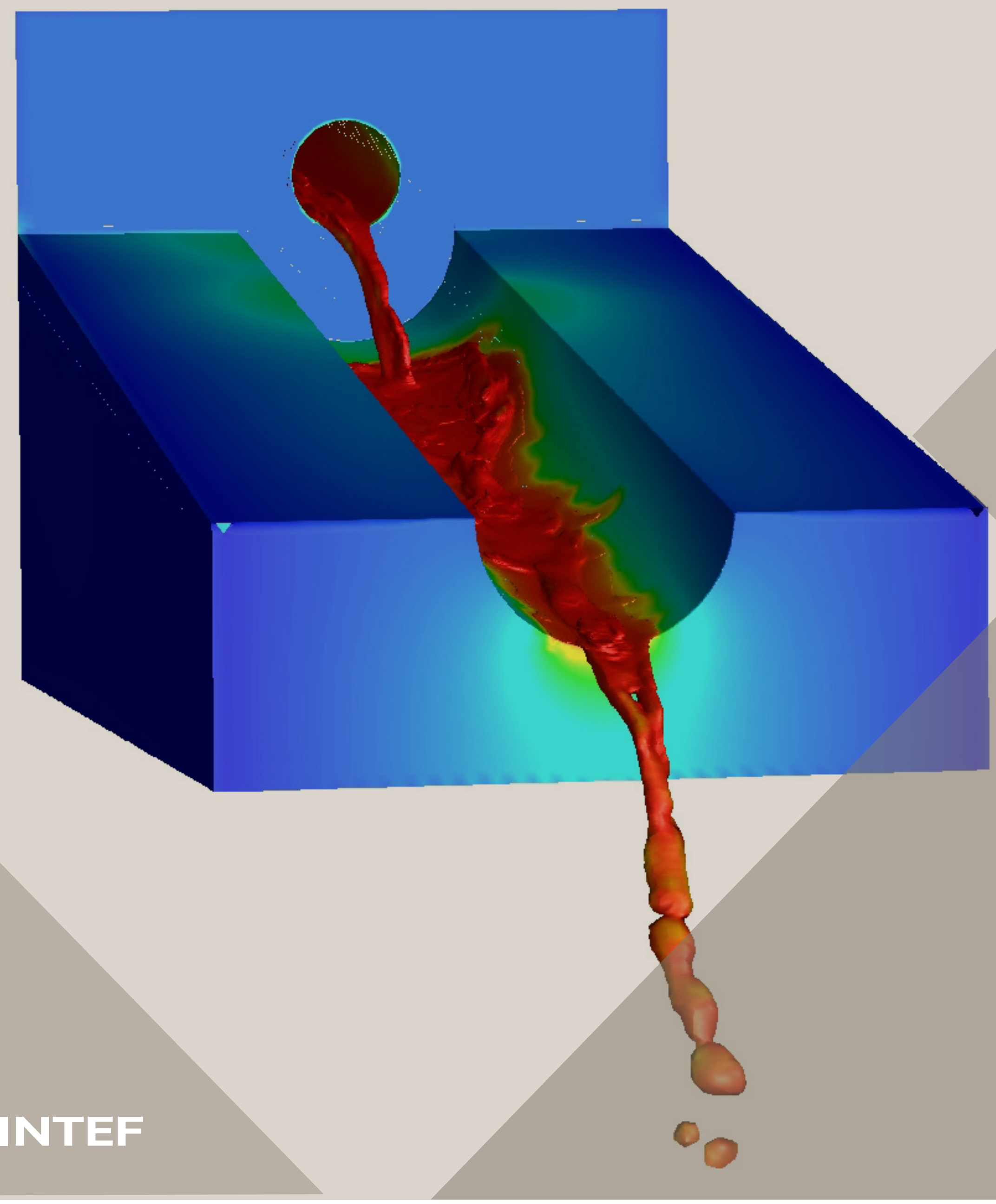
SINTEF
**PROCEEDINGS**

6

14th International Conference on CFD in
Oil & Gas, Metallurgical and Process Industries
SINTEF, Trondheim, Norway, October 12–14, 2020

# Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries



SINTEF

SINTEF Proceedings

Editors:
Jan Erik Olsen, Jan Hendrik Cloete and Stein Tore Johansen

# Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries

SINTEF, Trondheim, Norway
October 12–14, 2020

SINTEF Academic Press

SINTEF Proceedings 6
Editors: Jan Erik Olsen, Jan Hendrik Cloete and Stein Tore Johansen

Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries, SINTEF, Trondheim, Norway, October 12–14, 2020

Keywords:
CFD, fluid dynamics, modelling

Cover illustration: Tapping of metal by Jan Erik Olsen

ISSN 2387-4295 (online)
ISBN 978-82-536-1684-1 (pdf)

SINTEF Academic Press
Address: Børrestuveien 3
PO Box 124 Blindern
N-0314 OSLO
Tel: +47 40 00 51 00

www.sintef.no/community
www.sintefbok.no

SINTEF Proceedings

SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings on a variety of scientific topics.

The processes of peer-reviewing of papers published in SINTEF Proceedings are administered by the conference organizers and proceedings editors. Detailed procedures will vary according to custom and practice in each scientific community.

*14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries*
*SINTEF, Trondheim, NORWAY*
*October 12-14, 2020*

# CONTROLLING FLUE GAS TEMPERATURE FROM FERRO SILICON SUBMERGED ARC FURNACES (SAF) USING FLUE GAS RECIRCULATION (FGR)

**Balram PANJWANI**[1 *], **Torbjørn PETTERSEN**[1 †], **Bernd WITTGENS**[1 ‡]
[1]SINTEF Industry, 7465 Trondheim, NORWAY

[*] E-mail: balram.panjwani@sintef.no
[†] E-mail: torbjorn.pettersen@sintef.no
[‡] E-mail: bernd.wittgens@sintef.no

## ABSTRACT

Flue gas Recycling (FGR) is a well-known method for NOx reduction. A feasibility study is presented on the potential use of FGR in ferro-silicon production. The aim of the study is to illustrate how recycling of flue gas into the furnace for temperature control will affect local temperatures and NOx formation in the furnace hood (the flue gas combustion zone) of a conventional furnace design. Computational fluid dynamic (CFD) simulations using a generic model of a submerged arc furnace (SAF) developed in previously NFR financed projects like ProMiljø are performed. The SAF model consists of seven charging pipes, three electrodes and one flue gas stack. ANSYS FLUENT was used for modelling the interaction between process gas, ambient air, and flue gas. The simulation results show that introduction of recirculated flue gas affects the peak temperatures since the reduced oxygen concentration of flue gas significantly reduce the reaction rates compared to injection of air. A corresponding effect on NOx formation has been demonstrated, results indicate an order of magnitude reduction in NOx formation when recirculated flue gas ( 6vol% $O_2$) is used in the combustion zone instead of air (21vol% $O_2$). Simulations of the rapid increase in NOx production during an avalanche within the furnace is simulated using theoretical flow profiles. The effects of 1) recirculated flue gas, 2) rapid increase in the process gases from charging bed (burst), and 3) effect of radiation on NOx have been studied. The study showed that FGR has significant effect on NOx reduction. The study also showed that accounting for radiation is very relevant for an accurate estimation of NOx. The formation of process gas burst through a charging surface increase the rate of NOx formation.

**Keywords:** CFD, NOx, radiation, combustion, flue gas recycling (FGR).

## INTRODUCTION

Ferroalloys are produced in a submerged arc furnaces (SAF) where ore and carbon (coke, coal, etc.) are mixed inside the furnace and allowed to react. The electric energy for the reaction is supplied through electrodes. Furnace operation and raw material properties determine the metal yield and quality of the metal and how much process gas will be formed. Process gases mainly consisting of CO and metal oxide are formed underneath the charge surface due to the reduction processes and these process gases travel upward through the charge surface into the furnace hood. The liquid metal sink to the bottom where it is collected in ladles through a tapping hole. As the hot process gas rises upwards through the charge surface into a furnace hood, in case of open furnace hood, air is sucked into the hood through various open areas on the furnace walls due to the pressure drop. The air and process gas reacts inside the hood in a combustion process and produces an off-gas containing $SiO_2$, $CO_2$, $H_2O$ and other components. In the open furnace, most of the chemical energy of the process gas is lost due to uncontrolled combustion of CO inside the hood. The uncontrolled combustion can be hindered by closing the furnace hood which prevents the reaction between fresh air and process gases. However, closing the Si furnace is a challenge due to many practical and technical constraints and therefore closed Si Furnaces are not in industrial use today. Nevertheless, in the Open furnace usually, temperature and material stream are high enough, and it is possible to produce electricity from the heat. In an open furnace, there are two main potential sources of energy recovery, 1) from the off-gas with a high temperature and 2) from the cooling water used for cooling the SAF. An energy analysis carried out by Kamfjord et al. (Kamfjord, 2012) has shown utilization of hot water obtained from the furnace for other industries including agriculture, sports etc. Some of the Si plant have installed a steam power plant to recover the energy.

One of the major challenges with open SAF is uncontrolled combustion of process gases inside the furnace hood resulting in an excessive NOx formation due to the formation of high temperature zones. There are various kinds of health related issues with NOx once it is released into the atmosphere. NOx can cause breathing problems, chronically reduced lung function, eye irritation, loss of appetite. It mainly contribute to the acid rain and formation of ground-level ozone that can damage the ecosystems. All the metallurgical companies have to follow the governmental regulations on the NOx emission and therefore these companies have been developing many techniques to reduce the NOx emissions. NOx emission can be reduced by primary methods such as water direct injection, water emulsification, flue gas recirculation (FGR) and secondary method such as selective catalytic reduction (SCR). Three main mechanisms have been identified for the NOx formation: the thermal or Zel'dovich mechanism, the Fennimore or prompt mechanism and $N_2O$ intermediate mechanism, corresponding NOx are respectively called thermal, prompt and fuel NOx. Thermal NOx generally dominates in high temperature turbulent diffusion flames. At temperature around 1527 °C, oxygen radicals are formed from the dissociation of atmospheric oxygen. These atoms react with nitrogen molecules and produce NOx and nitrogen atoms. The nitrogen atom again reacts with oxygen molecules and OH radicals and produces NOx. A detailed mechanism of NOx formation have been discussed in this paper.

An improved understanding of the combustion process inside the SAF is essential for minimization of the NOx formation and maximization of energy recovery. In SAF, high temperature processes makes it difficult to perform extensive experiments and most of these experiments are performed on the small pilot scale experiments. Numerical techniques such as Computational fluid dynamic (CFD) is a good alternative to understand the dynamics and functioning of SAF, CFD techniques can be deployed to understand complex solid-gas, liquid-solid, and liquid-gas reactions prevalent in SAF. Many studies have been performed on the modelling of SAF to understand the operational behavior of the furnace (Scheepers *et al.*, 2006a; Darmana *et al.*, 2012; Scheepers *et al.*, 2006b; Panjwani and Olsen, 2013; Kadkhodabeigi *et al.*, 2010). CFD tools validated with experiments enable us to understand the complex reactions between process gas and air taking place inside hood. CFD simulation of furnace hood (Panjwani and Olsen, 2013) indicates, many hot pockets inside the furnace hood which results in excessive NOx formation. The hot pockets are also responsible for radiation losses and reduces the potential for energy recovery. To alleviate this problem, a NOx reduction techniques flue gas recirculation (FGR) have been utilized.

The results from previous techno economical evaluations (Pettersen et al. (Pettersen *et al.*, 2017)) indicated the potential of using recirculated flue gas in silicon production as a mean for both increased energy recovery and simultaneously improved temperature control in a semi-closed submerged arc furnace. This potential has been further explored through the following activities:

- A base case has been defined for silicon production in a semi-closed submerged arc furnace
- Results from a computational fluid dynamics simulation of the combustion zone are presented.
- Some critical design trade-offs are discussed

FGR involves recirculating part of the flue gas back into the furnace or the burners to modify conditions in the combustion zone by lowering the peak flame temperature and reducing the oxygen concentration, thereby reducing thermal NOx formation. FGR has been used commercially for many years at coal-fired units, waste incinerators (WI), gas turbines(Tsiliyannis, 2013; Liuzzo *et al.*, 2007; Chen *et al.*, 2015; Guethe and Burdet, 2009). Flue gas recirculation (FGR) emerges as a promising method for reducing WI atmospheric pollution, mainly NOx and volatile metal emissions by resulting in lower total off gas volumes. In WI plant, a portion of flue gas is recycled back to the incinerator and the secondary combustion air is manipulated by measuring the oxygen concentration of the incineration chamber flue gas. Being a mass recycle, FGR is fundamentally different than the heat integration (exchange of the heat of flue gases with the feed, air or wastes); the latter redirects enthalpy to the WI and raises its temperature, whereas oxygen concentration in both the primary and secondary air are not affected. The application of FGR in new plants has allowed a reduction of the total amount of incineration air and flue gas in the range of 10–15%.

In the present study a CFD model of generic furnace is developed and effect of FGR on the NOx reduction and temperature controlled have bee studied with the help of CFD simulations.

## FLUE GAS RECYCLING IN SAF

Figure 1 shows a conceptual design of a semi-closed silicon production process, where recirculated flue gas is used for temperature control in the combustion zone[1] following the submerged arc furnace. Semi-closed in this context means that the amount of ingress air into the SAF is limited and used to actively control the oxygen concentration in the flue gas at the exit of the combustion zone. The ingress air which in current processes are used for temperature control is in this concept replaced by flue gas taken downstream of the hear recovery steam boiler and filter system. The recirculated flue gas which is available at "flue gas stack temperature", typically around 150 °Cand is used to control the flue gas temperature at the exit of the combustion zone up-stream inlet to the convective part of the heat recovery steam boiler.

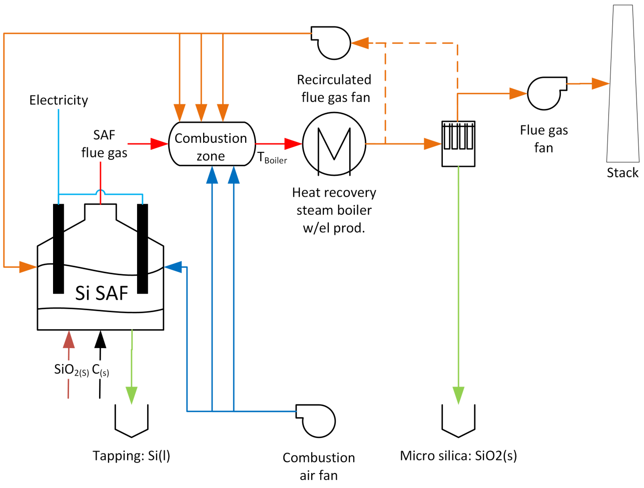


**Figure 1:** Conceptual design of a semi-closed silicon production process, where recirculated flue gas is used for temperature control in the combustion zone following the submerged arc furnace (SAF

A dedicated flue gas combustion zone is indicated in Figure 1. This corresponds to the staged combustion chamber as described by Wittgens et al. (Wittgens *et al.*, 2018) in the SCORE-project. In this study, we have focused only on the effect of replacing ingress air with recirculated flue gas in a conventional SAF design. Base case conditions are defined in Table 1. The combustion of the SAF process gas (leaving the charge surface) is simplified by merging the content of tar components and SiO (g) into CO on a volumetric basis, nevertheless the overall energy balance is correct and combustion is described properly. The SAF process flow rate correspond to a furnace with around 40 MW electric effect.

## DESIGN TRADE-OFFS

The oxygen concentration in the flue gas leaving the combustion zone (and the stack) is the most important design variable in terms of mass and energy balance for a semi-closed silicon process, as outlined in Figure 1. The $O_2$-concentration in the flue gas will have major impact on both the stack-loss and the gross flue gas flow rate through the SAF combustion zone and the heat recovery steam boiler. The stack loss is affected

---

[1] The combustion zone is defined as the void volume above the charge until flue gas is exiting the furnace hood.

**Table 1:** Base case conditions used as basis for CFD simulations.

| Components | Nm3/h | °C |
|---|---|---|
| SAF process gas | 11 500 | 1500 |
| CO=70.3%; $H_2$=6.4%; $H_2O$=23.3%; $N_2$ rest. | | |
| Net flue gas to stack | 40 703 | 150 |
| $O_2$=6.2%; $CO_2$=20%; $H_2O$=9%; $N_2$ rest. | | |
| Total combustion air | 33 583 | 20 |
| Recirculated flue gas from boiler | 94 975 | 150 |
| Gross flue gas to boiler | 135 678 | 745 |

through the amount of "excess" air which is heated up from ambient to stack temperature. The lower the oxygen concentration is (less excess air), the lower the stack loss will be. The gross flue gas flow rate is a result of the amount of recirculated flue gas which is needed to meet the desired flue gas temperature at the entrance of the convective part of the heat recovery steam boiler system (another key design parameter). With less excess air available for cooling in the combustion zone, more recirculated flue gas is needed to meet the desired design temperature.

The oxygen concentration of the recirculated flue gas has a significant effect on local peak temperatures within the combustion zone. Figure 2 shows theoretical flue gas temperature (from a simple energy balance model) versus stoichiometric ratio, $\lambda$ during combustion of the flue gas using mixtures of combustion air and recirculated flue gas. The stoichiometric ratio, $\lambda$ is here defined as the ratio between the amount of $O_2$ added to the SAF flue gas and the amount of $O_2$ needed to completely convert CO and $H_2$ in the SAF flue gas to $CO_2$ and $H_2O$. Thus, $\lambda = 0$ corresponds to the flue gas leaving the SAF charge, $\lambda = 1$ corresponds to completely combusted flue gas (with $O_2$ = 0 vol in the flue gas) and $\lambda = 1.57$ corresponds to the specified base case $O_2$ concentration of 6.2 vol% $O_2$.

**Case 1** shows the theoretical flue gas temperature as a function of $\lambda$ when combustion air is added first to the flue gas leaving the SAF charge. The theoretical flue gas temperature at $\lambda = 1$ is very high - above 2500degC and represents a theoretical peak temperature during combustion of the SAF process gas in air. After combustion to the specified excess air concentration of 6.2 vol% $O_2$ which corresponds to $\lambda = 1.57$ the flue gas temperature is slightly below 2000 °C. Cooling down to the target temperature at 745 °Cis achieved by injection of recirculated flue gas available at 150 °Cdownstream of the heat recovery steam generation system.

**Case 2** shows the corresponding temperature profile if combustion air and recirculated flue gas streams are mixed prior to injection to the SAF combustion zone (which is assumed to start above the SAF charge surface). This mixture of air and flue gas will with the given base case conditions have a concentration of around 10 vol% $O_2$ which leads to significantly lower temperatures. The theoretical peak temperature (at $\lambda = 1$) is below 1600 °C. Although still above the temperature where thermal NOx formation is likely to dominate the extreme temperatures are avoided when compared to Case 1.

**Case 3** shows the temperature profile which can be achieved if the process gas leaving the SAF charge surface is combusted using recirculated flue gas at 6.2 vol% $O_2$. Where the flue gas temperature drops down to below 1200 °Cat $\lambda = 1$ and the specified exit conditions for the SAF charge represents the peak temperature in this case.

The results from this simple model (energy and mass balances) provides the following insights relevant for implementation of a semi-closed silicon furnace:

- Avoid introducing air to flue gas at sub stoichiometric conditions otherwise extreme peak temperatures may occur.
- Recirculated flue gas with sufficiently low oxygen content is an efficient way of avoiding extreme peak temperatures while ensuring complete burnout of the flue gas.
- Sufficiently low oxygen content for recirculated flue gas is in this case most likely below 10vol% $O_2$ if significant NOx formation and operational challenges with high temperatures are to be avoided.

The potential in a conventional SAF furnace is explored further using 3D CFD models in the following chapter.

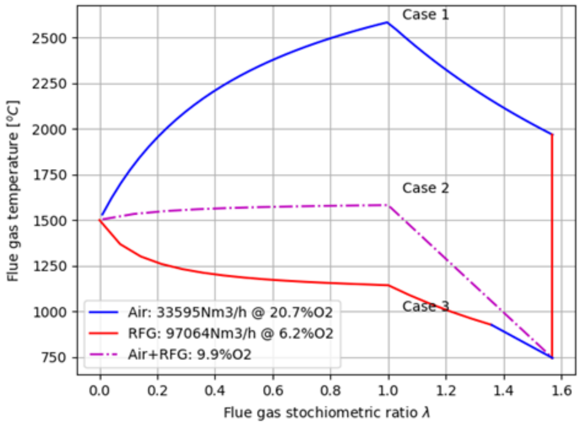


**Figure 2:** Theoretical flue gas temperature versus stoichiometric ratio for base case conditions outlined in Table 1. **Case 1:** Combustion of process gas with combustion air followed by cooling with recirculated flue gas; **Case 2:** Combustion and subsequent cooling of air using a mixture of combustion air and recirculated flue gas; **Case 3:** Combustion of recirculated flue gas and subsequent cooling using combustion air.

## MODELLING

The main objective of CFD simulations was to study the how recirculated flue gases affects the NOx emission during SAF operation. As described in the previous section, when recirculated flue gases are admitted into a combustion system, it primarily reduces the peak temperature which results in NOx reductions. As we know that the NOx formation depends on the local temperature resulting from the reaction between fuel (process gas) and oxygen concentration (air + FGR). In FGR, the oxygen concentration is altered by introducing the additional flue gas with low oxygen concentration and high $CO_2$ concentration. However, there are many questions which need to be answered before implementing the FGR in the real operating SAF

- Which strategy for recirculation of the gas into the furnace hood need to be adapted is not very clear?
- At what location the flue gas should be injected?
- How "burst" will affect the overall NOx formation?

- How radiation losses will affect both the temperature and NOx formation?

To answer these questions, a CFD model of a SAF is constructed. In the present study, the SAF charge surface is modelled as a wall, where flow of process gas is modelled as a mass flux boundary condition. The escaping of process gas through the charge surface is non-uniform and in the real operation a strong burst of SiO and CO have been observed inside the furnace. These bursts are responsible for increased in a local temperature and also increase in a NOx formation. The other objective of this study is to asses the effect of these burst on the NOx formation.

## Geometry

The geometry used for understanding the effect of recirculated flue gases on the NOx formation is shown in Figure 3. This model geometry of furnace hood was developed in two NFR financed projects, namely Promiljø and FUME. The model furnace consists of seven charging pipes, three electrodes, charging surfaces (top surface of the charge bed), simplified single stack, gates, slits and gapes. The charging surface is further divided into three different zones inner, middle and outer.

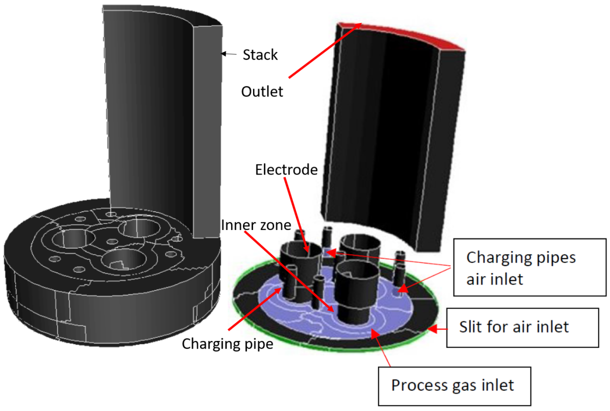


**Figure 3:** Schematic of the model furnace

## Mathematical modeling

ANSYS Fluent was used in the present study. Steady state model solving for continuity, momentum, energy, species balances and radiation equations was used. Flow inside the furnace is turbulent in nature and flow dynamics depend on the initial temperature and turbulence distribution. Fluent solves the Reynolds averaged Navier-Stokes equation (RANS) on the grid scales and turbulence is modeled using an appropriate turbulence model. In the present study we used the RNG k-$\epsilon$ model for turbulence modelling and radiation is modelled with Discrete Ordinates (DO) model. Both the k-$\epsilon$ model for turbulence and DO model for radiation were chosen based on our previous experience on modelling such kind of furnaces. In DO model, the number of theta and phi divisions were set to 5 and theta and phi pixels were set to 3.

## Combustion and kinetics

A variety of combustion models (i.e. Eddy dissipation concept (EDC), Flamelet model, Presumed PDF model) can be used for simulating the combustion related problems with great difference in terms of computational costs, accuracy and complexity. A non-premixed turbulent methane flame was studies by Rohani et al. (Rohani *et al.*, 2012) using EDC model and Presumed PDF model. Their studies revealed that presumed-PDF model shows considerably better agreement with experimental data which is mainly due to implementing a probabilistic approach for relating the turbulence characteristics of the flow field to thermo-chemical properties of the flame. However, the presumed PDF model was computational demanding and very complex and therefore it is very challenging to use this model for simulating the complex geometries such as SAF. Therefore, in the present study, EDC model is applied. The original version of the EDC model was developed by Magnussen (Magnussen and Hjertager, 1977) and after that various version of EDC model have been developed. In the current study the model available in FLUENT have been used in which the chemical reaction rate is governed by the large-eddy mixing time scale and also by the chemical time scale. ANSYS FLUENT provides the finite-rate/eddy-dissipation model, where both the Arrhenius and eddy-dissipation reaction rates are calculated. The net reaction rate is taken as the minimum of these two rates. The EDC model accounts for the chemical kinetics through Perfectly Stirred Reactor (PSR) concept in which each numerical grid is assumed as a stand alone reactor.

Combustion of CO does not take place in a single step, there are many intermediate steps involved in CO combustion. Accounting all the intermediate steps is indeed computationally expensive because many species transport equations need to be solved in CFD framework. Nevertheless, combustion inside the furnace takes place at very high temperatures and therefore a simplified CO mechanism is assumed to be sufficient for our analysis. In the present study, a two-step reaction mechanism for the CO combustion is used. How a detailed kinetic model could affect the NOx formation is presented in our previous publication (Panjwani and Olsen, 2013). The soot formation is not considered in the current paper.

## NOx modeling

In present study it is believed that the main source of NOx production is thermal NOx, therefore only thermal NOx is considered. Formation of NOx depends on the instantaneous temperature, species concentration, and radical concentration. NOx is the common notion for the two gases nitric oxide (NO) and nitrogen dioxide ($NO_2$). For simplification, we only calculate the formation of NO. In combustion system the NO kinetics is very slow, and concentration is generally low, and because of this NO chemistry has negligible influence on the overall flow pattern, temperature field and other species concentration. Therefore, a post processing approach is used for estimation of NO concentration. The post processing approach implemented in FLUENT solves the transport equation for NO concentration with a source term expressed as an Arrhenius rate of law. With post processing-based approach we expect to capture trends based on different furnace hood designs, but we do not expect to find the correct overall emission of NO.

First, the fundamental equations of mass, momentum and energy were solved. Once the chosen convergence criteria were fulfilled, then the post processing tool was used for modeling the NO. The principal reactions governing the formation of thermal NOx from molecular nitrogen are as follows:

$$O + N_2 \Leftrightarrow N + NO \tag{1}$$

The forward and backward coefficient for above reaction were $1.8 \times 10^8 e^{-38370/T}$ and $3.8 \times 10^7 e^{-425/T}$ respectively.

$$N + O_2 \Leftrightarrow O + NO \tag{2}$$

The forward and backward coefficient for above reaction were $1.8 \times 10^4 e^{-4680/T}$ and $3.8 \times 10^3 e^{-20820/T}$ respectively.
A third reaction has been shown to contribute, particularly at near-stoichiometric conditions and in fuel-rich mixtures:

$$N + OH \Leftrightarrow H + NO \tag{3}$$

The forward and backward coefficient for above reaction were $7.1 \times 10^7 e^{-450/T}$ and $1.7 \times 10^8 e^{-24560/T}$ respectively.
The net rate of NOx formation now depends on the above mentioned forward and backward reaction rates and also $N_2$ and $O_2$ concentration

### Boundary conditions and case description

In Figure 3, some of the boundary conditions are illustrated. The inlet of process gases is divided into three zones and most of the gas comes from the center (zone 1) near the electrodes. The zone 1 (inner) gets 60 % of the fuel (process gas), Zone 2 gets 30 % and Zone 3 gets 10 % of the fuel. The fuel consists of 77 % CO and 23 % of $H_2O$ (mass basis). The flow rates, temperature and species concentration at various inlets were specified based on the value provided in Table 1. In total four simulation were carried out and description of these simulations are provided in Table 2 In all the simulations, it was assumed that all the gates of SAF were closed. In Case-1, 25% fresh air ($O_2$=21%, $N_2$=79%) was supplied at charging pipes and 75% fresh air ($O_2$=21%, $N_2$=79%) was supplied through gaps and slits. In Case-2, 75% fresh air was supplied at charging pipes and 25% fresh air was supplied through gaps and slits. In Case-3, 75% recirculated flue gas ($O_2$=6%, $CO_2$=20%, $H_2O$=9%, $N_2$=65%) was supplied at charging pipes and 25% fresh air was supplied through gaps and slits. In Case-4, 25% fresh air was supplied at charging pipes and 75% recirculated flue gas supplied through gaps and slits.

**Table 2:** Simulation description

| Simulation | Flow rate charging pipes (Nm3/h) | Flow rate gaps (Nm3/h) |
|---|---|---|
| Case-1 | Fresh air: 33 583 | Fresh air: 94 953 |
| Case-2 | Fresh air: 94 953 | Fresh air: 33 583 |
| Case-3 | Flue gas: 94 953 | Fresh air: 33 583 |
| Case-4 | Fresh air: 33 583 | Flue gas: 94 953 |

## RESULTS

### Steady state simulations

The above mentioned model has been validated with pilot furnace and the results from this validation has been presented in a previous conference (Panjwani and Olsen, 2013). FGR can be a highly effective technique for lowering NOx emissions and it is relatively inexpensive to apply. The recirculation ratio R, as a key parameter in gases combustion system, was defined as mass ratio between the amounts of recycled and total flue gases, as expressed:

$$R = \frac{MRFG}{MRFG + MTFG} \tag{4}$$

where MRFG is the recycled flue gas mass flow rate and MTFG is the total produced flue gas mass flow. For example, $R = 0$ means that there is no flue gas that was recycled back to the hood, and in extreme case, $R = 1$ represents that total flue gas was recycled back to the hood. The effect of FGR on NOx is shown in Table 3. In Case-1, NOx is around 150 g/s but by replacing the fresh air with flue gases on the slits and other opening(Case-4), the NOx was reduced to 17 g/s. Similarly, in Case-2 and Case-3, when fresh air through charging surface was replaced with flue gas NOx was reduced from 90 g/s to 10 g/s.

**Table 3:** CFD simulation results

| Simulation description | NOx [g/s] | $T_{out}$ degC | $O_2$ | R |
|---|---|---|---|---|
| Case-1 (without FGR) | 150 | 640 | 16.5% | 0 |
| Case-2 (without FGR) | 90 | 560 | 17% | 0 |
| Case-3 (with FGR ) | 10 | 640 | 6.2% | 0.75 |
| Case-4 (with FGR ) | 17 | 720 | 5.8% | 0.75 |

Thermal NOx is produced by the reaction of atmospheric oxygen and nitrogen at elevated temperatures, and is considered to be the dominant mechanism. ISO clip temperature of the furnace for Case-1 (without FGR) and Case-4 (with FGR through the gaps) are shown in Figure 4, and 5 respectively to understand the effect of FGR on temperature.

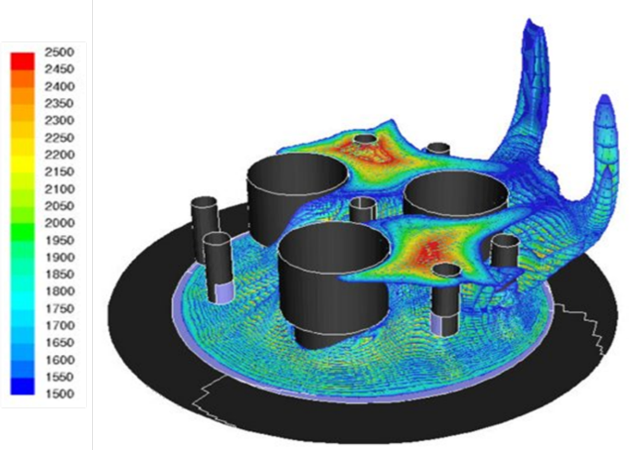


**Figure 4:** Case-1: Iso-clip of temperature °C

High temperature pockets responsible for thermal NOx formation are clearly visible in CASE-1 (see Figure 4). The number of hot pockets has been reduced with FGR (see Figure 5).

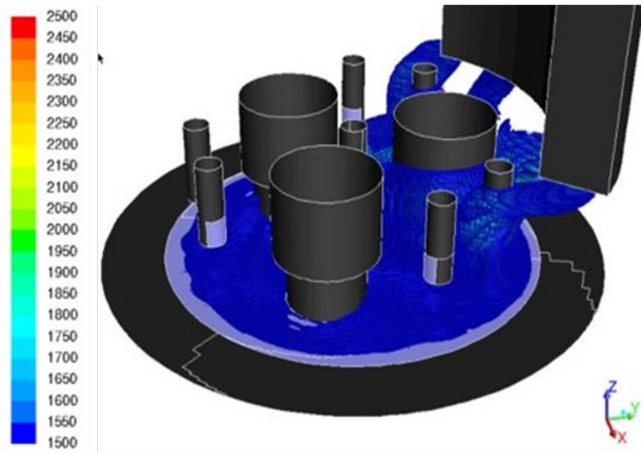


**Figure 5:** Case-4: Iso-clip of temperature °C

The effect of temperature on the NOx is exponential and small reduction in temperature causes significant reduction in NOx formation and thus concentration of NOx in the furnace off-gas. The amount of thermal NOx formed not only depends on the temperature of the flame but also the residence time. Both higher temperature and the longer residence time result in more NOx production. However these simulations have been performed assuming steady state conditions and therefore the effect of residence time is not shown in the current calculations. The prediction of temperature in a turbulent flame is important because the temperature affects the chemical kinetics considerably (i.e. reaction rates of all intermediate reactions) and therefore the combustion behavior. If the temperature is poorly predicted a realistic estimation of the pollution (i.e soot, NOx) is less likely to be achieved.

## Effect of radiation

Due to the high temperatures involved in combustion processes, e.g. 2000 °C, radiation heat transfer appears as an important heat transfer mechanism in many combustion devices including combustion inside the furnace hood. Infrared (IR)-active species such as $CO_2$, CO and $H_2O$ are often present in the products of combustion of fuel consisting of Carbon and Hydrogen. The combustion products such as $CO_2$, CO and $H_2O$ are responsible for the non-luminous radiation.

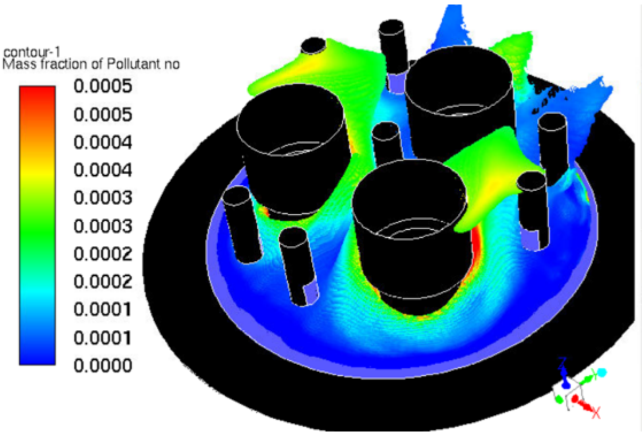


**Figure 6:** Iso-clip of NOx without Radiation

In fuel-rich flames, soot is also formed and, as a consequence, a significant amount of radiation is emitted in the luminous region of the spectrum (Pessoa-Filho, 1999). In industrial furnaces, there are many hot particles such as dust and soot and because of these hot particles a significant amount of radiation is emitted in the luminous region of the spectrum. Many studies have indicated that radiation is found to reduce the flame temperature with a consequent reduction of the overall reduction in NOx concentration. Radiation is the dominant mechanism of energy transfer in combustion systems like furnaces, turbines, engines, combustion chambers, etc. Many studies have shown that heat transfer by radiation should be considered in the CFD combustion simulation.

The radiation effects proved to have a great influence on the velocity and temperature profiles. Although in many combustion calculation effect of radiation is neglected assuming combustion gases as a transparent. However, the combustion gas inside the furnace contain $H_2O$ and $CO_2$ and both these gas components are not transparent to the radiation. The effectiveness of $CO_2$ addition in reducing flame temperature and NOx emission have been studied widely and $CO_2$ primarily increase the heat capacity and radiation loss (Park *et al.*, 2008).

The previous simulations as given in Table 3 were carried out with radiation model but it was assumed that gas is transparent to the radiation. This had resulted in higher temperature and therefore higher NOx formation. These assumptions were modified and additional simulations with and without radiation were performed. WSGGM-domain-based model was used for estimating the absorption coefficient. Although scattering also affects the radiation intensity along the path for purposes of simplicity, here scattering is neglected. The simulations results with and without radiation are shown in Table 4. The NOx formation rate without radiation is 90 (g/s) and with radiation the rate of NOx formation is 2.44 (g/s). The results from this study without radiation and with radiation are shown in Figure 6 and Figure 7. The figures 6 and Figure 7 show the iso-clip of the local NOx concentration without and with radiation. A peak in the NOx corresponding to the high temperature zone are quite visible. In absence of the radiation, the heat losses to the surrounding is extremely small which results in higher temperature inside the furnace and therefore many pockets with a larger concentration of NOx (as shown in Figure 6) are observed. However, the temperature became much lower when simulations were repeated with radiation and therefore the larger NOx concentration disappear see Figure 7.

**Table 4:** The effect of radiation model on NOx formation

| Case | Charge pipes (Nm3/h) | Gaps (Nm3/h) | Radiation | NOx (g/s) |
|---|---|---|---|---|
| Case-2 | 94 953 (F)* | 33 583 (F)* | NO | 90 |
| Case-2A | 94 953 (F)* | 33 583 (F)* | YES | 2.44 |

## Effect of gas burst

One of the challenging issue with the submerged arc furnace used in the silicon and high silicon alloy industries is the existence of high gas pressure condition inside the crater zone of the furnace. Crater zone is formed as cavity in the bulk of charge materials around the electrodes tip. In fact due to the existence of chemical reactions inside the charge materials, there is always a high gas pressure situation in the furnace heart. Because of reduced permeability of charge materials and melting of charge materials in the region near by the electrode tips (the crater walls), the gas pressure in the crater zone increases to higher levels than what can be expected from a porous bed. Furthermore, a submerged-arc

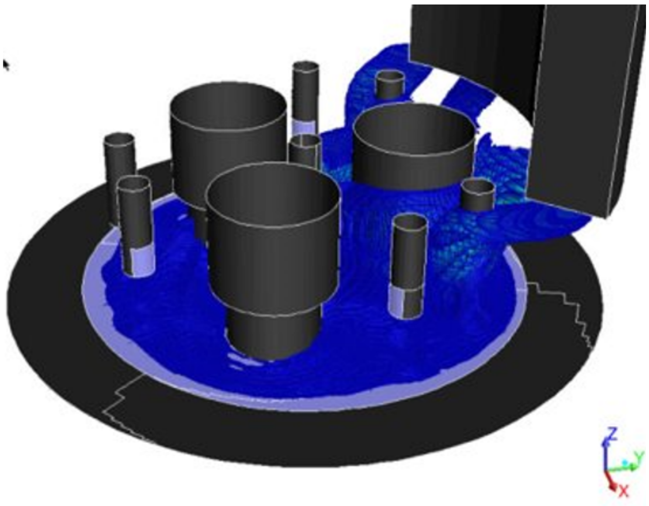


**Figure 7:** Iso-clip of NOx with Radiation (scales are given in Figure 6)

**Table 5:** Base case conditions used as basis for CFD simulations.

| Case | Charge pipes (Nm3/h) | Gaps (Nm3/h) | burst | NOx (g/s) |
|---|---|---|---|---|
| Case-2A | 94 953 (F)* | 33 583 (F)* | NO | 2.44 |
| Case-5 | 94 953 (F)* | 33 583 (F)* | Inner | 2,73 |
| Case-6 | 94 953 (F)* | 33 583 (F)* | Outer | 2.94 |
| Case-7 | 94 953 (FGR)* | 33 583 (F)* | Inner | 0.026 |
| Case-8 | 94 953 (FGR)* | 33 583 (F)* | Outer | 0.286 |

furnace is continually fed with carbon and quart to produce liquid metal which is tapped from the base of the furnace). To maintain the continuous production of liquid metal, the raw material such as ore and carbon sources (coke, coal, and wood chips) are injected in an regular intervals. During the reduction process, gas cavities are formed underneath the charge surface and a solid crust region builds up above a gas cavity. This crust is composed of a mixture of carbon, molten quartz, silicon carbide, and condensate. (Sloman *et al.*, 2017). During the furnace operation, the cavity pressure becomes so high that gas escapes rapidly to the charge surface and the rapid release of the gas are known as a burst (Schei *et al.*, 1998; Kadkhodabeigi *et al.*, 2010).

Burst of high velocity hot gases from the furnace charging is one of phenomena which seems to be related to the existence of the high pressure crater zone of the furnace. A 2D view of the inside of the silicon furnace is presented in Figure 8. Results of furnace excavations confirm the formation of the cavities around the electrodes tips. Normally these burst consist of the process gases with a high concentration of SiO and CO gas. These gases reacts with air and produces high temperature zone responsible for NOx formations. The proposed CFD model of the furnace with burst allows better understanding of the burst effects on the NOx formation.

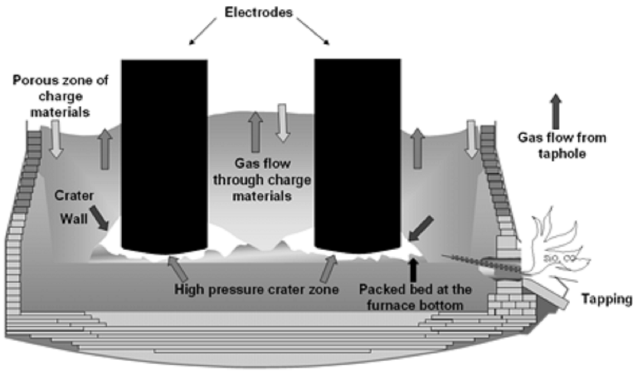


**Figure 8:** Schematic of the inside conditions of a submerged arc furnace used for Silicon production (Kadkhodabeigi *et al.*, 2010)

In the present study, CFD simulations of the submerged arc furnace with and without burst were performed. The simulations and corresponding results for burst simulations are given in Table 5. In Table 5, (F)* Means fresh air and (FGR)* Means Flue gas recirculation

In total, four simulations were performed to establish the effect of burst and FGR on the temperature and NOx distribution. The burst results in a higher rate of process gas formation and the combustion of additional process gases from the burst results in an increased temperature and therefore increase in thermal NOx formation. The process gas in silicon furnace consist of both CO and SiO and the combustion of SiO results in micro-silica ($SiO_2$) formation. The heat of formation of $SiO_2$ is approximately three times higher than $CO_2$. In practice, the burst results in more SiO formation and this SiO will react with air resulting in an increase flame temperature. However, the SiO reaction is not considered in the present calculations instead equivalent CO reactions are considered. Again to model the burst, mass flow rate boundary condition was used at the charge surface. For burst, the process gas mass flow rate was provided as an input. Case-5 is a baseline case where the fresh air is injected through charging pipes and also through other openings. The process gas was released from the charging surfaces and burst at inner location of charging surface was considered. The Case-6 is similar to the Case-5 except the burst was considered at the outer location of the charging surface. In case-7, flue gas was injected equally through the seven charging pipes and fresh air was injected through other openings. The burst was considered at the inner location of the charging surface. The Case-8 is similar to the Case-6 but with FGR through charging pipes. The effect of burst on the NOx can be seen in Table 5. The NOx without burst is 2.44 g/s (Case-2A) and with burst at outer location (Case-6) is 2.94 g/s. With FGR and with burst, the NOx production rate was decreased from 2.93 g/s (Case-6) to 0.286 g/s (Case-8). The Iso-clip of Case-6 and Case-8 are shown in Figures 9 and 10.

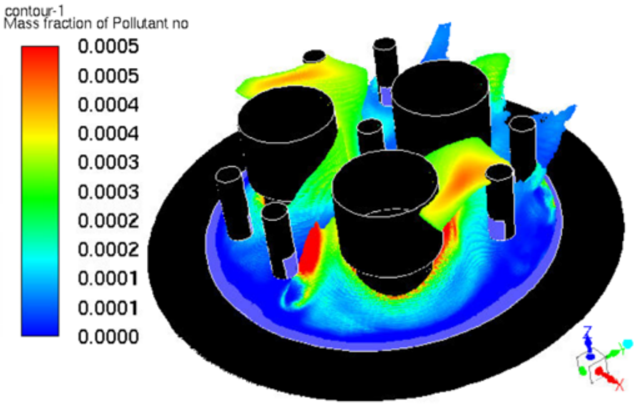


**Figure 9:** Iso-Clip of NOx concentration with radiation, without FGR, and with burst

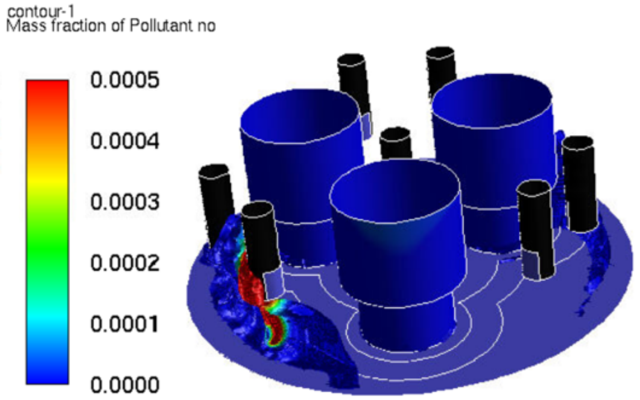


**Figure 10:** Iso-Clip of NOx concentration with radiation, with FGR, and with burst

A localized pocket of NOx at the burst location can be seen that indicates the localized high temperature zone resulting from the reaction between burst gases and ambient air. However, with FGR, with a reduced amount of fresh air the rate of NOx formation is also reduced. The location of burst also affects the NOx formation, the rate of NOx formation is higher when burst occurs at the outer location of the charge surface. Therefore, the effect of location of burst has a prominent effect on the rate of NOx formation with FGR.

## CONCLUSIONS

It is demonstrated how the use of recirculated flue gas in the flue gas combustion zone of a ferro-silicon furnace affects the peak temperatures. The reduction in peak temperature is achieved because recirculated flue gas with reduced oxygen concentration has a significantly lower adiabatic flame temperature than air. The corresponding effect on NOx formation has been demonstrated through computational fluid dynamics simulations. The results indicate an order of magnitude reduction in NOx formation when recirculated flue gas ( 6vol% $O_2$) is used in the combustion zone instead of air (21vol% $O_2$). The recirculation ratio ($R$) has significant influence on both peak temperature and NOx rate. Furthermore, the studies also indicated that the accounting of radiation model is extremely important for modeling the combustion related problems. The studies show that both the burst and its location play a significant role in the NOx production.

## ACKNOWLEDGMENT

The research work carried out in this article is funded by FME HighEff supported by Norwegian research council and many industrial partners. Funding from the FME HighEff project is greatly acknowledged

## REFERENCES

CHEN, Q., ZHANG, X., ZHOU, J., SHARIFI, V.N. and SWITHENBANK, J. (2015). "Effects of Flue Gas Recirculation on Emissions from a Small Scale Wood Chip Fired Boiler". *Energy Procedia*, **66**, 65–68.

DARMANA, D., OLSEN, J.E., TANG, K. and RINGDALEN, E. (2012). "Modelling concept for submerged arc furnaces".

GUETHE, F. and BURDET, A. (2009). "Flue gas recirculation in gas turbine: Investigation of combustion reactivity and nox emission". *ASME Turbo Expo*, **2**, 179–191.

KADKHODABEIGI, M., TVEIT, H. and JOHANSEN, S.T. (2010). "CFD modelling of the effect of furnace crater pressure on the melt and gas flows in the submerged arc furnaces used for silicon production". *Progress in Computational Fluid Dynamics, An International Journal*, **10(5/6)**, 374.

KAMFJORD, N.E. (2012). *Mass and Energy Balances of the Silicon Process:-Improved Emission Standards*. Ph.D. thesis, Noregs teknisk-naturvitskaplege universitet.

LIUZZO, G., VERDONE, N. and BRAVI, M. (2007). "The benefits of flue gas recirculation in waste incineration". *Waste Management*, **27(1)**, 106–116.

MAGNUSSEN, B. and HJERTAGER, B. (1977). "On mathematical modeling of turbulent combustion with special emphasis on soot formation and combustion". *Symposium (International) on Combustion*, **16(1)**, 719–729.

PANJWANI, B. and OLSEN, J.E. (2013). "Combustion and mechanisms for NOx formation in ferrosilicon electric arc furnaces". *European combustion meeting, ECM 2013*, 7.

PARK, J., BAE, D.S., CHA, M.S., YUN, J.H., KEEL, S.I., Chang Cho, H., KIM, T.K. and HA, J.S. (2008). "Flame characteristics in h2/co synthetic gas diffusion flames diluted with co2: Effects of radiative heat loss and mixture composition". *International Journal of Hydrogen Energy*, **33(23)**, 7256 – 7264.

PESSOA-FILHO, J.A.B. (1999). "Thermal radiation in combustion systems". *Journal of the Brazilian Society of Mechanical Sciences*, **21**, 537 – 547.

PETTERSEN, T., WITTGENS, B. and BERGLIHN, O.T. (2017). "Technical and economical evaluations of selected concepts: SCORE". Tech. Rep. D4.1._2017.04, SINTEF, Trondheim Norway.

ROHANI, B., WAHID, M.A., SIES, M.M. and SAQR, K.M. (2012). "Comparison of eddy dissipation model and presumed probability density function model for temperature prediction in a non-premixed turbulent methane flame". 384–391. Melaka, Malaysia.

SCHEEPERS, E., ADEMA, A., YANG, Y. and REUTER, M. (2006a). "The development of a cfd model of a submerged arc furnace for phosphorus production". *Minerals Engineering*, **19(10)**, 1115 – 1125. Selected papers from Computational Modelling â€™05, Cape Town, South Africa.

SCHEEPERS, E., YANG, Y., REUTER, M. and ADEMA, A. (2006b). "A dynamic-cfd hybrid model of a submerged arc furnace for phosphorus production". *Minerals Engineering*, **19(3)**, 309 – 317. Selected papers from Pyrometallurgy â€™05, Cape Town, South Africa, March 2005.

SCHEI, A., TUSET, J. and TVEIT, H. (1998). *Production of High Silicon Alloys*. 1st ed. TAPIR Forlag, Trondheim, Norway.

SLOMAN, B.M., PLEASE, C.P., VAN GORDER, R.A., VALDERHAUG, A.M., BIRKELAND, R.G. and WEGGE, H. (2017). "A Heat and Mass Transfer Model of a Silicon Pilot Furnace". *Metallurgical and Materials Transactions B*, **48(5)**, 2664–2676.

TSILIYANNIS, C.A. (2013). "Flue Gas Recirculation and Enhanced Performance of Waste Incinerators under Waste Uncertainty". *Environmental Science & Technology*, **47(14)**, 8051–8061.

WITTGENS, B., PANJWANI, B., PETTERSEN, T., JENSEN, R., RAVARY, B. and HJERTENES, D.O. (2018). "SCORE: Staged Combustion for Energy Recovery in Ferroalloy Industries â€“ Experimental Validation". *Infacon XV: International Ferro-Alloys Congress*.