ISSN 1611-616X VULKAN-VERLAG · ESSEN
2 2007
HEAT PROCESSING
INTERNATIONAL MAGAZINE FOR INDUSTRIAL FURNACES · HEAT TREATMENT PLANTS · EQUIPMENT
http://www.heatprocessing-online.com

# CFD-simulation of melting furnaces for secondary aluminum

Dr.-Ing. Stefan Murza, Ingenieurbüro für technische und wissenschaftliche Berechnungen, Bochum, Germany
Dr.-Ing. Björn Henning, Dipl.-Ing. Hans-Dieter Jasper, Jasper GmbH, Germany

**Published in HEAT PROCESSING 2/2007**
Vulkan-Verlag GmbH, Essen (Germany)
Editor: Dipl.-Ing. Stephan Schalm, Tel. +49 (0) 201/82002-12, E-Mail: s.schalm@vulkan-verlag.de

# CFD-simulation of melting furnaces for secondary aluminum

Stefan Murza, Björn Henning, Hans-Dieter Jasper

Hall 4 / A55

*The optimal utilisation of fuel per product quantity in combination with a reduction of investment costs has always been a permanent challenge for every manufacturer of furnaces and burners for industrial use – and this not only owing to the current increase in energy cost. The dimensioning and the design of melting furnaces depend on the one hand on the type of scrap metals and on the other hand on the specification of the substance to be produced (e.g. casting or forgeable alloys for the aluminium industry). Today mathematical tools are used in the further development and optimisation of melting furnaces. CFD (computational fluid dynamics) simulation is a tool which has been successfully established in many diverse sectors of industry. It is the intention of this article to present the simulation of a melting furnace which has been done using the commercial software FLUENT®. Numerical simulation shall provide the plant constructor with information on the guiding of flows, temperature distribution, heat input and heat losses in a furnace.*

One of the major challenges of industry is to increase production at a simultaneous reduction of energy use and environmental protection in combination with reduced investment and operation costs. In parallel the demand for increased availability at reduced maintenance is growing – following continuous rationalisation activities. Measures are tightened by codes on environmental protection and more restrictive requirements on the emission of contaminants. These challenges and the quality requirements mentioned afore call for considerable research and development efforts and a high readiness for innovation from plant constructors.

State-of-the-art melting furnaces – respectively heat treatment plants – require well adjusted burner systems. Highest energy savings can be achieved with regenerative heat recovery. The selection of a suitable (regenerative) burner system is based on a number of criteria which have to be tailored to the respective process technology. An extremely interesting aspect is related to the fact that state-of-the-art regenerator systems are not only able to achieve considerable economic advantages (energy savings), but also to accomplish considerable improvements in process technology. There is proven evidence from the field of the aluminium industry which demonstrate that the melting loss of aluminium is reduced. A combination of regenerators and an alteration of the furnace technology have been proven to be very successful also during reconstruction of furnaces in the steel industry. Modifications of rotary heard furnaces resulted in increases in production of up to 33 % at a simultaneous reduction of energy consumption of up to 20 % [1, 2].

Firm insight resulting from experiment as well as the utilisation of mathematical computing models is inevitable to meet the requirements for further development and optimisation of melting furnaces and heat treatment plants. Elementary calculations reflecting the actual condition of the plant can be performed on the basis of experimental data. Simulations of different variations provide information on temperature profiles, flow patterns and radiative and convective heat input. The actual plant condition of a melting furnace for aluminium shall be described in the following.

## Numerical simulation with CFD

Computational fluid dynamics (CFD) has become an important part in modern product development. Due to the fact that most natural flows (e.g. air, water) cannot be seen directly people can hardly relate to them. Using computational fluid dynamics we are able to visualise and analyse flows and the related physical processes like turbulence or heat distributions. CFD acts as an eye in the flow and, so to speak, provide the engineer with an additional 'sense'. They can be used to interpret technical flows already during design processes and also for product optimisation. CFD analyses consist of three main parts. The precise definition of the tasks combined with the module selection (pre-processing), the actual calculation (processing), and the analysis of the results including data visualisation (post-processing). Unfortunately, in the review of CFD projects it sometimes becomes evident that insufficient attention was paid to the pre-processing with regard to the potential results and their significance. The time required for the second work step, the actual calculation time for the processing, depends on the computing power of the system which is available for the simulation. Using actual CFD tools most analyses can be solved on parallel computers or computer cluster systems. The performance of modern computers has grown dramatically, but this does not automatically result in shorter computation time in that growing computational power typically leads to the application of more complex models. Additionally, CFD engineers use finer grids to describe the considered domain. Ten years ago a massive parallel computer had to be used in order to solve a simulation of reacting flows with one million grid points in acceptable time. Now, using the same models and discretisation the simulation can be done on a normal personal computer in approximately the same time. Doing

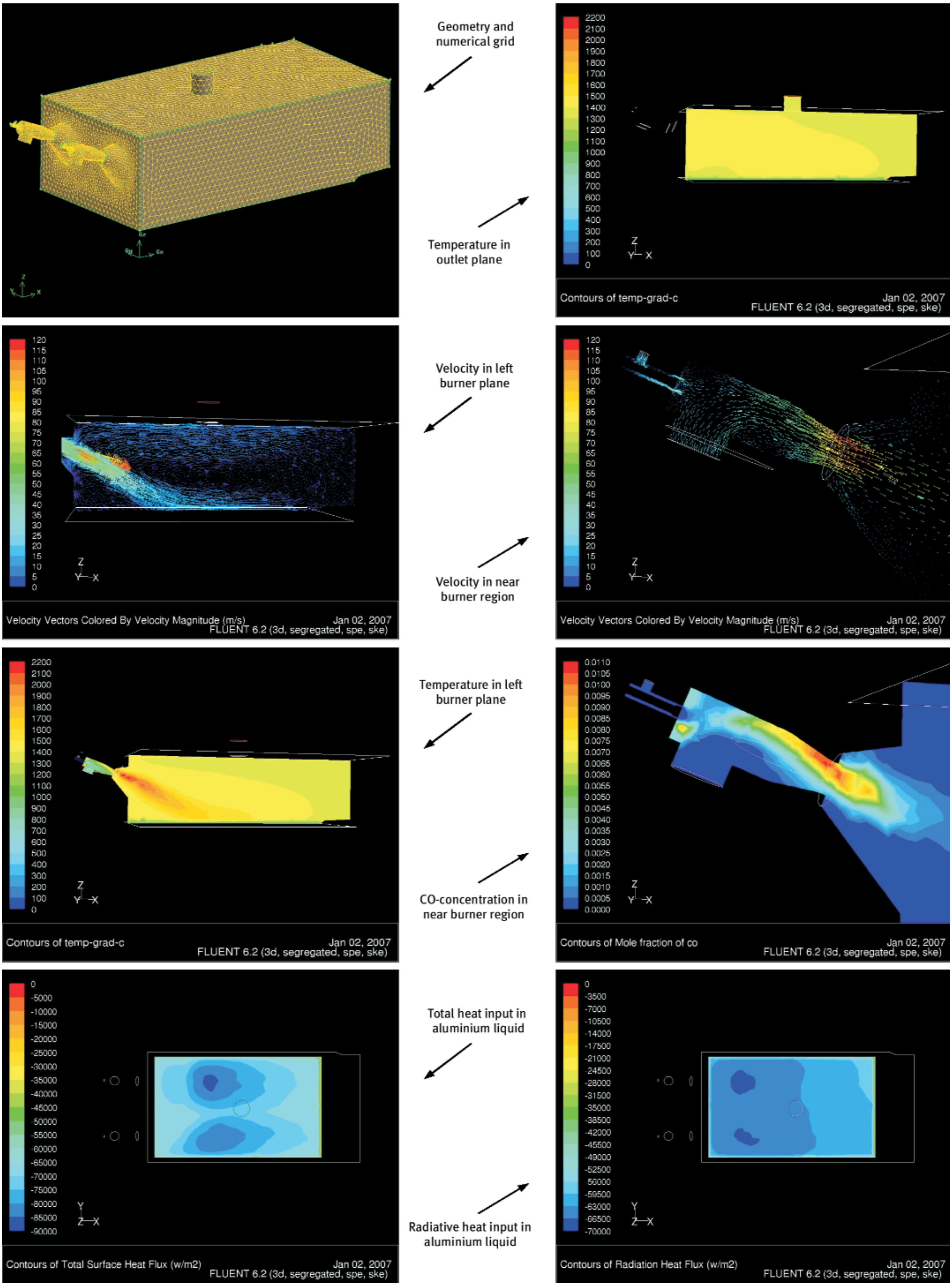


**Fig. 1:** Numerical simulation of an aluminium melting furnace

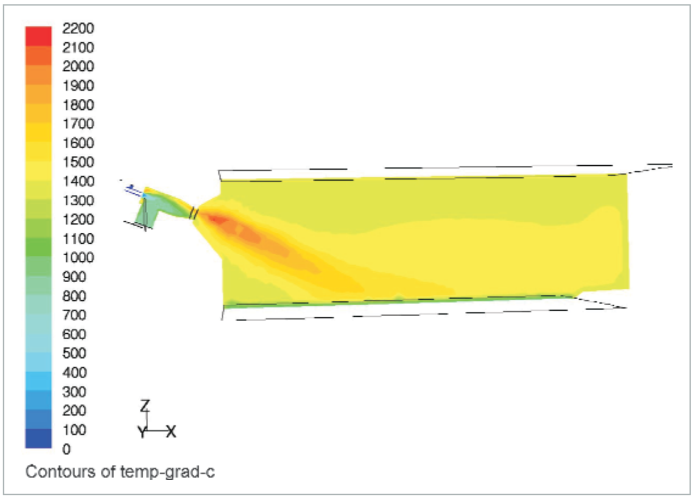


**Fig. 2:** Temperature distribution in an axial plane (high impulse)

post-processing, the third work step, the CFD engineer should keep in mind that it is sometimes better to focus the results to the answer to the original question, instead of showing any possible graphical representation.

The presented simulation of an aluminium melting furnace was done by using the CFD program FLUENT® in version 6.2. The analysis focussed on the influence of different geometries and the fuelling parameters effecting the efficiency and the output of melting furnaces for secondary aluminium. Due to the fact that multiple varying simulations had to be made – amongst others for the definition and validation of boundary conditions – the calculations could not be solved on a very fine grid. Otherwise the computational effort would have gone far beyond the scope of project time and cost. There was no explicit need for highest academic accuracy of single results. The focus was rather put on a couple of relevant parameters which are of practical importance for the furnace design and operation, in order to help the manufacturer to continually optimise his product.

The entire calculation domain was set up completely three dimensional with unstructured grids, using the program GAMBIT® in version 2.3. The domains had a size of 110 to 160 thousand volume elements. The standard k-ε-model was used to describe the effect of turbulence onto the flow field. For the description of the gas phase combustion a model was used which solved the transport phenomena of every relevant gas phase species. To take into account the effects of turbulence on the chemical conversion rates an eddy-break-up model was used. The radiative heat transfer in the melting furnace was solved by using the 'discrete-transfer' model. For this analysis it was particularly important to use common models with standard parameters to assure comparability and traceability of the calculation results. Due to the fact that heat transfer in melting furnaces is mainly dominated by radiation it was very important to determine the most accurate emission/absorption coefficient for the surface of the liquid aluminium. This was done in numerous pre-calculations. Based on empirical data obtained from literature a coefficient was determined which was used for all subsequent simulations. To ensure comparability and traceability this was the only adjustment of the standard models and their parameters.

Different geometrical variations were calculated. For each of them a new computational grid had to be generated. Varying arrangements of the exhaust gas exit were investigated as well as geometric modifications of the burners used and their arrangement. In addition domains with different shapes of the melting furnace covers were simulated.

A further analysis discussed the effects of different air flap settings in the feed pipe of the burners with special emphasis on the asymmetric momentum and its consequence on the heat input into the liquid aluminium.

## Results

**Fig. 1** shows some selected figures of the computation results of one version of an aluminium melting furnace with a straight cover of the furnace cover and a centred exhaust gas exit in the middle of the roof. The computational domain with the numerical grid is shown in upper left figure. Temperature and velocity drawings show a distinctive flow nearby the aluminium surface. As intended by the manufacturer the flow is cooled down and ascends in the rear section of the furnace where it is redirected towards the exhaust gas exit. This steady flow through the furnace generates a good convective heat transfer which has an amount of 17 percent of the total heat transfer into the liquid aluminium. The radiative heat transfer - shown in the right lower figure of picture 1 – dominates in the first third of the furnace where the regions with high flame temperatures are located. This distribution of heat input caused by radiation is beneficial for the liquid aluminium which is in counter flow with the gas phase.

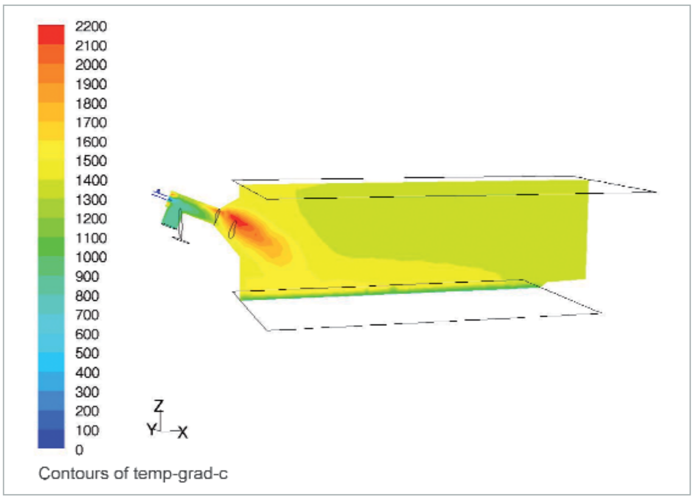


**Fig. 3:** Temperature distribution in an axial plane (low impulse)

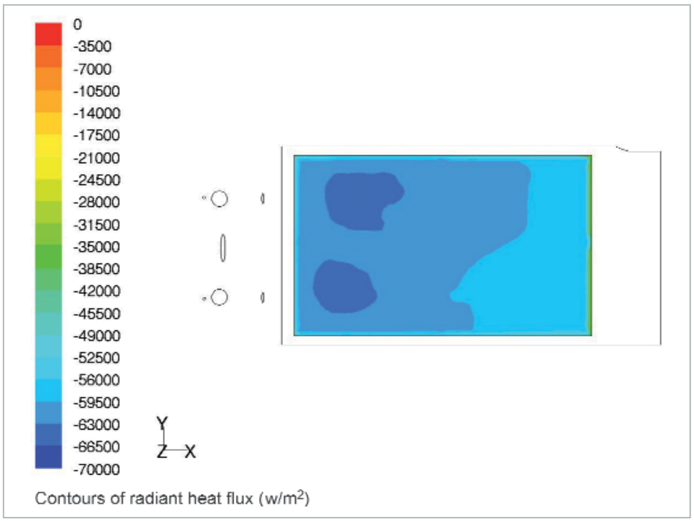


**Fig. 4:** Radiative heat transfer into the aluminium bath (high impulse)

**Fig. 2** and **Fig. 3** show a comparison of the temperature distribution along a burner axis for two variant simulations of the melting furnace to show the effect of different exit momentum caused by geometrical alterations at the burners. In combination with the **Fig. 4** and **Fig. 5** for the same variant calculation a significant increase in radiative heat transfer can be observed. This is shown by the white areas in the figures caused by a heat transfer in a magnitude which is beyond the colour-map. For comparison purpose the colour-map was set consistently for all variant calculations. The low momentum leads to shorter flames with a higher energy density, which is responsible for the very high radiative heat input in small regions. Looking at the results of the calculation it can be seen that this variant with low momentum is inappropriate. In addition to the higher NOx emissions a lower total heat input into the aluminium and an unbalanced temperature distribution in the liquid can be demonstrated.

## Conclusion

This article describes a CFD-Simulation of a melting furnace for secondary aluminium with the commercial avail-able tool FLUENT®.

Looking at the calculated melting capacities, a difference up to five percent between the variant simulations provoked by geometrical modifications can be demonstrated. In particular, variants with low flow impulse at the burner exit lead to poorer melting capacities caused by a significant lower convective heat transfer into the liquid aluminium. A comparison of the results for the simulations of melting furnaces with different cover shapes and the same energy input show an up to two percent higher melting output achieved by a convex cover shape. It can be seen that the increase in melting capacity is mainly caused by the modified flow field and the adjunctive convective heat transfer. A similar effect can be found for variants with displaced exhaust gas exits which also lead to a modified flow field inside the melting furnace. A comparison of the variants demonstrates that even minor alterations – for example a different setting of the air flap position – have a non-negligible effect on the heat input and the melting capacity.

### Literature

[1] Henning, B.; Jasper, R.: MultiMelter© - The new generation of aluminium melting furnaces, Heat Processing 2/2004, Vulkan-Verlag

[2] Henning, B.; Klatecki, P.; Jasper, R.: Optimierte Regeneratortechnologie an industriellen Feuerungssystemen, Gaswärme-International 6/2005, Vulkan-Verlag

[3] von Starck, A.; Mühlbauer, A.; Kramer, C.: Praxishandbuch Thermoprozesstechnik, Vulkan-Verlag 2003 ■

**Dr.-Ing. Stefan Murza**

Ingenieurbüro für technische und wissenschaftliche Berechnungen, Bochum Germany

Tel. +49 (0)234 / 3387996
stefan.murza@murza.net

**Dr.-Ing. Björn Henning**

Jasper GmbH
Germany

Tel. +49 (0)4106 / 8043-0
b.henning@jasper-gmbh.de

**Dipl.-Ing. Hans-Dieter Jasper**

Jasper GmbH
Germany

Tel. +49 (0)4106 / 8043-0
hd.jasper@jasper-gmbh.de

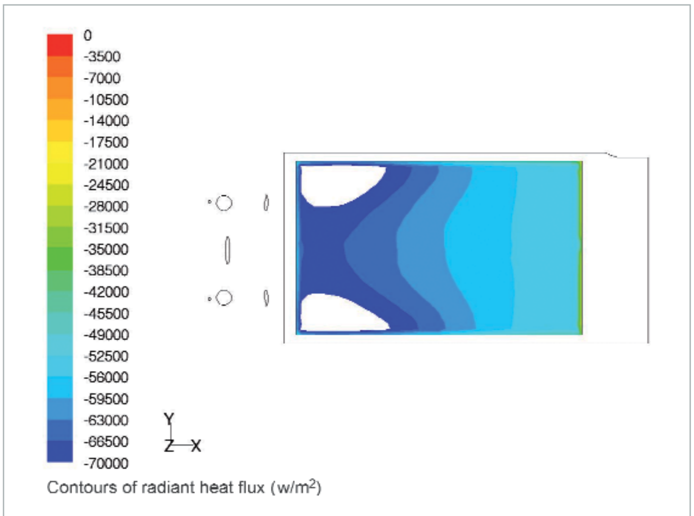


**Fig. 5:** Radiative heat transfer into the aluminium bath (low impulse)