Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 3, 2010, vol. LVI
article No. 1807

Tomáš BLEJCHAŘ*

CFD SIMULATION OF JET FLOW AND COMPARISION WITH MEASUREMENT

CFD SIMULACE VOLNÉHO SMYKOVÉHO PROUDU A POROVNÁNÍ S MĚŘENÍM

Abstract

The paper deals with numerical modeling of free shear jet flow, and spraying of liquid which is dispersed in combustion chamber. Numerical modeling of SNCR [3] in combustion chamber is strongly influenced by intensity of reagent dispersion in flue gas. That is why the numerical model of reagent spraying has to be verified with a view to real measured parameters of nozzles. Numerical model for cone and plane spraying nozzle was compiled. Results of numerical model were compared with results of experimental measurement for both types of nozzles.

Abstrakt

Článek se zabývá numerickým modelováním proudění na výstupu z trysky, a rozstřiku kapaliny, která je tryskou dopravována do požadované oblasti ve spalovací komoře. Modelování procesu SNCR [3] ve spalovací komoře kotle je dominantně ovlivněn intenzitou rozstřiku reagentu do spalin. Aby bylo možné proces modelovat korektně bylo nutné ověřit, zda model rozstřiku a výrobcem trysek deklarované parametry odpovídají skutečnosti. Modelována byla tryska s kuželovým a rovinným rozstřikem. Výsledky těchto modelů byly dále porovnány s experimentálním měřením, které bylo provedeno u identických trysek.

1 INTRODUCTION

The SNCR technology is relatively simply-technological unit, but it includes some important parts, which dominantly influence the efficiency of NO_x reduction. One of important part is nozzle. The nozzle is located on the end of injection lance. Injection lance spray the reagent into flue gas with temperature 850-1050°C. Three types of nozzles are commonly used on the present. The nozzles are 1) cone spraying nozzle, 2) plane spraying nozzle, 3) cone spraying nozzle with direction 45°.

Main parameters of the nozzle and injection lance respectively are area influenced by spayed reagent and deep of penetration of flue gas by reagent. The measuring of spraying process is not realisable in situ, because the nozzles are located in combustion chamber. With regard to minimum information about process of spraying was decided on detail analyze of flow in nozzle. The CFD and experimental measurement are used to detail analysis of flow field. The spaying process is theoretically represented by mixing of two perpendicular flows of liquids. Main flow is flue gas and second flow is air with droplets of urea solution i.e. reagent. The spraying process analysis was divided into two steps. First step was experimental measuring of free stream flow so-called jet stream. The experimental measurement included visualisation of droplets volume fraction too. Second step was CFD simulation of cases, which are identical with experimental measurement.

* Ing., Ph.D., VŠB – Technical University of Ostrava, Faculty of Mechanical Engineering, Department of Hydrodynamics and Hydraulic Equipment, 17. listopadu 15, Ostrava - Poruba, 708 33, Czech Republic tel. (+420) 59 732 5753, e-mail tomas.blejchar@vsb.cz

2 MEASUREMENT

2.1 Physical Principe of Measurement

Velocity measurement was realised by micro Prandtl probe. The Prandtl probe used for experimental measurement is laboratory type. Velocity measurement is commonly performed by measurement of dynamic pressure in fluid mechanics, and short form of Bernoulli equation is used to specify dynamic pressure. Velocity is given by following equation [4]:

$$v = \sqrt{2 \cdot \frac{p_d}{\rho}} \tag{1}$$

where:

v – is velocity $\left[\frac{\mathrm{m}}{\mathrm{s}}\right]$,

p_d – is dynamic pressure $[\mathrm{Pa}]$,

ρ – is density $\left[\frac{\mathrm{kg}}{\mathrm{m}^3}\right]$,

2.2 Measuring of Flow Field

Experimental measurement was realised in free space. The measuring grid was specified as rectangular see Fig.1. Velocity was measured in every points of measuring grid. Then the contour of velocity and velocity profiles was specified on the basis of measured data. Measuring was performed for all types of nozzles. Dynamic pressure was measured by digital micro manometer with range 10000 Pa

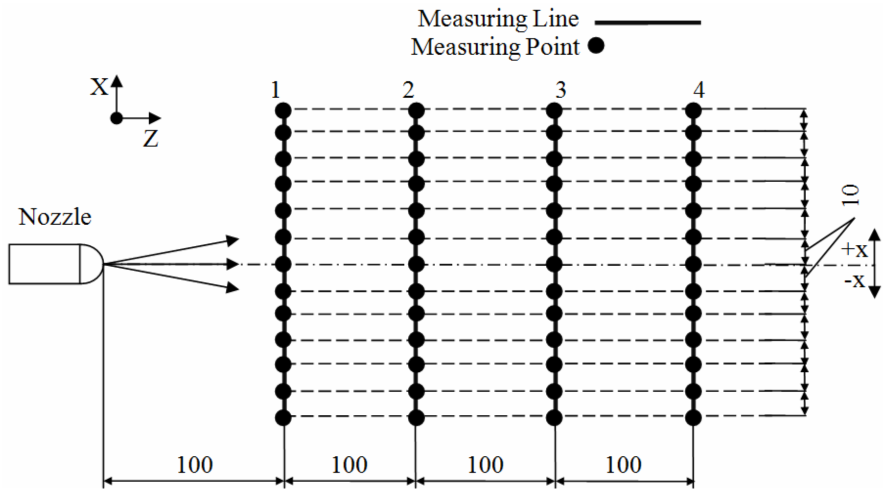

Fig. 1 Measuring grid

Tab. 1 Measured Axial Velocity, Cone Nozzle

		x [mm]												
v_{axi} $\left[\frac{m}{s}\right]$		-60	-50	-40	-30	-20	-10	0	10	20	30	40	50	60
z[mm]	100	0.0	0.0	0.0	0.0	2.9	54.9	180	54.9	2.9	0.0	0.0	0.0	0.0
	200	0.0	0.0	0.0	5.7	26.9	55.4	77.4	55.4	26.9	5.7	0.0	0.0	0.0
	300	0.0	4.0	8.0	17.0	27.2	37.9	45.9	37.9	27.2	17.0	8.0	4.0	0.0
	400	4.0	8.1	13.1	18.7	25.6	29.8	32.1	29.8	25.6	18.7	13.1	8.1	4.0

2.3 Visualisation of Particle Spraying

Visualisation of flow field and liquid droplets respectively was performed by long time exposition photo. Light slice was used to creation of focusing plane. The plane of focusing was identical to measuring plane. This result was used to evaluation of angel of spraying and verification of CFD model.

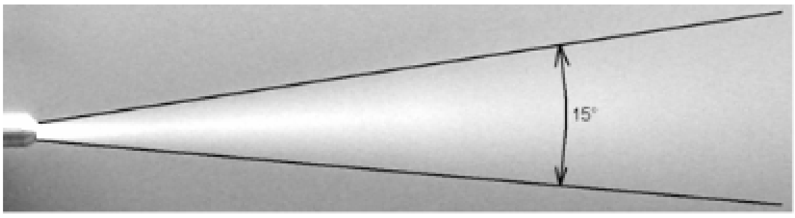

Fig. 2 Visualisation of water particles concentration, cone nozzle

3 CFD MODEL OF NOZZLES

The basic equation set which describes the bought regimes of flow represents application of conservation law. The law of mass, momentum and energy conservation is used to numerical modelling of flow. The law of momentum conservation represents Navier-Stokes equation. The Law of mass conservation represents continuity equation, and the law of energy equation represents energy equation.

Turbulent flow is stochastic, but it is statistic stable. Arbitrary physical quantity can be decompiled on sum of average value and fluctuation value $u = \overline{u} + u'$. Commonly used turbulent models are based on description of local turbulent intensity by length and velocity scale. Flow regime was analysed by means of Reynolds number.

$$\mathrm{Re} = \frac{\rho \cdot v \cdot D}{\eta} \approx 100000 \tag{2}$$

where:

v – is velocity $\left[\frac{\mathrm{m}}{\mathrm{s}}\right]$,

D – is characteristics dimension $[\mathrm{m}]$,

ρ – is density $\left[\frac{\mathrm{kg}}{\mathrm{m}^3}\right]$,

η – is dynamics viscosity $[\mathrm{Pa.s}]$, It is fully turbulent flow and the jet flow produces shear stress. That is why Shear Stress Transport k-ω turbulence model is used for CFD simulation [1]. The turbulent viscosity is specified by two transport equations. These equations solve turbulent kinetic energy k and vorticity ω [1],[2]. The production of turbulence by expansion is included in the simulation too.

Computation domain was build for all type of nozzles, cone, plane and cone 45°. Domain was simplified by symmetry planes. Cone and/or Plane Nozzle domain include two symmetry planes, so only 1/4 domain was included in CFD model. Cone 45° includes only one symmetry plane so only 1/2 domain was included in CFD model. Boundary condition in CFD was identical with case of experimental measurement. Because the high pressure gradient is in throttle area of the nozzles, the compressibility of air has to be included in simulations. Inlet boundary condition was specified as Total pressure inlet. Outlet was specified as opening boundary condition. Other surfaces were specified as adiabatic wall. Particles were simulated by Lagrange particle transport model. Second order interpolation was applied to numerical solution of differential equations.

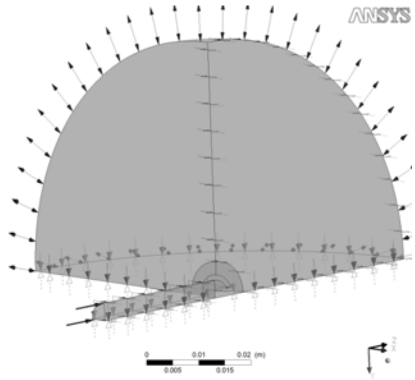

Fig. 3 Computation domain, cone nozzle.

4 RESULTS

The nozzle flow analysis produces a lot of results, and it can be only small part of results presented. The results of CFD and experimental measurement were compared by means of same quantities. First comparison rests in visualisation of the flow field see Fig. 3. Second comparison includes graphs of axial velocity and velocity profiles see Fig.4. In the picture we can see profiles of velocity on axis of the nozzle, left side of Fig. 4. Velocity profile on line, which distance from nozzle is 100mm is shown in the Fig.4. right side. Same evaluation of the results was made for other lines (distance 200, 300mm). Identical analysis of the results was made for plane nozzle and cone 45°nozzle.

Measurement CFD

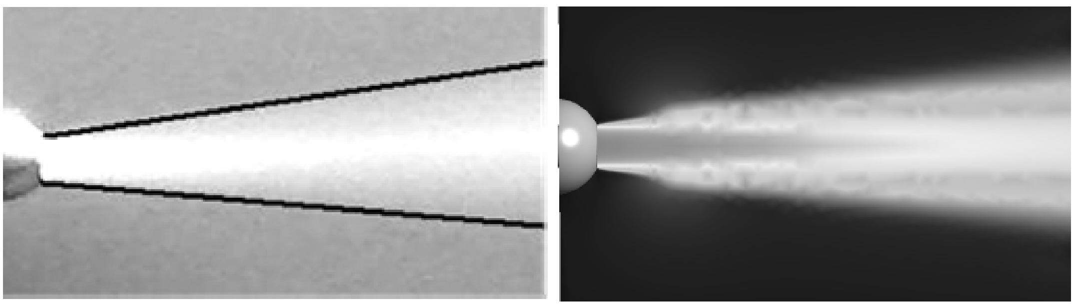

Fig. 4 Comparison of Flow Field, Measurement vs. CFD.

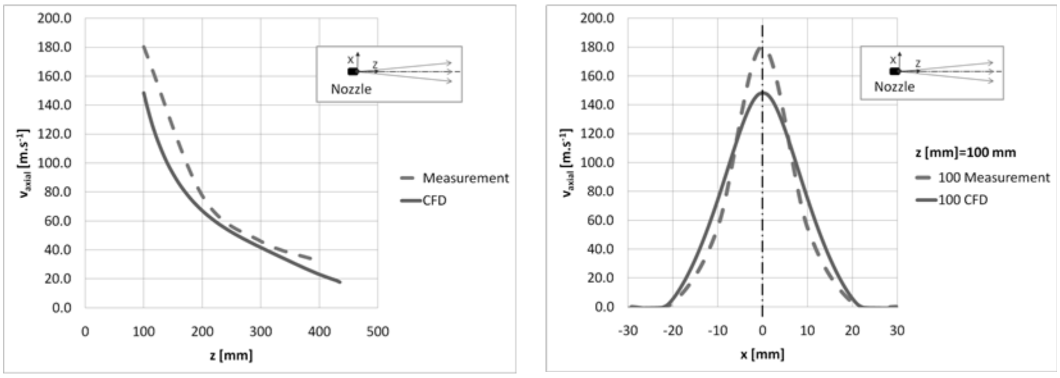

Fig. 5 Comparison of axial velocity and velocity profiles, measurement vs. CFD, cone nozzle.

5 CONCLUSION

The goal of this project is proposition of verifications of the nozzles parameters which are the basic part of SNCR technology. First the experimental measurement of the flow field was performed. Velocity was measured by means of micro Prandtl probe. Spraying of the liquid of the droplets was visualised too. Next the CFD simulation of same cases was accomplished. Results of experimental measurement and CFD simulation were thoroughly compared. Differences between measurement and CFD simulation are relatively small. So the model of droplets dispersion is simulated correctly and incorrectness of the results of CFD can be accepted. These results will be applied in complex simulation of combustion chamber, where the nozzles have to be simulated only as injection point solution and the real geometry of the nozzle cannot be included in the simulation.

AKNOWLEDGMENT

This project was supported by financial resources of state budget of The Ministry of Industry and Trade. TIP FR-TI1/547

REFERENCES

[1] ANSYS INC. *ANSYS_CFX, V12. User's guide* ANSYS Inc., 2009.

[2] KOZUBKOVÁ, M. *Modelování proudění - Fluent I,* VŠB-TU Ostrava, 2008. Available from: <URL: http://www.338.vsb.cz/studium9.htm>.

[3] BLEJCHAŘ, T. CFD Model of NOx Reduction by SNCR Metod. In *Sborník vědeckých prací Vysoké školy báňské - Technické univerzity Ostrava, Řada strojní,* 2009, LV, č. 3, s.1-6. ISSN 1210-0471

[4] DRÁBKOVÁ, S., KOZUBKOVÁ, M. *Cvičení z Mechaniky tekutin*. *Sbírka příkladů*. VŠB-TU Ostrava, 2004. Available from: <URL: http://www.338.vsb.cz/studium9a.htm