ASYMPTOTIC CFD STUDY OF A PLENUM CHAMBER IN SWIRLING FLUID BED

Tammam S.Naji[1], Safiah Othman[1,*,] Vijay R.Raghavan[2]

1Universiti Tun Hussein Onn, MALAYSIA
2Universiti Teknologi Petronas, *MALAYSIA*

*Tammam_scop@yahoo.com

Abstract:

In the present study, the technique of combining asymptotics with computational fluid dynamics (CFD), called the asymptotic computational fluid dynamics (ACFD), has been used to obtain a correlation between pressure drop coefficient and the number of pipe inlet into the plenum chamber of a swirling fluidized bed based on CFD results obtained by Othman [2010]. Othman [2010] investigated the influence of various inlet types and hub designs as well as the introduction of multiple inlets on the aerodynamic behavior in the plenum chamber of a swirling fluidized bed. The final design chosen was the four radial offset (R/2) air inlets into the plenum chamber containing a full length cylindrical hub. This design offers a balance in the required performance characteristics, thus leads to optimum performance of the fluidized bed. The obtained correlation provide a full range of CFD results without actually carrying out the entire set of calculations. This powerful tool is still very rarely applied, especially in Malaysia. Therefore this study has provided a significant benchmark of ACFD study for fluid flow.

Keywords: asymptotic CFD ,plemun chamber swirling fluidized bed, pressure drop coefficient, geometrical configurations

1. Introduction

In mathematics, particularly in solving singularly perturbed differential equations, the method of matched asymptotic expansions is a common approach to find an accurate approximation to a problem's solution [1]. In a large class of singularly perturbed problems, the domain may be divided into two sub domains. On one of these, the solution is accurately approximated by an asymptotic series found by treating the problem as a regular perturbation. The other sub domain consists of one or more small areas in which that approximation is inaccurate, generally because the perturbation terms in the problem are not negligible there. These areas are referred to as transition layers, or boundary or interior layers depending on whether they occur at the domain boundary (as is the usual case in applications) or inside the domain. An approximation in the form of an asymptotic series is obtained in the transition layers by treating that part of the domain as a separate perturbation problem. This approximation is called the "inner solution," and the other is the "outer solution," named for their relationship to the transition layers. The outer and inner solutions are then combined through a process called "matching" in such a way that an approximate solution for the whole domain is obtained.

2. Methodolgy

In this work describes the numerical solution methodology in investigating the flow structure of the plenum chamber in a swirling fluidized bed. As the current study will propose a correlation for the CFD results obtained by simulation, the validated numerical methodology applied in the study [6] . for this study consists of two section . This major section includes five sub topics: computational domain, governing equations, boundary conditions, the finite volume method, and grid independence study. The second section elaborates the method of asymptotic expansion. The last part describes the planes chosen to determine the asymptotic computational fluid dynamics (ACFD)
.

2.1 Governing Equations

For the simulation, the following steady-state 3-D equations in Cartesian coordinates form have been solved numerically for a Newtonian, incompressible fluid [10,6]:

Continuity Equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{2.1}$$

Conservation of Momentum Equations in three dimensions

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$

$$\tag{2.2}$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$

$$\tag{2.3}$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial uw}{\partial z}\right) = \rho g_z - \frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right)$$

$$\tag{2.4}$$

2.2 Boundary Conditions

Prior to solving and post-processing the results, the boundary conditions and the fluid and solid properties had to be specified (figure 2.1). The air inlet was modeled as a velocity inlet boundary condition of 12 m/s (0.11062 m^3/s volume flow rate) [6].

$P_{out} = 1.01325$ bar (P_{atm})

No slip wall condition

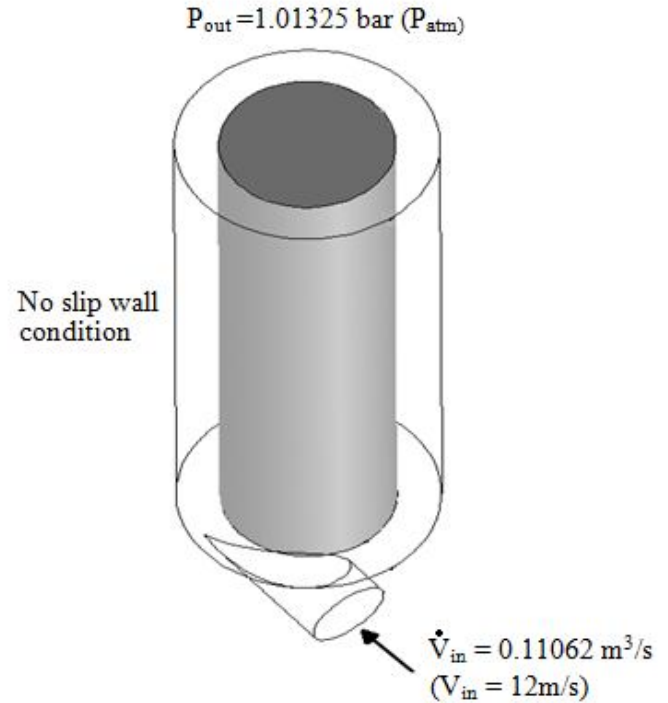$\dot{V}_{in} = 0.11062$ m^3/s
($V_{in} = 12$m/s)

Figure 2.1 : Boundary conditions [6]

The air outlet was modeled as a pressure outlet of 1.01325 bar (atmospheric pressure). In order to minimize convergence difficulties, realistic values for backflow quantities were entered. No slip wall conditions were applied. Air has been taken as the fluid domain, while the cylindrical hub taken as a solid medium.

2.3 The Finite Volume Method

Applying the finite volume method, the commercial CFD code FLUENT had been used to analyse the above flow characteristics by solving the governing equations for fluid flow (equations 2.1 to 2.4). This was affected by integrating the relevant equations over a finite control volume; discretising the resulting integral equation to obtain a system of linear algebraic equations for all control volumes in the calculation domain; and subsequently solving these equations once boundary conditions had been taken into account. Equations were taken from Versteeg and Malalasekera [10].

Simulations were performed using double precision solver in order to capture small gradients and minimize round-off error. Segregated implicit solver and Reynolds-Averaged Navier-Stokes (RANS) Equations Models with standard wall treatment were applied to simulate the turbulent flow in the chamber. A standard discretisation scheme was used for the continuity equation. To reduce numerical diffusion, a second-order upwind scheme was selected for the discretisation of the momentum equations, the turbulence kinetic energy equation and the turbulence dissipation rate equation. The SIMPLE algorithm was then applied to solve the pressure-velocity coupling algorithms. Default values for all under-relaxation factors were applied, except for the turbulence kinetic energy and the turbulence dissipation rate. In order to enhance convergence, the under-relaxation factors of these two turbulence quantities were lowered to a value of 0.6. The governing equations for flow; turbulence and energy were solved iteratively until convergence was obtained. The convergence criterion was set to 10^{-3} for all variables. A solution was considered converged (1) when the scaled residuals had dropped three orders of magnitude for all simulated variables and (2) when the conservation of overall mass balance through domain boundary exceeds 99 %. Typical compute times of about a day were consumed for each case (PC used was Intel Core 2 Duo 2.33GHz, 3.00 GB of RAM) [6]

3 . RESULTS AND CONCLUSION

The primary objective of this study is to obtain a correlation of pressure drop coefficient with Reynolds Number and the number of pipe inlet into the plenum chamber of a swirling fluidized bed by means of ACFD.

The ACFD method will be applied to the CFD results obtained by simulation to finally achieve the objective of study.

In this chapter we will discuss results of this work which are obtained from computational fluid dynamics (CFD) and asymptotic computational fluid dynamic (ACFD).

3-1Computational Fluid Dynamic (CFD)

The CFD works give the following results for two cases: In the first case we investigated the influence of four inlet velocities (and their corresponding Reynolds Numbers) on the pressure coefficient, C_p of the system. Figure 3.1 presents the computational domain under consideration. The model comprises a 20 cm long tangential entry inlet pipe of 10 cm diameter and a plenum chamber with 30 cm diameter

and 100 cm height, through which ambient air is directed from a blower into the particle bed via the air distributor. A 20 cm flow modifying center-body (hub) of 100 cm height is implanted at the center of the chamber. Consequently, the airflow is restricted within an annular path between the two cylinders.
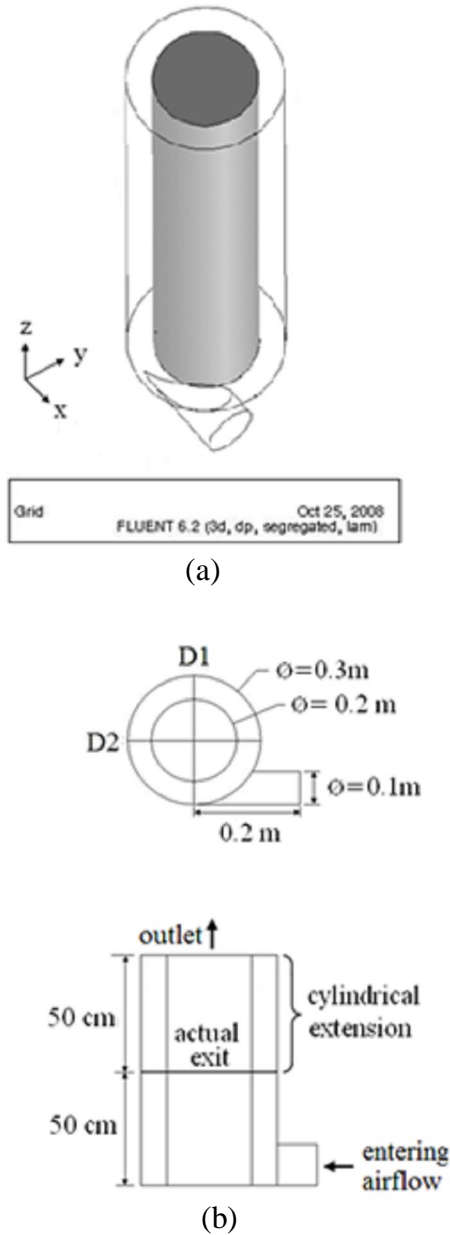

(a)

(b)

Figure 3.1 : Numerical domain; (a) 3-D; (b) plan and side view;

3.1.1 Various velocity at inlet

The first approach in the design is to choose the suitable inlet velocity boundary condition entering the plenum chamber. For that purpose, computations were carried out for four inlet velocities as listed in table 3.1. It is clear that the effects of fluid turbulence (Reynolds number ($\frac{\rho V D}{\mu} > 2300$)) have to be considered for all cases in the current work.

Table 3.1 : Four inlet velocities considered

V	Re
10	68 459
12	82 150
15	102 688
20	136 917

$$Cp = \Delta P/(0.5 \rho V^2) \qquad (3.1)$$

Where :

Cp = Pressure Coefficient

ρ = Density

ΔP = $P_{inlet} - P_{out}$ = Pressure Drop

V   =   Velocity at Inlet

From equation 3 .1 we get results on table 3 .2.

Table 3 . 2 : Computational pressure coefficient for various velocities at inlet

V [m/s]	ΔP [Pa]	Cp
10	7.91	0.142
12	11.08	0.138
15	18.27	0.145
20	28	0.125

3.1.2 Multiple inlets

In the second case the number of inlet is varied to obtain its influence to the pressure coefficient, C_p of the system. Figure 3.2 shows this variance.

(a)   (b)
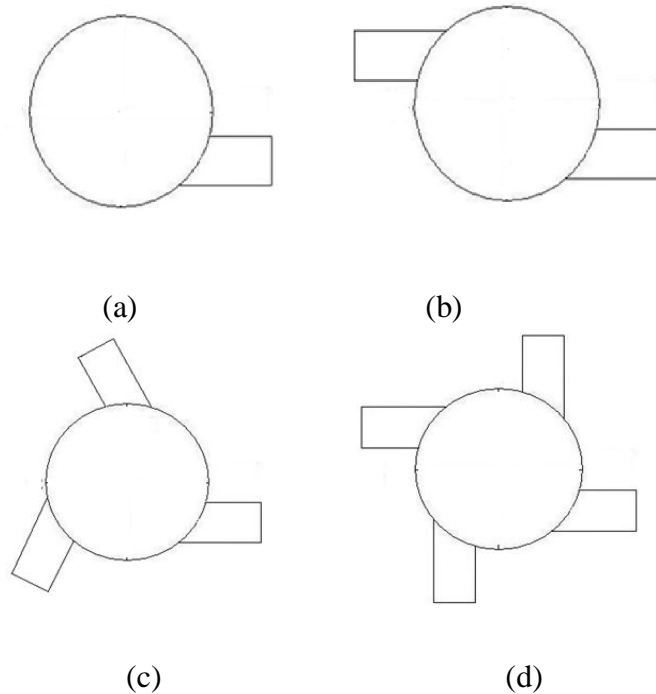

(c)   (d)

Figure 3.2 : Multiple inlet pipes into the plenum chamber; (a) single inlet; (b) two inlets; (c) three inlets; (d) four inlets

From CFD works we will get results as tabulated in table 3.3.

Table 3.3 : Computational Pressure coefficient for various number of inlets

Inlet type	Pressure drop ΔP [pa]	Cp_{averg}
One inlet	2.62	0.047
Two inlet	1.88	0.034
Three inlet	1.5	0.027
Four inlet	1.04	0.019

3-2 Asymptotic Computational Fluid Dynamics (ACFD)

3-2-1 Effect of Reynolds Number, Re

The equation 3.1 and data from simulations (CFD) as in table 3.2 are used to find relationship between Reynolds Number, Re and pressure coefficient, Cp.

In order to apply ACFD, the relationship between Re/Re$_{ref}$ and Cp need to be determined instead. In this case, Re $_{ref}$ =82150. However $(Re/Re_{ref})^{0.22}$ has to be considered in order to fit the line in the required scale. Table 3.4 presents the calculated data. These data then plotted in figure 3.3.

Table 3.4 : $(Re/Re_{ref})^{0.22}$ and the corresponding pressure coefficient for various velocities at inlet

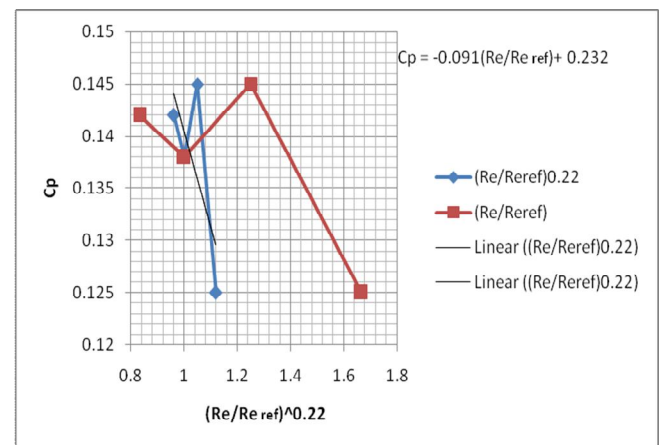| Re	Re/Re$_{ref}$	$(Re/Re_{ref})^{0.22}$	Cp
68 459	0.8333	0.9607	0.142
82 150	1	1	0.138
102 688	1.25	1.0503	0.145
136 917	1.6667	1.1189	0.125

Figure 3.3 : Pressure coefficient,Cp with respect to $(Re/Re_{ref})^{0.22}$

3-2-2-Effect of Multiple inlets, No$_{inlet}$

The equation 3.1 and data from simulations (CFD) as in table 3.3 are used to find relationship between multiple inlets, No$_{inlet}$ and pressure coefficient, Cp.
In order to apply ACFD, the relationship between No$_{inlet}$ / No$_{inlet ,ref}$ and Cp need to be determined instead. In this case, No$_{inlet, ref}$ =1. However (No$_{inlet}$ / No$_{inlet ,ref}$)$^{0.87}$ has to be considered in order to fit the line in the required scale. Table 3.5 presents the calculated data. These data then plotted in figure 3.4.

Table 3.5 : (No$_{inlet}$ / No$_{inlet ,ref}$)$^{0.87}$ and the corresponding pressure coefficient for various number of inlet

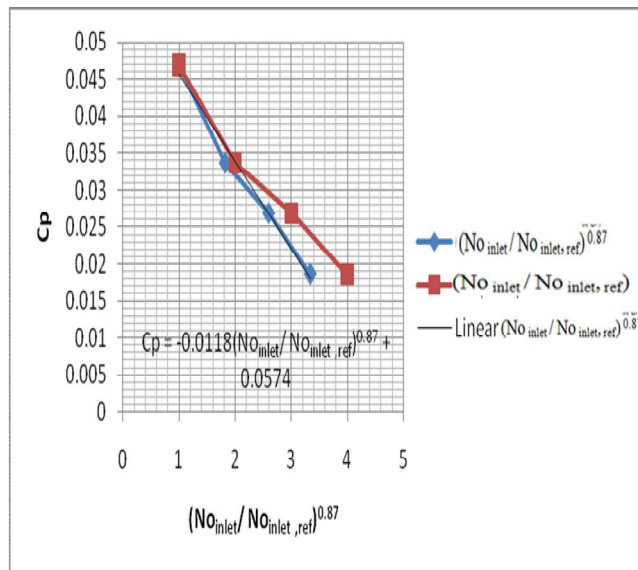| Inlet type	(No$_{inlet}$/ No$_{inlet ,ref}$)$^{0.87}$	Cp$_{averg}$
One inlet	1	0.047
Two inlet	1.828	0.034
Three inlet	2.601	0.027
Four inlet	3.340	0.019

Figure 3.4 : Pressure coefficient,Cp with respect to (No$_{inlet}$/ No$_{inlet ,ref}$)$^{0.87}$

Taylor's series expansion,

$$Cp_{max} = Cp_{max,ref} + (\phi_1 - \phi_{1,ref})\frac{\partial Cp_{max,ref}}{\partial \phi_1} + (\phi_2 - \phi_{2,ref})\frac{\partial Cp_{max,ref}}{\partial \phi_2}$$

(3.2)

Let Cp $_{max, ref}$ is the intersection point of curves Cp vs (Re/Re$_{ref}$)$^{0.22}$ and Cp vs (No$_{inlet}$/ No$_{inlet ,ref}$)$^{0.87}$.

$Cp_{max, ref}$ =0.0314

And the slope of linear line Cp vs (Re/Re$_{ref}$)$^{0.22}$,

$$\frac{\partial Cp_{max, ref}}{\partial \phi_1} = -0.091$$

While the slope of linear line Cp vs (No$_{inlet}$/ No$_{inlet ,ref}$)$^{0.87}$,

$$\frac{\partial Cp_{max, ref}}{\partial \phi_2} = -0.0118$$

From Taylor's series expansion (equation 3.2) and the above conditions the final correlation of pressure drop coefficient with Reynolds Number and the number of pipe inlet into the plenum chamber of a swirling fluidized bed has been obtained (equation 3.3).

Cp $_{max (correlation)}$ = Cp $_{max, ref}$ – 0.091 (Ø1 -1) – 0.0118 (Ø2 -1)   (3 . 3)

where

Cp $_{max, ref}$ = 0.0365
Ø$_1$=(Re/Re$_{ref}$)$^{0.22}$
Ø2=(No$_{inlet}$/ No$_{inlet ,ref}$)$^{0.87}$,

Then using the correlation (equation 3.3), the following Cp_{max} as tabulated in tables 3.6 and 3.7 are obtained. These data then plotted with respect to Cp_{max} obtained from simulations (tables 3.4 and 3.5) in figure 3.5. Finally well agreed correlation is of evident as all 8 points are distributed nearly to line of

Cp_{max}(correlation)=Cp_{max}(CFD). This correlation will provide a full range of CFD results without actually carrying out the entire set of calculations.

Table 3.6 : $(Re/Re_{ref})^{0.22}$ and the corresponding pressure coefficient, Cp_{max} for various velocities at inlet obtained from correlation

$(Re/Re_{ref})^{0.22}$	Cp (cfd)	Cp_{max} (correlation)
0.960685103	0.142	0.034977656
1	0.138	0.0314
1.050317661	0.145	0.026821093
1.118940408	0.125	0.020576423

Table 3.7 : $(No_{inlet} / No_{inlet\ ,ref})^{0.87}$ and the corresponding pressure coefficient, Cp_{max} for various velocities at inlet obtained from correlation

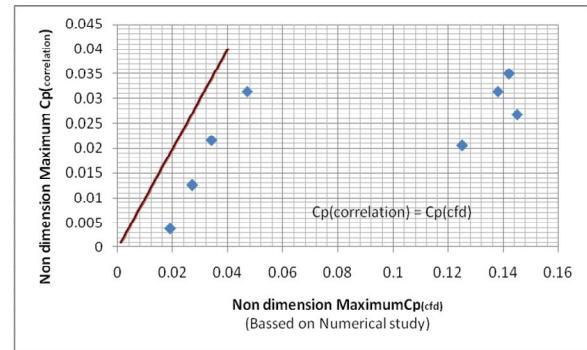| $(No_{inlet}/ No_{inlet\ ,ref})^{0.87}$,	Cp (cfd)	Cp_{max} (correlation)
1	0.047	0.0314
1.828	0.034	0.0216296
2.601	0.027	0.0125082
3.34	0.019	0.003788

Figure 3.6 : Non dimension Maximum Cp(Bassed on Correlation) and Non dimension Maximum Cp (Bassed on Numerical Study)

- However only 4 data for various inlet numbers correlates well. Another 4 data unfortunately are incoherent to the correlation.
- This must due to errors in the CFD simulations. This thus will be improved in the next study.

4 . Conclusion

The first objective of this study which is to carry out CFD works on various number of pipe inlets as well as Reynolds Number has been achieved. The second objective which is to obtain a correlation of pressure drop coefficient with Reynolds Number and the number of pipe inlet into the plenum chamber of a swirling fluidized bed by means of ACFD also has been addressed. The second part use the technique of combining asymptotics with computational fluid dynamics (CFD), called the asymptotic computational fluid dynamics (ACFD), to obtain a correlation of pressure drop coefficient with Reynolds Number and the number of pipe inlet into the plenum chamber of a swirling fluidized bed based on CFD results obtained by simulation. The obtained correlation provides a full range of

CFD results without actually carrying out the entire set of calculations

5 . REFFERENCES

[1] Verhulst, F. (2005). Methods and Applications of Singular Perturbations: Boundary Layers and Multiple Timescale Dynamics. Springer.

[2] Howard, J.R. (1989). "Fluidized bed technology – principles and applications." Bristol: Adam Hilger.

[3]Davies, R.M. and Taylor, G. (1950). "The mechanics of large bubbles rising through extended liquids and through liquids in tubes." Proc. R. Soc. A. 200. 375-390.

[4] Gabor, J.D. (1971). "Boundary effects on a bubble rising in a finite two-dimensional fluidized bed." Chem. Eng Sci. 26. 1247-1257.

[5] Sreenivasan, B. and Raghavan V.R. (2002). "Hydrodynamics of a swirling fluidised bed." Chemical Engineering and Processing 41. 99 – 106.

[6] Dr.Othman,S,(2010). "Hydrodynamics Study of plenum Chamber in a swirling fluidized bed"

[7] C. Balaji □, M. Hölling, H. Herwig.(2007). "A general methodology for treating mixed convection problems using asymptotic computational fluid dynamics (ACFD) " Hamburg University of Technology, Denickestrasse 17, D- 21073, Hamburg, Germany

 [8] Sathiyamoorthy, D. and Horio, M. (2003). "On the influence of aspect ratio and distributor in gas fluidized beds." Chem. Eng. J. 93. 2. 151 – 161.

[9] Depypere, F., Pieters, J.G. and Dewettinck, K. (2004). "CFD analysis of air distribution in fluidised bed equipment." Powder Technology 145. 176 – 189.

[10] Versteeg, H.K. and Malalasekera, W. (1995). "An introduction to computational fluid dynamics." Harlow: Pearson Education Limited.

[11] Thompson, J. F. (1985). "Numerical grid generation." New York: North-Holland.