M.Sc Aryoso Nirmolo
Supervisor: Prof. Dr.-Ing. E. Specht

# CFD Simulation of Combustion in a Cross-flow by Radial Injection

**Key words:** combustion, mixing, cross-flow, jets injection, penetration depth

In the research dealing with mixing in a cross-flow, the treatment was limited to multiple jets oriented perpendicular to cylindrical chamber centerline ($\theta_j$ = 90°) like shown in **Figure 1**. Dynamic pressure ratio of jet-to-mainstream (*J*) is found to be an important parameter for a mixing in the crossflow. The parametric variables that were investigated include variation in the number of jet nozzles, variation in jets velocity, and variation in nozzle diameter (*d*), that were made dependent on the number of nozzles, jet velocity, and jet volumetric flow in order to keep a constant Volumetric Ratio.

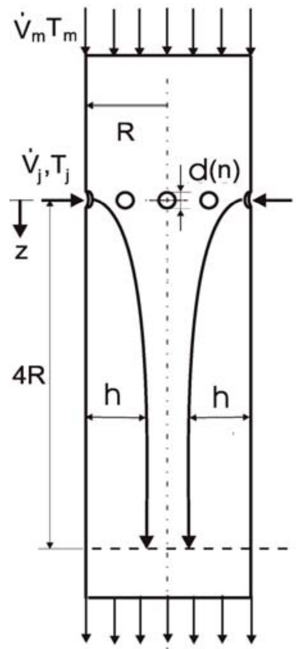


**Figure 1**. Schematic Diagram of Chamber

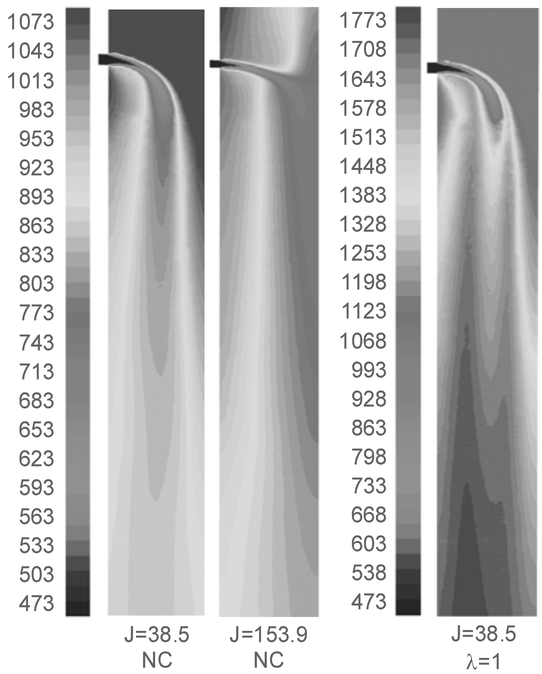


**Figure 2**. Contours of Temperature Field in K

Two types of gases were used for the axial mainstream, which were: hot waste gas for calculation with combustion and hot air for non-combustion calculation. Pre-heated air is used in both calculation as oxidizer entering radially into combustion chamber. **Figure 2** shows the temperature contours for non-combustion and for combustion with $\lambda = 1$, which is shown as a temperature distributions in axial and radial plane through the geometric center of the nozzle. The penetration depth (*h/R*) is defined as the maximum distance from the chamber wall (*h*) divided by chamber radius (*R*) which is located along the jet penetration trajectory. In the case of non-combustion, *h/R* increases as *J* increases. It is obvious that the flow at *J*=153.9 is overpenetrated. Overpenetration forms unmixed regions along the chamber walls, which can also increases the pressure needed for the jets to penetrate. Underpenetration must also be avoided because it forms a relatively unmixed core that persists at downstream locations. Flow with *J*=38.5 is desirable, because it gives moderate penetration depth. The third picture of the same figure shows combustion with $\lambda = 1$ with higher temperature difference along the chamber length as compared to the first figure with the same value of *J*. But it also shows a similar jet penetration trajectory as the non-combustion case that has the same value of *J*. Thus, the difference between reactive flows and non-reactive flows by means of its influence on *h/R* is insignificant. It is also concluded that increases in the number of nozzles provide better mixing.

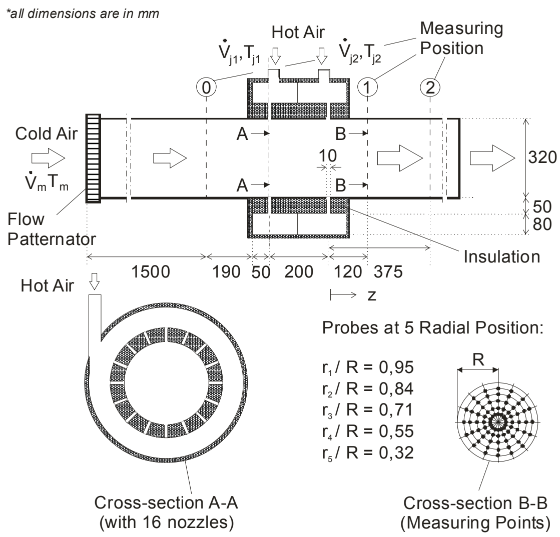


**Figure 3.** Schematic of experimental facility

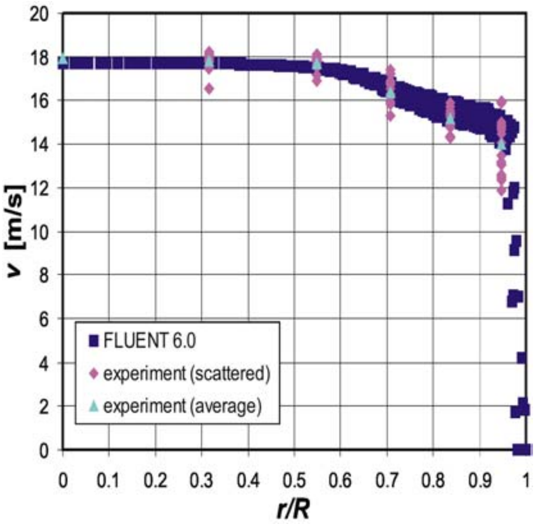


**Figure 4.** Typical Comparison Results

Comparison study was also done, but many effort will be needed for experiment, so this study concentrates mainly on CFD calculations. A schematic of the experimental facility from the laboratory at Lurgi Bischoff GmbH that were used is illustrated at **Figure 3**. A cold ambient air of about 20 $^oC$ at the amount around 1 $m^3/s$ was used as a main inlet, and 200 $m^3/s$ hot air with 385 $^oC$ were then injected into double rows of jets. Three measuring position both for velocities and temperatures were chosen along the cylindrical chamber. Local velocities were measured using Pitot tubes, and the local temperatures were measured using thermocouples. A typical of comparison result is shown at **Figure 4**. It is shown that the air velocity both for the experiment and CFD calculation give ($T-T_a$) in each radial position that were scattered. This is due to the highly turbulence flow and also different points in the second rows air injection.

**Table 1.** Comparison between CFD software packages

| CFD Software | Penetration Depth (h/R) |
|---|---|
| Phoenics 3.4 | 0.53 |
| CFX-TASC Flow 2.11 | 0.54 |
| CFX 5.5 | 0.57 |
| Fluent 6.0 | 0.51 |

Comparisons in determination of penetration depth between commercial CFD software were also conducted. A single row jets with cold air of 200 $^oC$ and 3.1 m/s was uses as the main input. The hot air of 1200 $^oC$ and 14.5 m/s was injected radialy through each 8 nozzles into a cylindrical chamber with diameter of 2 m. **Table 1** shows the penetration depth determination using some of the most popular commercial CFD softwares.

PUBLICATIONS & REFERENCES:


Present — Nirmolo, A., Woche, H., Praetor, R., Skroch, R., Specht, E., *Gas Mixing in Cylindrical Chamber after Radial Jets Injection*,
(In the process of submitting to an International CFD Journal)

2003 — Nirmolo, A., Woche, H., Specht, E., *Mixing of Combustible Gases in a Cylindrical Chamber with Radial Injection of Air*, Conference on Modelling Flow (CMFF'03) Proceeding, Budapest, Hungary, September 3-6, 2003

2002 — Tao, Y., Adler, W., and Specht, E., *Numerical Analysis of Multiple Jets Discharging into a Confined Cylindrical Crossflow,* Proc. Instn. Mech. Engrs., Vol. 216, Part E: J. Process Mechanical Engineering.

2000 — Nirmolo, A., Sediawan, B. W., *Eksperimen dan Pemodelan Pengeringan Gabah dengan Fixed-Bed Dryer* (*Experiment and Modelling of Grain Drying with a Fixed-Bed Dryer)*, 'Forum Teknik' Journal, Vol. 24., No. 1, Yogyakarta, Indonesia, 2000