom reached during heating and occurs after approximately 100 minutes (in zone 3). \(\Delta T = 18^\circ\text{C}\) is the target temperature difference between surface and centre of the bloom at the discharge side of the furnace after 320 minutes (zone 7). \(\Delta T = 23^\circ\text{C}\) is the calculated temperature difference along the length of the bloom measured along its central axis. It has to be compared with the target value of 18\(^\circ\text{C}\) for the furnace.

A 3D view of the surface temperature map of the furnace is shown in Fig. 8 along with surface temperature maps for individual blooms at the exit point of each zone of the furnace. An in-depth analysis of the heat flux on the bloom (Fig. 9) illustrates – as expected – that the main contribution of heating is by radiation (red line) with the exception of the furnace in/out zone where the wall temperature is the lowest (700\(^\circ\text{C}\) in Fig. 10) and the recirculation zones are generated by the presence of the furnace baffles and ploughshare – the function of the ploughshare being to remove scale from the hearth.

The ploughshare is installed between two baffles located close to the charging and discharging doors. In contrast, in zone 3 the radiant heat flux is unexpectedly negative indicating that the bloom surface temperature is higher than that of the wall here. The high value of convective heat flux indicates that this anomaly is mainly due to the impingement of the flames from the side wall burners onto the bloom surface. Thus, more detailed information is obtained compared to that available from the other models discussed earlier.

Looking at furnace heat balance (Fig. 10) and Table 5, it is possible to evaluate the efficiency of the furnace after revamping. The temperature of the combustion air for the side burners due to the installation of the regenerative system rises from 450\(^\circ\text{C}\) to 890 – 1120\(^\circ\text{C}\) which results in an increase in efficiency from 45% to 67% and the maximum production to increase from 160\(\text{t/h}\) to 215\(\text{t/h}\).

The CFD simulation shows that the different sources of heat losses can also be evaluated. It shows 8.8% are thermal losses from the furnace walls made up of 5.4% from the walls and 3.4% from the water cooled ploughshare.

The heat content of the flue gases from the furnace is 63.6%, of which 68% is recovered by the high efficiency regenerative burners and the remaining 32% remaining in the flue gases exits the furnace outlet and is recovered by a central recuperator of lower efficiency.

Fig. 11 shows the comparison between the zone temperatures measured after the revamp of the furnace by Tenova in August 2010 when the furnace was operating in a condition similar to that for which the simulation was made. Table 5 shows that the temperature selected for the furnace control system (the mean value between the measured value at the inlet and the outlet of each zone) is very close to the calculated average temperatures of the wall zones.

Conclusions

The CFD modelling developed for the single burner simulation has been extended to the complete reheating furnace including the effect of the passage of the charge through the furnace zones. A multiscale approach to represent the burners has been applied to reduce the number of cells in the computational grid for the simulation without losing accuracy.

This is used in conjunction with a model for the charged steel that couples the steady state CFD calculation of the heat flow inside the furnace with the unsteady simulation of the charge as it advances through the furnace. This approach has been experimentally validated in a previous study.

The results of the validation work and the CPU time required (5-12 days) on a cluster (ie a high performance cluster) are used to reduce the number of cells in the computational grid necessary for the simulation with acceptable accuracy.

While traditional methods of process design of a furnace are faster, this innovative approach requires computing time of the same order of time even although it is more complex. A greater number of results are achieved allowing a better understanding of the behaviour of a reheating furnace.

The application of the CFD simulation in the framework of revamping an industrial reheating furnace and the comparison with its operation in industry has confirmed the potential of CFD as a tool for the analysis of fluid dynamics, flame interactions, detailed representation of furnace wall heat losses, steel charge temperature and heat fluxes necessary to support the furnace design. Moreover, the availability of the charge surface temperature, heat fluxes and local concentration of combustion products potentially allow a multi-physical methodology.

CFD could be coupled with a structural analysis tool to evaluate the deformation of the charge due to the thermal gradient or the growth of surface scale to evaluate process yield.

The principal directions for future research between Tenova and CSM are to reduce the CPU time following the current multiscale approach and investigate the use of alternative gaseous fuels (COG, BFG and producer gases) or oxygen enriched combustion air inside reheating furnaces. CFD will enable the evaluation of such factors at the design phase of a furnace by analysing all the relevant details for the combustion process inside the furnace.

Acknowledgments

The authors wish to thank A Caprera & M Galliano of TenarisDalmine who made available the reference data for the rotary hearth furnace.

References