

## CFD-Simulation of Siphone For primary aluminum production

Mohsen ameri siahooei<sup>1</sup>, Darioush hossini roozbehani<sup>\*2</sup>,

<sup>1,2</sup> Almahdi-Hormozal Aluminium Smelter, Bandar abass, P.O. Box:79171-7-6385, Iran

**Abstract:** Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

At the time of launching the Almahdi aluminum production line, the utility group faced many problems with metal depletion siphon for evacuating metal from pots, that these problems were mainly due to the lack of adequate suction of the siphon during the tapping.

In that time firstly, joints of the syphon door and the crucibles overflow were empirically sealed by changing the insulation type, and the size of the bolts and nuts of these joint was changed to use a power wrench for tightening the joint so that they can be sealed completely. However, one of the most important parts involved in the rate of the three way suction was the metal depletion siphon door, which was made of a three-way shape chamber, and two inlet and outlet nozzles. With spending large time and experimentally these parts were ultimately calibrated determined for the above mentioned purpose and their distance from each other in the three way was determined exactly. Then, the tapping operation was carried out in the normal way. Today mathematical tools are used in the further development and optimisation of crocyble. 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 siphone 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 siphone.

**Keywords):** “CFD-Simulation; primary aluminum production”

### Introduction

At the time of launching the Almahdi aluminum production line, the utility group faced many problems with metal depletion syphon for evacuating metal from pots, that these problems were mainly due to the lack of adequate suction of the syphon during the tapping.

In that time firstly, joints of the syphon door and the crucibles overflow were empirically sealed by changing the insulation type, and the size of the bolts and nuts of these joint was changed to use a power wrench for tightening the joint so that they can be sealed completely. However, one of the most important parts involved in the rate of the three way suction was the metal depletion syphon door, which was made of a three-way shape chamber, and two inlet and outlet nozzles. With spending large time and experimentally these parts were ultimately calibrated determined for the above mentioned purpose and their distance from each other in the three way was determined exactly. Then, the tapping operation was carried out in the normal way.

Now, considering improvement of the knowledge and the experience of the employees in this complex, solving such problems can be done by simulating with relevant software or using mathematical and analytical methods. Therefore, in this research we have tried to simulate the fluid behavior in the metal depletion three-way to increase the technical knowledge for new designs at a low cost and less time.

The following photo shows a sample of the syphon and crucible set, which is ready to drain the metal from the pot, a three-way is shown on top of it by a black arrow.

Gambit software was used for modeling because this software is very powerful and efficient in terms of meshing the piece for further analysis in the Flaunt software.

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.

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 Crucibles and siphons for aluminium shall be described in the following.

Crucibles and siphons used in aluminium smelters must be cleaned and maintained to optimize metal transportation from the potroom to the casthouse. As an increase in cell amperage leads to higher potroom production and metal transportation, this cleaning step becomes more crucial. Crucible cleaning involves the removal of metal/bath deposits and must be followed by crucible preheating. Siphon and tapping tube cleaning are also very important in the process.



Figure 1 -Crucibles and siphons

## Methodology in Gambit Modeling

This tutorial illustrates geometry creation and mesh generation for a simple geometry using GAMBIT.

- Start GAMBIT
- Use the Operation toolpad
- Create a brick and an elliptical cylinder
- Unite two volumes
- Manipulate the display of your model
- Mesh a volume
- Examine the quality of the mesh
- Save the session and exit GAMBIT

This first tutorial illustrates some of the basic operations for generating a mesh using GAMBIT. In particular, it demonstrates:

- How to build the geometry easily using the “top-down” solid modeling approach
- How to create a hexahedral mesh automatically

The “top-down” approach means that you will construct the geometry by creating volumes (bricks, cylinders, etc.) and then manipulating them through Boolean operations (unite, subtract, etc.). In this way, you can quickly build complicated shapes without first creating the underlying vertices, edges, and faces.

Once you have built a valid geometry model, you can directly and (in many cases) automatically create the mesh. In this example, the Cooper meshing algorithm is used to automatically create an unstructured, hexahedral mesh. More complicated geometries may require some manual decomposition before you can create the mesh; this is demonstrated in subsequent tutorials.

The steps we would follow in this tutorial are listed below:

- Create two volumes (a brick and an elliptical cylinder).
- Unite the two volumes.
- Automatically generate the mesh.

- Examine the quality of the resulting mesh.

To keep this introductory tutorial short and simple, certain steps that we would normally follow have been omitted:

- Adjusting the distribution of nodes on individual edges of the geometry
- Setting continuum types (for example, identifying which mesh zones are fluid and which are solid) and boundary types

These details, as well as others, are covered in subsequent tutorials.

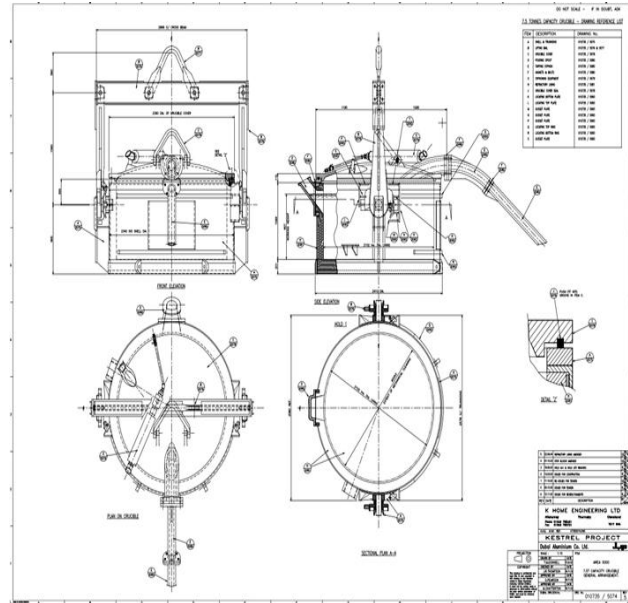


Figure 3- Crucibles and siphons maps

The model consists of an intersecting brick and elliptical cylinder. The basic geometry is shown schematically in Figure 2,3

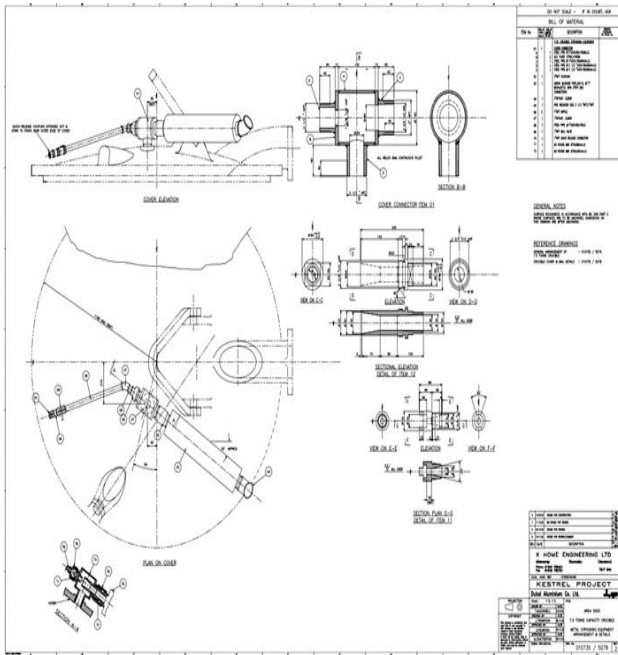


Figure 2- Crucibles and siphons maps

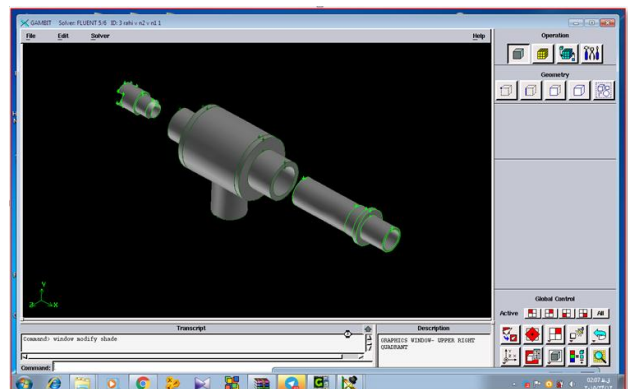


Figure 4- Gambit Software Modeling View

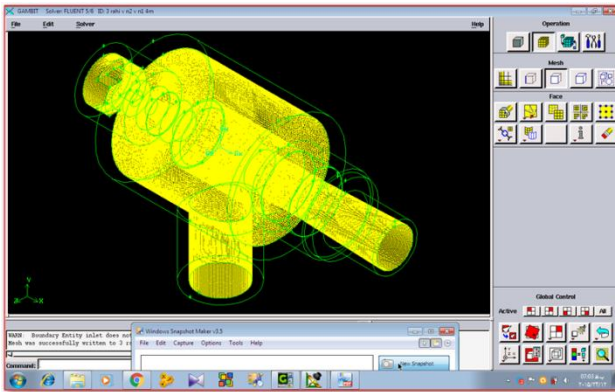


Figure 5- Gambit Software Modeling View- Gambit-meshed model

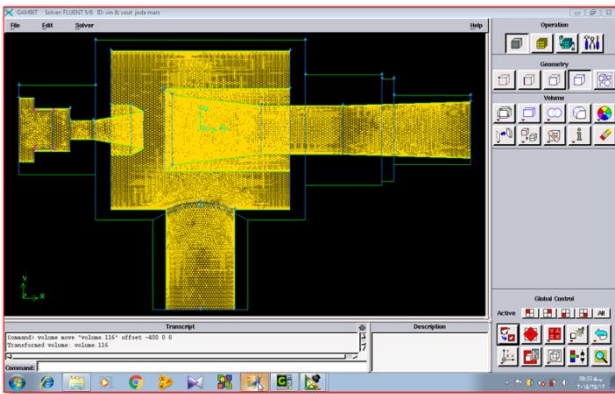


Figure 6- Gambit Software Modeling View- Gambit-based model

The basic steps for generating a vector plot are as follows:

1. In the Vectors of drop-down list, select the vector quantity to be plotted. By default, only velocity and relative velocity are available, but you can create my own custom vectors as described below.
2. In the Surfaces list, choose the surface(s) on which we want to display vectors. If we want to display vectors on the entire domain, select none of the surfaces in the list.

If we want to select several surfaces of the same type, you can select that type in the Surface Types list instead. All of the surfaces of that type will be selected automatically in the Surfaces list (or deselected, if they are all selected already).

Another shortcut is to specify a Surface Name Pattern and click Match to select surfaces with names that match the specified pattern. If they are all selected already, they will be deselected.

3. Set any of the vector plot options described below.
4. Click the Display button to draw the vectors in the active graphics window.

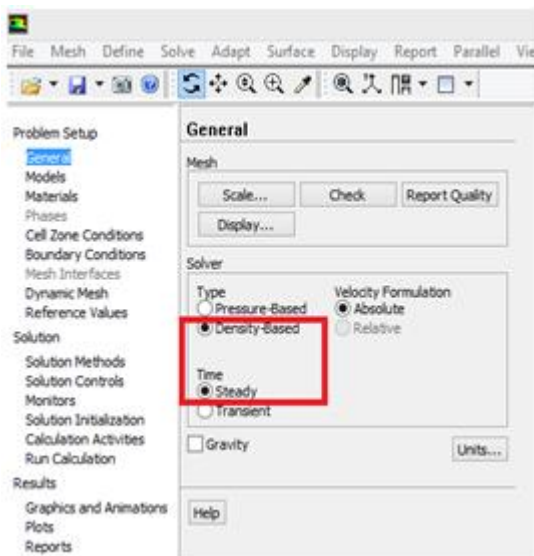


Figure 7- Display picture model information in Fluent



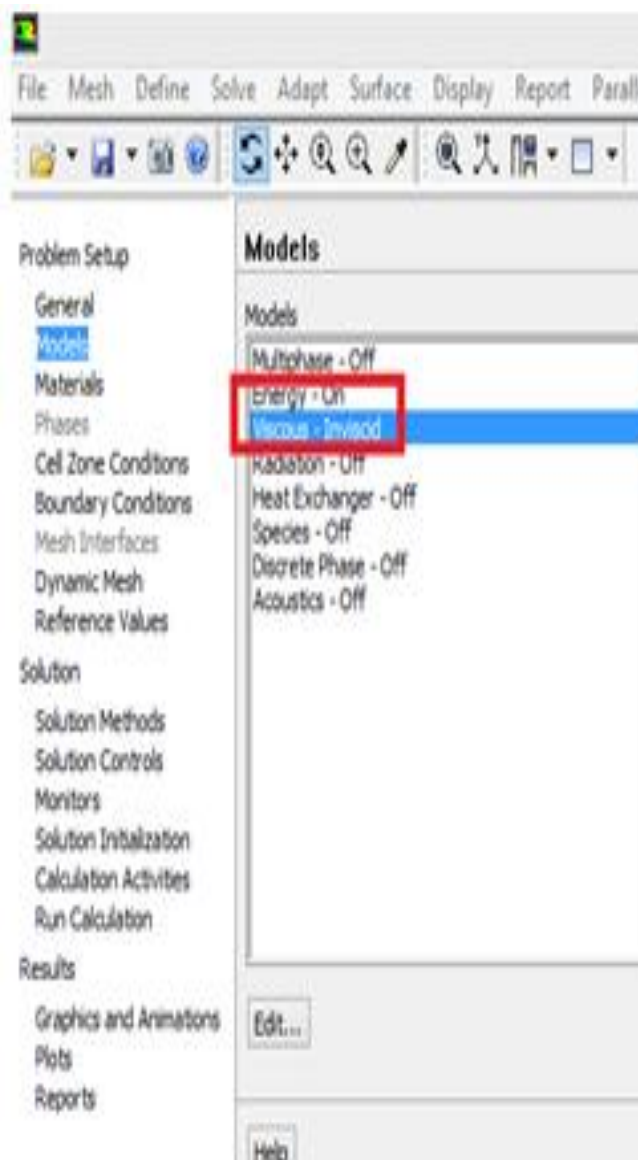


Figure 8- Display picture model information in Fluent

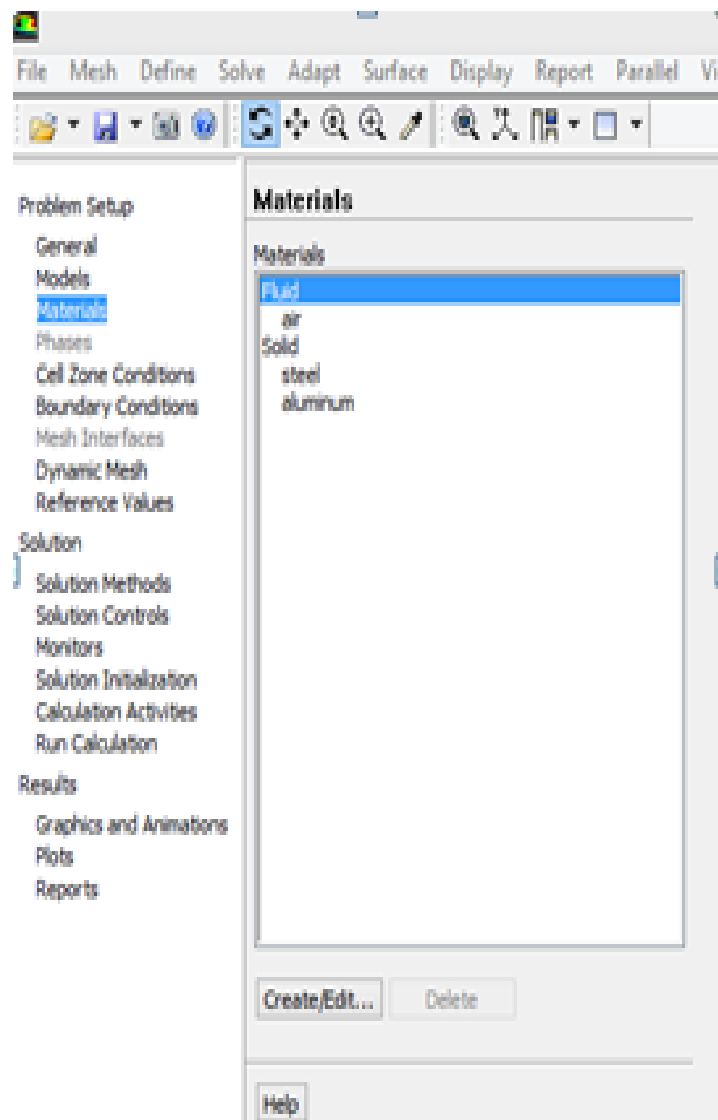


Figure 9- Display picture model information in Fluent

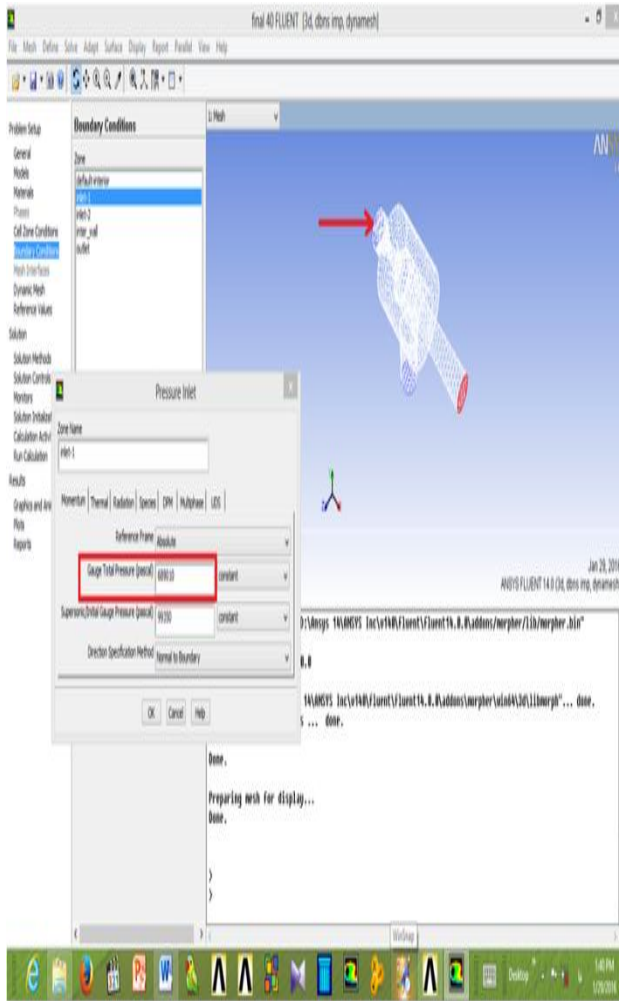


Figure 10- Display picture model information in Fluent

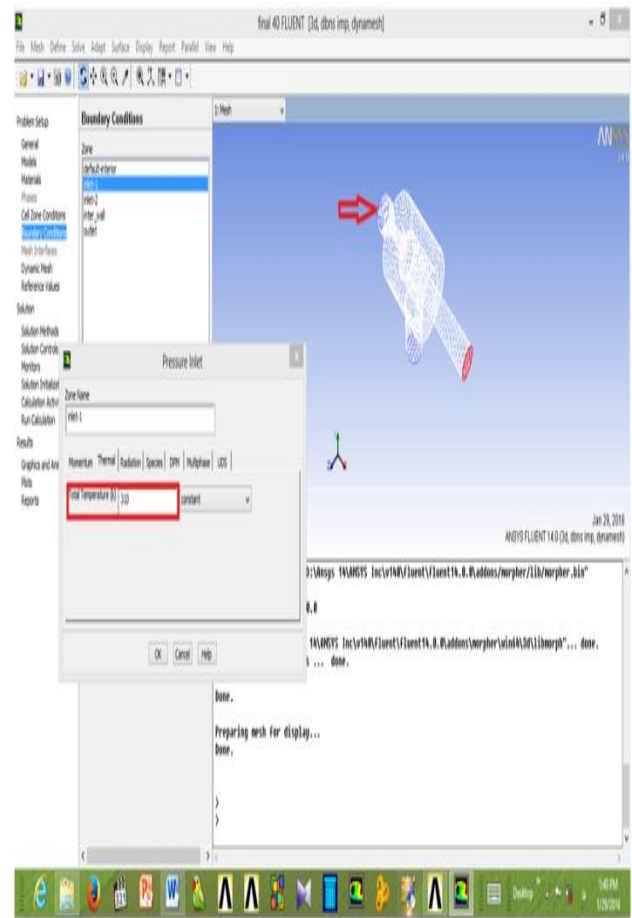


Figure 11- Display picture model information in Fluent

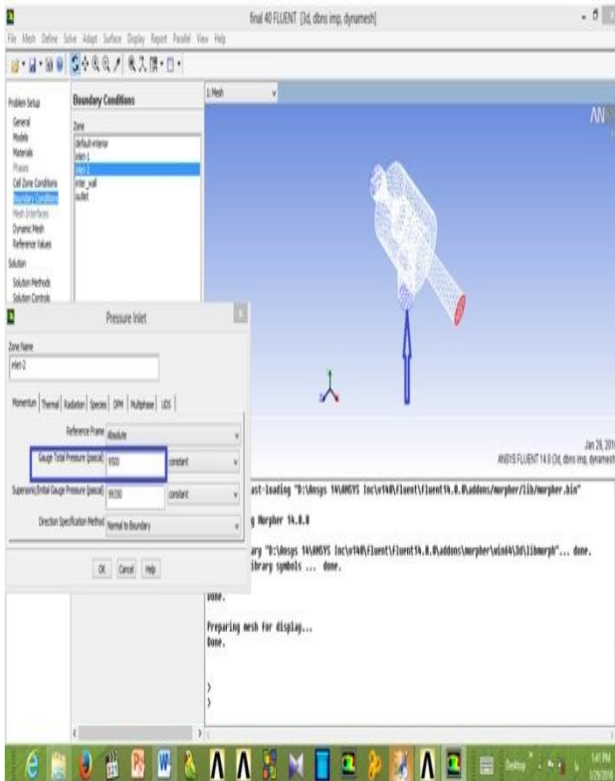


Figure 12- Display picture model information in Fluent

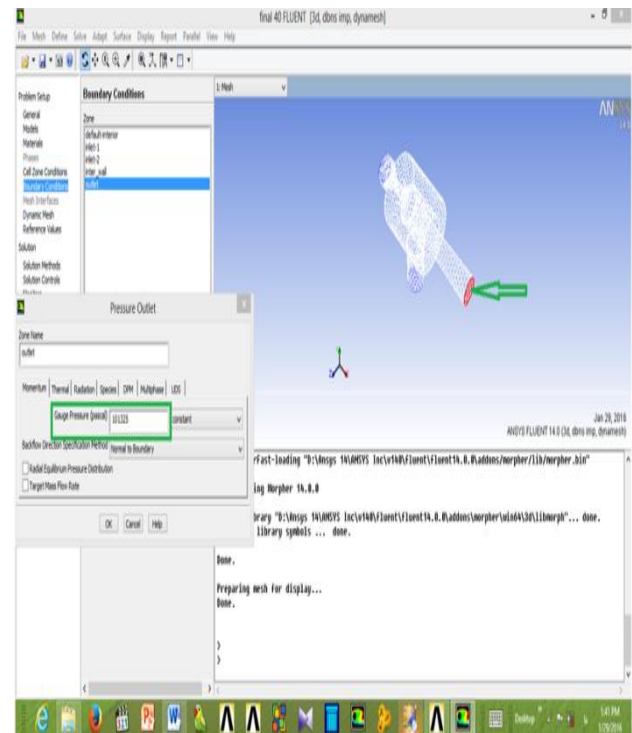


Figure 14- Display picture model information in Fluent

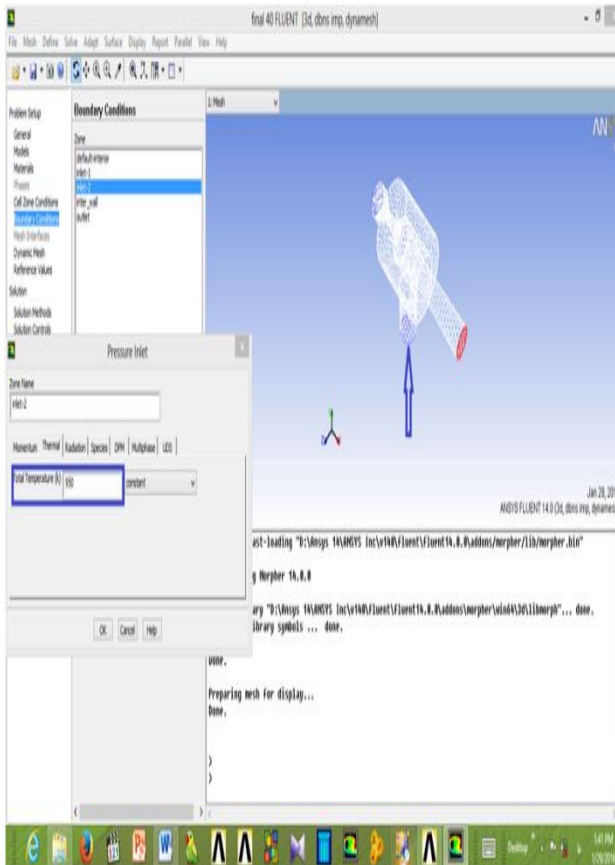


Figure 13- Display picture model information in Fluent

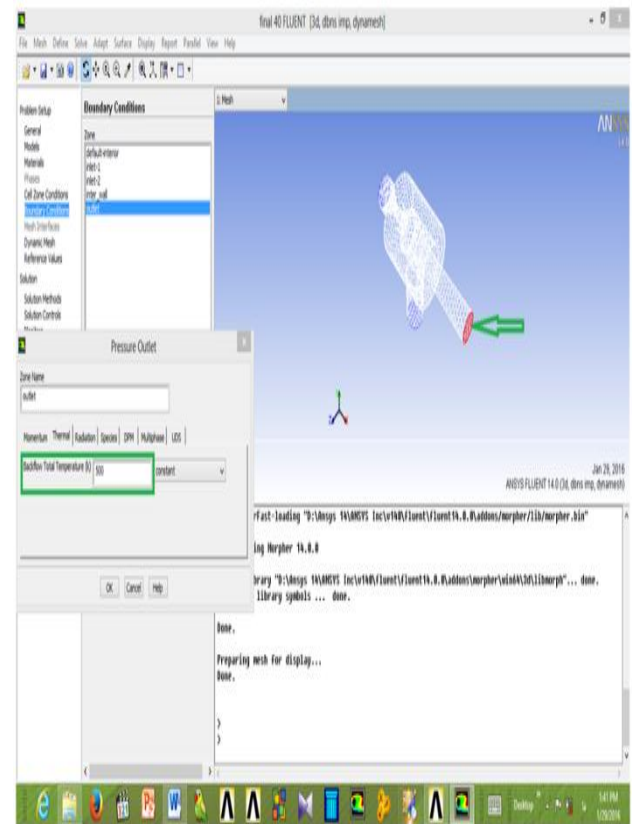


Figure 15- Display picture model information in Fluent

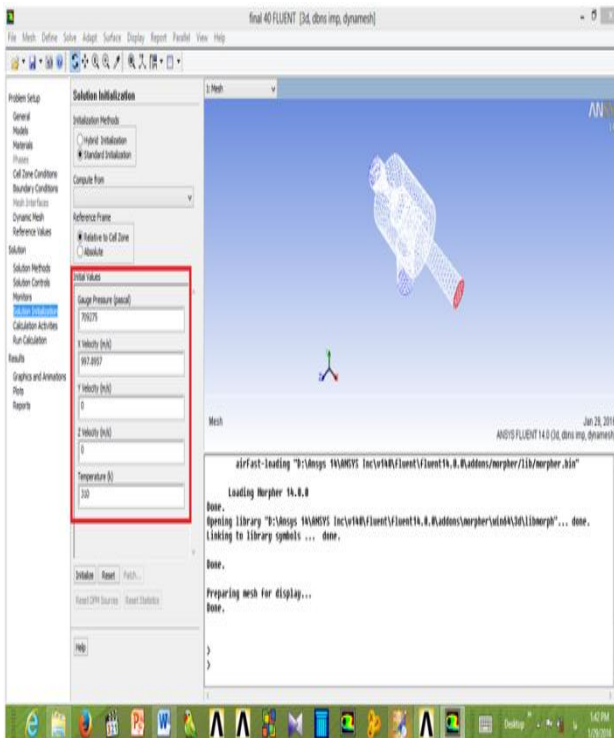


Figure 16- Display picture model information in Fluent

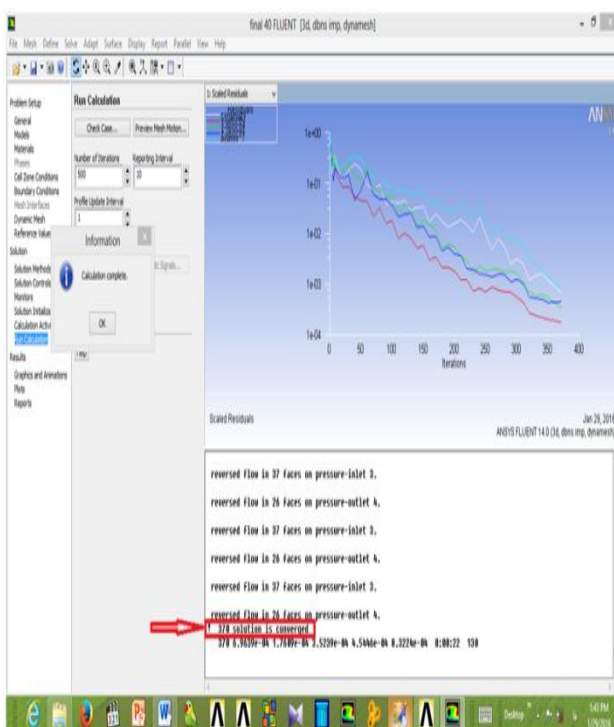


Figure 17- Display picture model information in Fluent

## Result and dissection

In this research we have tried to simulate the fluid behavior in the metal depletion three-way to

increase the technical knowledge for new designs at a low cost and less time.

The following photo shows a sample of the syphon and crucible set, which is ready to drain the metal from the pot, a three-way is shown on top of it by a black arrow.

Gambit software was used for modeling because this software is very powerful and efficient in terms of meshing the piece for further analysis in the Flaunt software.

Computational fluid dynamics (CFD) is a computer-based simulation method for analysing fluid flow, heat transfer, and related phenomena such as chemical reactions. This project uses CFD for analysis of flow and heat transfer (not for analysis of chemical reactions). Some examples of application areas are: aerodynamic lift and drag (i.e. airplanes or windmill wings), power plant combustion, chemical processes, heating/ventilation, and even biomedical engineering (simulating blood flow through arteries and veins). CFD analyses carried out in the various industries are used in R&D and manufacture of aircraft, combustion engines, as well as many other industrial products.

It can be advantageous to use CFD over traditional experimental-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimise equipment are very inexpensive with CFD when compared to experiments.

This section briefly describes the general concepts and theory related to using CFD to analyse fluid flow and heat transfer, as relevant to this project. It begins with a review of the tools needed for carrying out the CFD analyses and the processes required, followed by a summary of the governing equations and turbulence models and finally a discussion of the discretisation schemes and solution algorithms is presented.

With ANSYS Fluent still running, you can perform a simple evaluation of the velocity and temperature contours on the symmetry plane. Later, you will use CFD-Post (from within ANSYS Workbench) to perform the same evaluation.



is step, you will display the results of the simulation in ANSYS Fluent, display the results in CFD-Post, then review the list of files generated by ANSYS Workbench.

Display results in ANSYS Fluent.

Note the addition of the compressed ANSYS Fluent case and data files to the list of files.

All Fig. 18,19 the results of Fluvent-Show Cantors AND FIG20 the results of Fluvent- ntors-Show vectors AND FIG21 View the results of Fluvent-Show diagram. temperature distribution along a burner axis for two variant simulations of the SIPHONE to show the effect of different exit momentum caused by geometrical alterations at the crocybles. In combination with the Fig. 18 and Fig. 19 for the same variant calculation a significant increase in 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 colourmap. For comparison purpose the colourmap was set consistently for all variant calculations. The low momentum leads to low temperature with a higher energy density, which is responsible for the very high 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 a lower total heat input into the aluminium and an unbalanced temperature distribution in the liquid can be demonstrated.



Figure 18- View the results of Fluent-Show Cantors

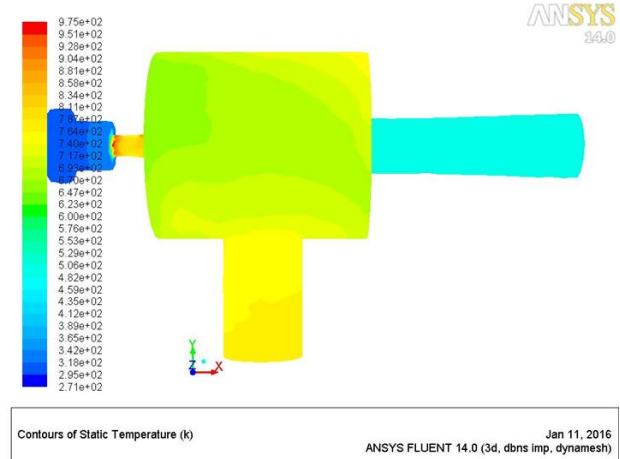


Figure 19- View the results of Fluent-Show Cantors

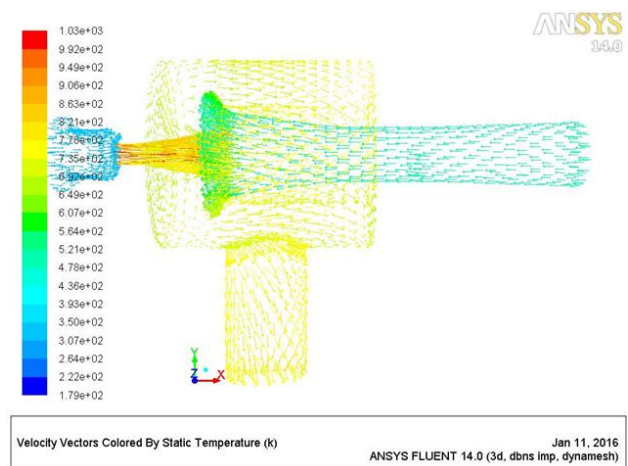


Figure 20- View the results of Fluent- ntors- Show vectors

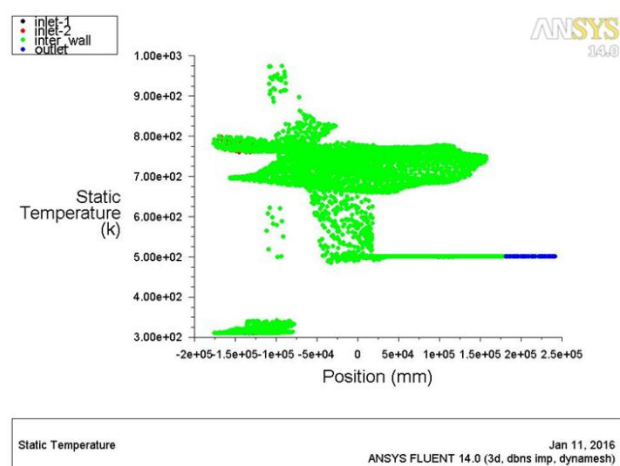


Figure 21- View the results of Fluent-Show diagram

## **Conclusion**

In This article describes a CFD-Simulation of a crucibles for primary aluminium with the commercial available tool FLUENT® and Gambit.

The many reasons CFD consulting is being widely used today are as follows:

CFD foresee performance before adapting or execution in systems

Without modifying or installing real system or a sample, CFD simulation can forecast which changes in design layout are most vital for improving performance.

CFD model offers precise and detailed information about heating, ventilation, and air conditioning design parameters

The advancement in HVAC/IAQ technology need wider and detailed information about the flow within an occupied zone while CFD meets the goal better than any other theoretical or experimental methods.

CFD modeling saves cost and turnaround time and it procures reliable results. This is the reason CFD analysis report is accounted as completely reliable.

## **References**

- [1] Henning,B.; Jasper,R.: MultiMelter© - The new generation of aluminium melt- ing furnaces, Heat Processing 2/2004, Vulkan-Verlag
- [2] Henning,B.;Klatecki,P.;Jasper,R.:Opti- mierte Regeneratortechnologie an indus- triellen Feuerungssystemen, Gaswärme- International 6/2005, Vulkan-Verlag
- [3] von Starck, A.; Mühlbauer, A.; Kramer, C.: Praxishandbuch Thermoprozesstech- nik, Vulkan-Verlag 2003