FLUE GAS CONDENSER MODELLING FOR IMPROVING THE CO2 CAPTURING EFFICIENCY

H. Nabati[1], J. Mahmoudi[2]
[1, 2] Mälardalen University, Västerås, Sweden

Corresponding Author: H. Nabati, Mälardalen University, School of Sustainable Development of Society and Technology (HST), Box 883, 721 23 Västerås, Sweden; hamid.nabati@mdh.se

Abstract. Among the main greenhouse gases that contribute in the global temperature rising, the main anthropogenic greenhouse gas is CO_2. CO_2 is produced mainly from fossil fuel combustion in the industrial process and plants. Thus carbon dioxide capturing and sequestration is of critical importance to reduce the release of CO_2 to atmosphere. One of the most recent proposed methods for CO_2 capturing from flue gas is Oxyfuel process. Oxyfuel system uses oxygen instead of air for combustion of the primary fuel. This results in production of a flue gas that contains mainly high CO_2 concentration (greater than 80% by volume) and water vapour. The water vapour is then removed by cooling in a flue gas condenser. To gain the high efficiency in water vapour removal from flue gas and preparing a pure CO_2 stream, there is a great demand for precise design of condenser. Most of the former studies in the CO_2 capturing field have pointed to the condensing method itself without any further discussion on technical aspects of condenser which is not a usual condenser. In the current paper, the available numerical multiphase flow approaches are investigated and governing mathematical equations for this flow regime are defined precisely. Then a numerical approach is implemented to simulate the two-phase flow behaviour in a simple condenser. It has been found that the Mixture and Eulerian two-phase flow models are both appropriate choices for simulation of condensation phenomena when a non-condensable gas like CO_2 exists in the flue gas stream.

1 Introduction

Currently fossil fuels account for 90 percent of global energy consumption and cannot be phased out rapidly that means we must rely on fossil fuels as the main energy source for the next several decades (1). Thus carbon capture and sequestration to reduce the release of CO_2 to atmosphere from stationary sources like fossil fuel fired power plants is of critical importance. One of the most recent proposed co2 capturing from flue gas is Oxyfuel process. Oxy fuel combustion systems use oxygen instead of air for combustion of the primary fuel to produce a flue gas that is mainly water vapour and CO_2. This results in a flue gas with high CO_2 concentrations (greater than 80% by volume). The water vapour is then removed by cooling in a CO_2/H_2O condenser. To gain the high efficiency in H_2O removal from flue gas and preparing a pure CO_2 stream, there is a great demand for precise design of the heat exchanger.

The idea in CO_2 capturing is to produce a high-pure stream of CO_2 that can be easily transported and stored in a suitable place. Considering the energy costs and other associated costs contributing in CO_2 transportation and storage, production of a nearly pure CO_2 flow is necessary to reduce the mentioned costs. Presently, the large industrial plants (like natural gas treatment plants and ammonia production facilities) use the CO_2 removing process to purify other industrial gas streams and in most cases, the CO_2 is released to the atmosphere. CO_2 removal for storage purposes has been done in few cases. CO_2 capturing processes also have been used to obtain commercially useful amounts of CO_2 from flue gas generated by the combustion of coal or natural gas. However, there have been no applications of CO_2 capture at large (e.g., 500 MW) power plants and this case is in research phase. In addition, it is useful here to mention that nowadays, the amount of CO_2 produced by a thermal plant is a major criterion of its performance, for environmental and therefore economic reasons. Thus in electrical power stations a new measure of the performance is the amount of CO_2 produced per unit of electricity generated, i.e., λ = kg(CO2)/kwh (2).

Depending on the process or power plant application, there are three main approaches to capture the CO_2 generated from a primary fossil fuel (coal, natural gas or oil), biomass, or mixtures of these fuels:
Post-combustion, Pre-combustion and Oxyfuel combustion.

Figure 1 shows the main steps in the CO_2 capturing process in a conventional fossil-fuel fired power plant. After the fossil fuel has been burnt to produce power, the CO_2 is separated from the flue gas stream. Then depend on available facilities, the CO2 is suitably used or is stored for a long period of time.

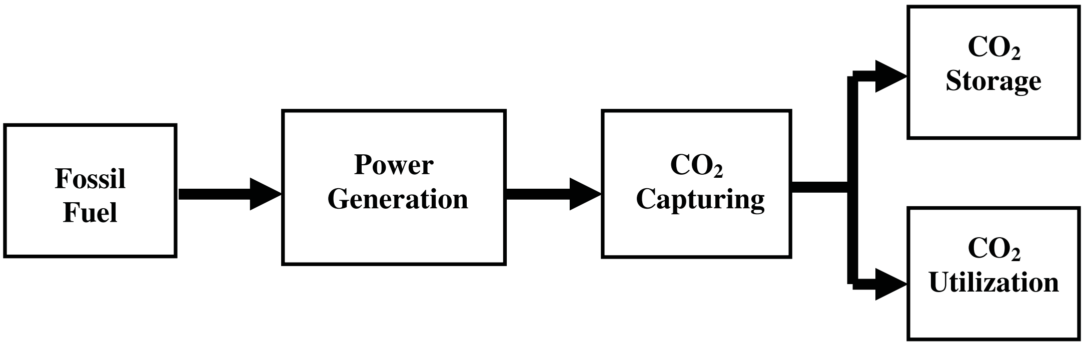

Figure 1. Main steps in CO_2 capturing process

As a method of CO_2 capturing in boilers, Oxyfuel combustion systems are in the demonstration phase. Oxyfuel systems are also being studied in gas turbine systems, but conceptual designs for such applications are still in the research phase. The Oxyfuel combustion systems use pure oxygen instead of air for combustion of the primary fuel and produces a flue gas consisting mainly water vapour and CO_2. At first, the nitrogen is separated from the air in an air separation unit (ASU, with a purity of 95–99%) and the fuel is then combusted with the oxygen. This results in a flue gas with high CO_2 concentrations (greater than 80% by volume). The water vapor could be then removed by flue gas cooling in a condenser.

In the Oxyfuel processes, the condenser for CO_2 and steam separation is the key device. Despite of wide referring to this kind of condensers in different Oxy fuel cycles, the complete design data are not yet available and new experimental and numerical studies are necessary to achieve the desired efficiency for CO_2 capturing and steam separation. There are several challenging topics in this field including: thermophysical properties of CO_2/H_2O mixture, condensation of water vapour in presence of CO_2 as a non-condensable gas, CO_2/H_2O phase equilibrium, different condensing design (3).

On the other hand, flow field inside the condenser plays important role on thermal efficiency of condenser. This paper implements CFD to study the flow field inside a proposed condenser. The results from this paper will be helpful and beneficial for the industrial development and design of the new condensers for oxy-fuel process with deeper knowledge of flow behaviour and multiphase flow models.

2 Numerical approaches for phase change

Presently two approaches are used in numerical modelling of multiphase flows: the Euler-Lagrange approach and the Euler-Euler approach. The first step in solving the multiphase problem is to assess which of the multiphase flow regimes best represents our case flow. Then we can select the appropriate model that can be used for numerical selection.

In the Euler-Lagrange approach(mentioned above), the fluid phase is considered as a continuum by solving the time-averaged Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. The fundamental assumption in this model is that the dispersed second phase occupies a low volume fraction, even though high mass loading is acceptable. This model is appropriate for the modelling of spray dryers, coal and liquid fuel combustion, but inappropriate for the modelling of liquid-liquid mixtures, fluidized beds, or any application where the volume fraction of the second phase is not negligible.

In the second (Euler-Euler) approach, the different phases are considered as interpenetrating continua. The volume of a phase cannot be occupied by the other phases, so the phasic volume fraction concept is used which are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases. These equations are closed by providing constitutive relations that are obtained from empirical information. There are three Euler-Euler models available for multiphase flow modelling in the FLUENT: the volume of fluid (VOF) model, the mixture model and the Eulerian model (4).

The Volume of Fluid (VOF) model is a surface tracking technique applied to a fixed Eulerian mesh. Wherever the position of the interface between the two or more immiscible fluids is of interest, this model is used. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. Applications of the VOF model include free-surface flows, sloshing, the motion of large bubbles in a liquid, the prediction of jet breakup (surface tension), and the steady or transient tracking of any liquid-gas interface.

The mixture model is a simplified multiphase model that can be used to model multiphase flows where the phases move at different velocities, but assume local equilibrium over short spatial length scales. The coupling between the phases should be strong. It can also be used to model homogeneous multiphase flows with very strong coupling and the phases moving at the same velocity.

Typical applications include sedimentation, cyclone separators, particle-laden flows with low loading, and bubbly flows where the gas volume fraction remains low. Only one of the phases can be defined as a compressible ideal gas. There is no limitation on using compressible liquids using user-defined functions. The Eulerian multiphase model is used for the modelling of multiple separate phases. The phases can be liquids, gases, or solids in nearly any combination. Compressible flow, inviscid flow, melting and solidification are not allowed.

During the rapid expansion of steam, a condensation process will take place shortly after the state path crosses the vapor-saturation line. The expansion process causes the super-heated dry steam to first subcool and then nucleate to form a two-phase mixture of saturated vapour and fine liquid droplets known as wet steam (5).

FLUENT has adopted the Eulerian-Eulerian approach for modelling the wet steam flow that is known as Wet Steam Model.

Among these models, the Eulerian multiphase model and the mixture model are the most suitable ones for condensation modelling.

3 Governing equations

The Governing equations for the problem are continuity equation, general transport equation and energy equation that are solved using Fluent software. The continuity equation for phase q in vector form is:

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q\right) + \nabla.\left(\alpha_q \rho_q \vec{v}_q\right) = \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp}) + S_q \quad (1)$$

Where $\vec{v}_q$ is the velocity of phase q and $\dot{m}_{pq}$ characterizes the mass transfer from the p^{th} to q^{th} phase, and $\dot{m}_{qp}$ stands for the mass transfer from phase q to phase p.

The momentum balance for phase q is:

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q \vec{v}_q\right) + \nabla.\left(\alpha_q \rho_q \vec{v}_q \vec{v}_q\right) = -\alpha_q \nabla p + \nabla.\bar{\bar{\tau}}_q + \alpha_q \rho_q \vec{g} + \sum_{p=1}^{n}\left(\vec{R}_{pq} + \dot{m}_{pq}\vec{v}_{pq} - \dot{m}_{qp}\vec{v}_{qp}\right) + \left(\vec{F}_q + \vec{F}_{lift,q} + \vec{F}_{vm,q}\right) \quad (2)$$

Where $\bar{\bar{\tau}}_q$ is the q^{th} phase stress-strain tensor and is expressed by following relation:

$$\bar{\bar{\tau}}_q = \alpha_q \mu_q\left(\nabla \vec{v}_q + \nabla \vec{v}_q^{\,T}\right) + \alpha_q\left(\lambda_q - \frac{2}{3}\mu_q\right)\nabla.\vec{v}_q \bar{\bar{I}} \quad (3)$$

μ_q and λ_q are the shear and bulk viscosity of phase q, $\vec{F}_q$ is an external body force, $\vec{F}_{lift,q}$ is a lift force, $\vec{F}_{vm,q}$ is a virtual mass force, $\vec{R}_{pq}$ is an interaction force between phases, and p is the pressure shared by all phases. $\vec{v}_{pq}$ is the inter-phase velocity and depends on $\dot{m}_{pq}$ as follows. If $\dot{m}_{pq} > 0$ that means phase p mass is being transferred to phase q then $\vec{v}_{pq} = \vec{v}_p$ and if $\dot{m}_{qp} < 0$ that means phase q mass is being transferred to phase p), $\vec{v}_{pq} = \vec{v}_q$. This is similarly valid for $\vec{v}_{qp}$.

The conservation of energy in multiphase applications comprises of a separate enthalpy equation for each phase:

$$\frac{\partial}{\partial t}\left(\alpha_q \rho_q h_q\right) + \nabla.\left(\alpha_q \rho_q \vec{u}_q h_q\right) = -\alpha_q.\frac{\partial p_q}{\partial t} + \bar{\bar{\tau}}_q.\nabla \vec{u}_q - \nabla.\vec{q}_q + S_q + \sum_{p=1}^{n}\left(\vec{Q}_{pq} + \dot{m}_{pq}h_{pq} - \dot{m}_{qp}h_{qp}\right) \quad (4)$$

I. Troch, F. Breitenecker, eds.   ISBN 978-3-901608-35-3

Where h_q is the specific enthalpy of the q^{th} phase, $\vec{q}_q$ is the heat flux, S_q is a source term that includes sources of enthalpy (e.g., due to chemical reaction or radiation), Q_{pq} is the intensity of heat exchange between the p^{th} and q^{th} phases, and h_{pq} is the interphase enthalpy. The heat exchange between phases must comply with the local balance conditions $Q_{pq} = -Q_{qp}$ and $Q_{qq} = 0$.

4 Numerical solution methodology

The 2-D model is a simplified geometry that contains the important aspects of main. The geometry consists of walls, three velocity inlets and three pressure outlets. The flow domain is shown in figure 1. Very fine mesh has been used for the whole geometry. This fine mesh also has been smoothed in the Fluent again to get the best results. Figure 1 shows also a close view of produced mesh in two important part of condenser.

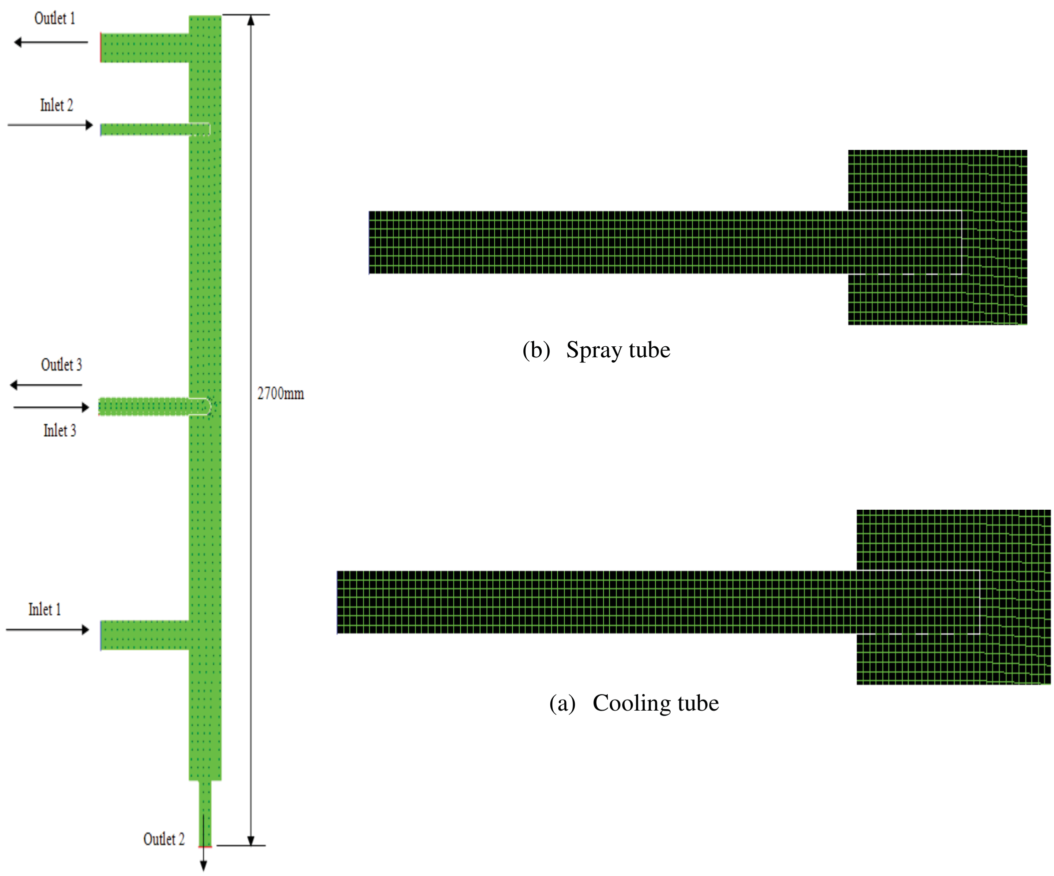

Figure 2. 2D geometry of condenser

Based on the Reynolds number, the k-ε model is used for turbulence modeling. The material, which is used for simulation, is considered as CO_2. All properties are considered as default values defined in the Fluent program. Gravity and buoyancy effect is considered for the flow field. The boundary conditions are as following:

5 Result and discussion

Following figures are showing some results taken from Flunet post-processor. The velocity vectors are illustrated in figure 3. It can be seen that the most part of flow goes out from the bottom without participation in condensing process (in a real condenser). The other interesting point is that there are some recirculation regions in the condenser without any participation in interactions. These are dead zones, which should be avoided, in a real design. A close view of vectors in the spray tube (figure 4) shows that the simulation is done in a right physical way and spray penetrates in the main flow.

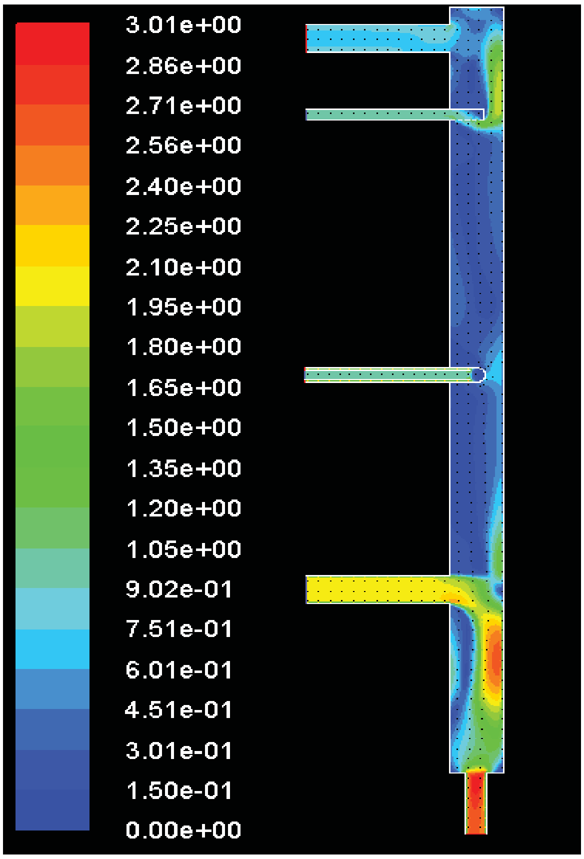

Figure 3. velocity contour in the solution domain

Figure 4. velocity Close view of spray tube showing spray penetration into the main flow

Contour of pressure shows that the pressure drop through the condenser is low. The pressure drop through the cooling tube is very high which affect the mass flow rate. So this configuration doesn't seem a good design for cooling water. The configuration here is one tube included inside the other which shows poor performance. Thus the usual tubes loop seems also to be better choice as a cooling device in such kind of condenser. It can be seen that there are some parts with negative pressure that can be a result of high entrance velocity of flue gas. These negative pressures result in recirculation inside the condenser and reduce the efficiency seriously.

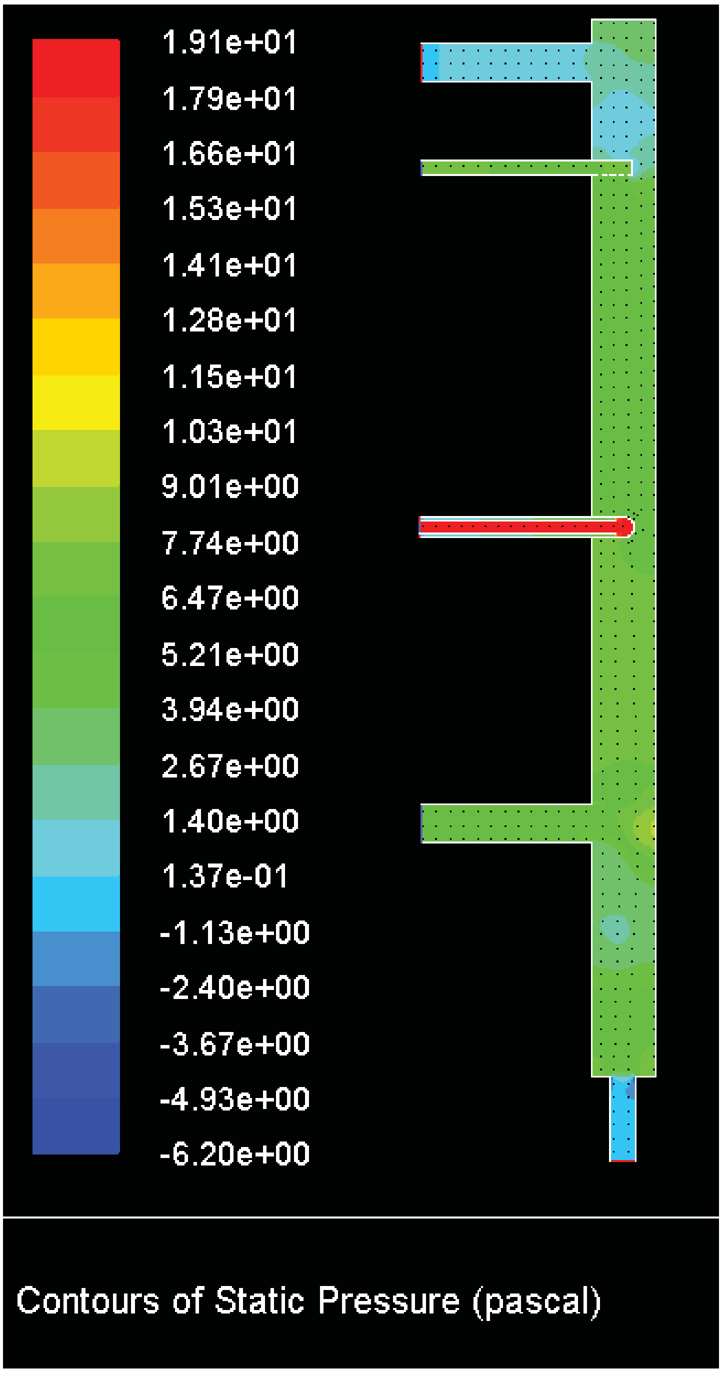

Figure 5. Contour of static pressure

Finally CO_2 volume fraction is illustrated in figure 6. It is obvious that CO2 fraction reduces in places near the cooling tube and spray water, as these locations contain more water in liquid phase. However low concentration of CO_2 near the spray tube seems to be a result of movement of water droplets with the flue gas, as the CO_2 volume fraction follows the velocity vectors profile.

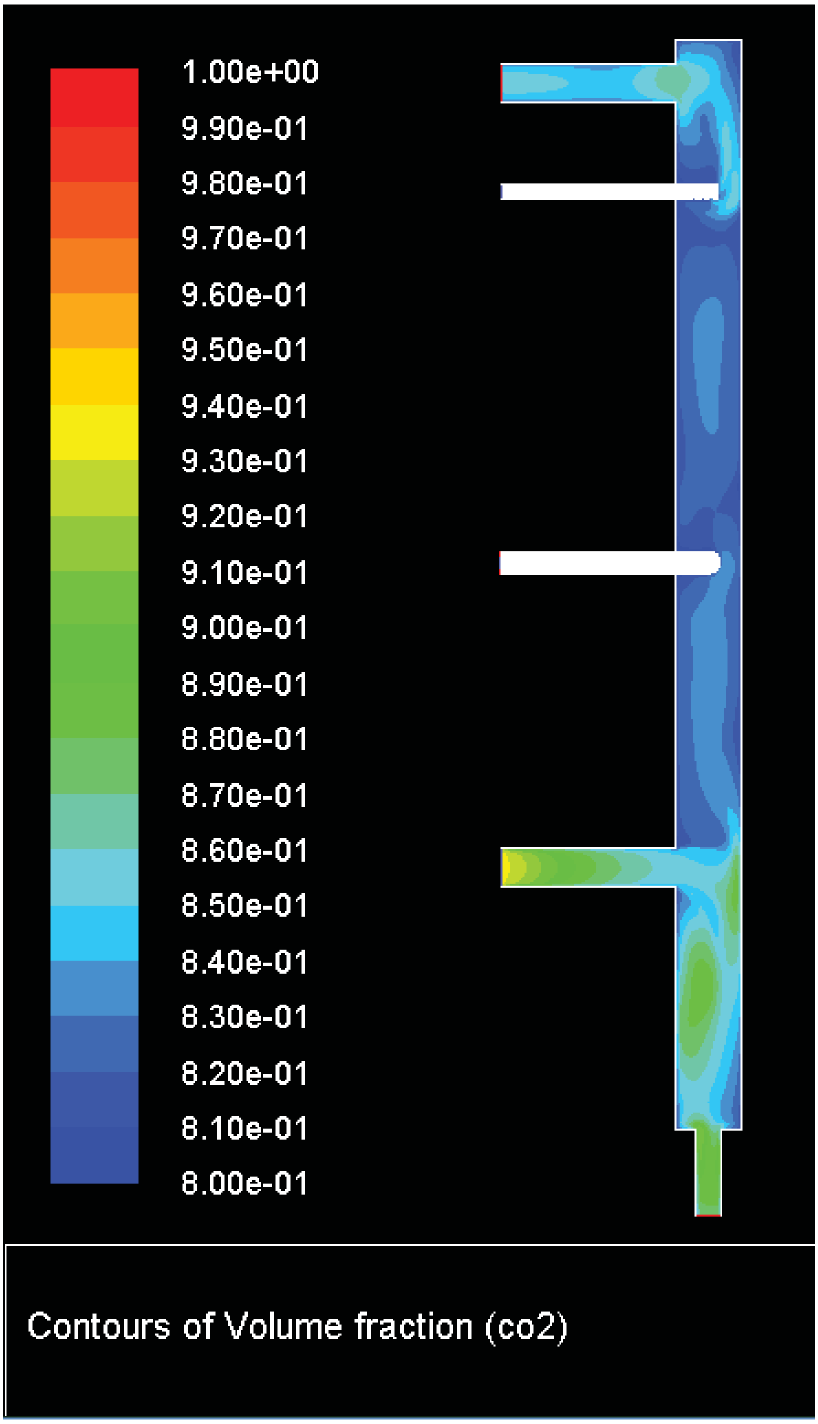

Figure 6. Contour of CO_2 volume fraction

6 Conclusion

This paper presents numerical models to solve the complex transport phenomena in a condenser which is studied to be implemented in an oxy fuel process. The “mixture” method is implemented to capture the multiphase flow and flow behaviour inside the condenser. This model is combined with the turbulent flow and energy equations via additional source terms in these equations. The calculated flow field shows flow mainly influenced by the cooling tube configuration and spray tube location. The distribution of the velocity and CO_2 fraction can be seen. Furthermore the model shows that CFD has ability to predict the condensation from flue gases. Anyhow the should be verified against some experimental data. Currently such data are not easily available as the oxy fuel process is under the research phase.

7 References

[1] Davison, J, Freund, P and Smith, A. *Putting carbon back into the ground.* IEA Greenhouse Gas R&D Programme. 2001. ISBN: 1 898373 28 0.

[2] Horlock, J. H. *Advanced GasTurbine Cycles.* s.l. : Elsevier Science Ltd., 2003.

[3] Wilhelmsson, Jinliang Yuan, Bengt Sundén.*Water condensation and two-phase flow modeling for a plate heat exchanger channel.* Charlotte Wilhelmsson, Jinliang Yuan, Bengt Sundén. Seattle, USA : s.n., 2007. ASME International Mechanical Engineering Congress and Exposition.

[4] Fluent6.2 User Guide. 2005.

[5] Cengel, Yunus A. and Boles, Michael A. *Thermodynamics, An Engineering Approach.* International student edition . s.l. : McGraw Hill , 2007. ISBN-13: 978-0071257718 .

[6] Swedish Biogas. [Online] 2004. http://www.sss.se.