A review on selection of turbulence model for CFD analysis of air flow within a cold storage

Pankaj Mishra[1,*] and K R Aharwal[2]

[1] Maulana Azad National Institute of Technology, Bhopal, M.P., India
[2] Maulana Azad National Institute of Technology, Bhopal, M.P., India

[*1] Corresponding Author: pankajhnmishra@yahoo.com

Abstract: Experimental analysis of flow Distribution inside a cold storage is a costly affair, thus many researchers are intensively using computational techniques. In designing of various air distributions arrangements in cold storage Computational Fluid Dynamics (CFD) can play a vital role. There are various factors which affects the results produced through CFD analysis of air circulation in close environment like cold storage. Selection of the specific turbulence model for particular flow condition is a big bottle-neck. This paper discussed CFD approaches and various turbulence models used in cold storage air flow evaluation. Selection of turbulence model affects the analysis as each model use different set of boundary conditions. It has been observed that most of the researchers adopt RANS K-Ɛ model because of its simplicity and ease of understanding. While on an average there is 26% error in results produced through RANS turbulence models and LES model provide good results but lots of skills and higher computational capacity required.

Key word: Computational Fluid Dynamic (CFD), Cold Storage, Air Circulation, RANS K-Ɛ model, LES model, Turbulence model

1. Introduction:

Cold storage is the place where perishable items are stored to preserve them for prolonged time under controlled conditions. For preserving perishable items in cold storage very low temperature conditions are required. Air distribution in cold storage is play a vital for proper production of cooling effect with the help of refrigeration system. Air circulation in cold storage depend upon various factors like mechanical axial blower fans, position evaporator coils, thermal buoyancy, etc. Flow mechanisms is combination of natural, artificial (forced or induced), and mixed convection. Combination of various flow produces multifaceted enclosed air flow characteristics with buoyancy effects, boundary layer separation at various surfaces, circulation, vortices, etc. Air flow regime inside the chamber vary from laminar to transitional, transitional further developed in turbulent flows or transient conditions i.e. combination of all the flow regimes. Experimental investigation of air flow in side cold room is extremely difficult and expensive affair, it is because of the complexity of airflow and size of cold storage facilities.

With the speedy advancement in computational capability and computing promptness, the Computational Fluid Dynamics (CFD) turn out to be a powerful tool for predicting fluid flow and its distribution in enclosed spaces. There are various approaches involve in CFD analysis, which based on fluid flow governing concepts.

CFD provides richer details and great flexibility at much lower cost than experimental setup. C R Boon reviewed and applied CFD to airflow simulation for enclosed room and achieved significant success, as he used numerical techniques for analysis of airflow and temperature distribution on an experimental setup [1]. H. Wang et al. (1990) used CFD for dynamic modeling of refrigerated room [2]. V. Yakhot (1986, 1992) use RNG analysis for turbulence flow field also work on development of turbulence models in shear flow conditions [3], [4]. Reynold's Average Navier Stocks (RANS) K –epsilon models with variety of boundary conditions were compare by Q. Chen (1995) for indoor air flow simulation [5]. A. Stamou et al. (2006) were worked on indoor air flow distribution and heat transfer with CFD simulation [6]. H. L. Choi (1990) use number of RANS models for air circulation with obstacles in airflow path in enclosed agricultural space [7]. P. S. Mirade et al (1998 and 2003) shows the use of computational fluid flow analysis in food processing industry [7], [8]. M. L. Hoang et al (2000) use CFD for air flow analysis in Cold store [9]. M. K. Chourasia (2001, $2007_{a,b,c}$) analyzed potato storage facility with various storing arrangements with the help of CFD tools and demonstrate air flow distribution , heat transfer and energy conservation for cold storage for batter productivity of system; CFD simulation and model selection for same ware validate with the experimental investigation [10], [11], [12], [13]. M. A. Delete et al. (2009), J. Xie et al (2006), N. J. Smale et al. and Toma's Norton (2007) were among those researchers who worked on cold storage cool air flow distribution and opting CFD as their supporting tool [14], [15], [16], [17]. Beside all above Ambaw et al. (2013), H. B. Nahor (2005) and Z. Zhai et al (2007) provide vital details for turbulence models and their applicability for enclosed environments [18], [19], [20]. Result produced with the help of CFD affected by many factors like geometry selection and its meshing, model selection etc. The precision and accuracy of the simulation depends on selection of turbulence model, as it is a crucial matter that affect the simulation.

2. CFD Approaches:

Different CFD modelling approaches have been proposed and used for study and evolution of air flow and heat transfer in a cold storage. Naiver Stokes equation is governing mathematical entity for describe energy momentum and mass transfer in any fluid flow problem. CFD simulate any fluid flow problem through resolve the partial differential equations which define flow analytically. CFD is proven tool for prediction of air flow (transport phenomenon i.e. flow velocity, temperature distribution, mass transfer etc.) inside cold store with consideration of various arrangements of storing and design of cold room [21].

By and large CFD predicts turbulent flow through either of any one approach out of following three: (1) Direct Numerical Simulation (DNS), (2) Large Eddy Simulation (LES) and (3) Reynolds- Averaged Naiver –Stokes (RANS) turbulence models. In DNS Naiver- Stokes equation is use without any approximation to compute turbulent flow. Precise fine grid resolution is essential for DNS to find out smallest eddy formation in the turbulent flow that's why large computer capacity desired [22]. Thus for large enclosed facility's like cold storage with agricultural stockpiling, simulation with DNS is not practically suitable, as there is high Reynolds number in flow field which needed high computational capacity.

LES is work on macroscopic structure of turbulent flow. For LES analysis turbulent flow can be separated into large eddies and small eddies and large eddies are considered in simulation. Exiting computing capabilities are enable to solve equations for large eddies corresponding to three dimensional and time dependent equations.

The Reynolds- Averaged Naiver –Stokes (RANS) is a custom with mean air flow parameters instead of using instantaneous flow parameters for simulating turbulent flow. RANS simulate turbulence in flow as a result of calculating average of both steady state and dynamic flows variables.

CFD modeling of cold store units required some specific features as there are large air flow obstacles by means of stockpiling of agricultural goods, air-conditioning systems affect room boundary condition, the evaporator unit can be treated as source of momentum and low air velocity ± 3m/s with turbulence [9].

2.1 Turbulence Modeling

Let’s considering the fluid flow over a horizontal thin plate, (Fig. 1). The fluid with uniform velocity as approaches the leading edge of the plate, laminar boundary layer development gets starts. In his region the flow is quite anticipated. Short distance from leading edge small disordered oscillations built and transition zone developed which further convert into fully develop turbulent zone.

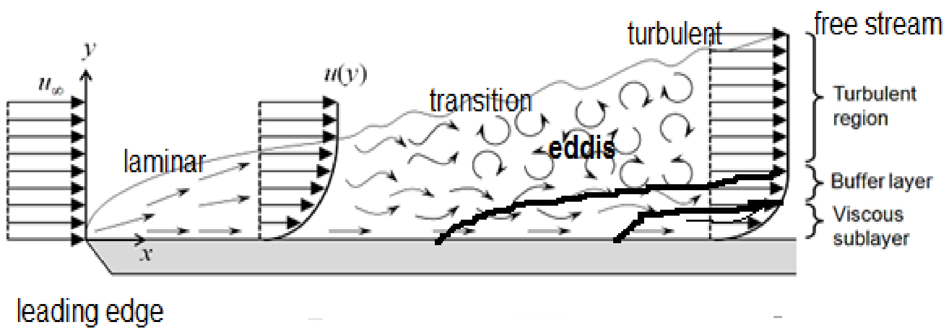

Figure 1. Fluid Flow over Thin Horizontal Plate (Source: www.comsol.com)

Reynolds number define the transition of flow between above discussed three regions of the flow over the flat plate. For steady state Naiver-Stokes equations predict the velocity and the pressure fields in laminar regime. To get an exact prediction of the flow behavior flow is considered steady and uniform. In laminar regime no averaging is needed as Reynolds number is remain constant.

As there is Reynolds number increases it cause the exhibitions of small eddies in flow and the too short spans of the oscillations happened, thus it is computationally impracticable to solve the Naiver-Stokes equations as value of Reynolds number is regularly varies with time and space. In such circumstance Reynolds-averaged Naiver-Stokes (RANS) formulation is more appropriate.

At near flat wall the turbulent flow can be divided up into four type’s wiz. viscous sublayer, or laminar sublayer, buffer layer, transition zone and free stream. At transition zone the in due course of time flow become completely turbulent and the average velocity of fluid flow is correlated to the log of the perpendicular distance from flat plate or fixed wall. Hence the region also termed as log-law region. In turbulence model thickness of laminar sub layer is too short thus approximation of this region is advantageous.

In the following part of paper various popular turbulence models are discusses which are used in cold store air circulations.

- **A) RANS Turbulence Models**
- **B) Large Eddy Simulation Turbulence Model (LES)**

C) Detached Eddy Simulation Turbulence Model (DES)

Further RANS turbulence models are divided in two types first Eddy Viscosity Models and other are Reynolds Stress Models.

A. RANS eddy viscosity model
   a) Zero equation models
   b) One Equation model
   c) Two Equation model
B. Reynolds Stress Model

2.2 RANS Eddy Viscosity Turbulence Models -

a) *Zero-equation eddy-viscosity models*:

Zero equation turbulence models are simplest eddy viscosity models. Prandtl (1925) had develop first known zero-equation model with the mixing-length hypothesis. Theoretically mixing-length model is not a complete one thus calibrations have to conduct for each specific type of flow is essential. It can provides satisfactory results for prediction of simple turbulent flow problems with low Reynolds numbers. With lots of obstacles in flow field make this model unreliable. In past no work marked on cold storage analysis with this model.

b) *One - equation eddy-viscosity models:*

Zero-equation models are not enough for simulation each types of flow as they are not considering the various physical characteristics like non local effect on turbulent eddy viscosity. It's also not considering flow-history to overcome these deficiency. In one-equation turbulence models turbulence variables (Turbulent Kinetic Energy, $k = \frac{1}{2}\overline{u_i^{'}}\,\overline{u_j^{'}}$) used to compute eddy viscosity v_t such as

$$v_t = Ck^{\frac{1}{2}}l \qquad \text{……… (1)}$$

Where on resolving transport equation kinetic energy (k) is obtained, turbulence length scale is denoted by l, and C denotes constant of coefficient. Similar to zero equation turbulent model it is required to set length scale for turbulent in this model. Results show that this model is not suitable for indoor environment as with obstacle in flow it is not able to simulate turbulent flow near corners. This model shows good results for zero separation of flow. These model are not suited for analysis of cold storage as these are not able to cover all transport phenomena which involve in flow inside chamber.

c) *Two- equations eddy- velocity models:*

In two equations eddy viscosity model, Reynolds transport equation is solve for kinetic energy as well as for dissipation rate of kinetic energy. This model comprise an additional partial differential transportation equation other than equation for turbulent kinetic energy 'k'. Hence two equations eddy viscosity models are can characterize more turbulence mechanism and also provide better visualization for flow. In the two equations models two variables are considered for solution of flow problem. Here first transportation variable is 'k' turbulent kinetic energy and second variable signify the rate of dissipation of kinetic energy. These models perform well for fluid flow around complex geometries, in open channels, enclose environment with uniform geometry conditions, etc. two equation models like k-epsilon or k -omega can predict the air flow around bluff bodies. Some two equations models which are in practice as listed here,

- k- epsilon model (k- Ɛ model)

- *RNG k*- epsilon model (RNG k- Ɛ model)
- *k*- omega model (k- ω model)

- ***k- Epsilon model (k- ε model)***

k-ε Model is the most popular turbulence model among other two equations models. Launder and Spalding (1974) developed the “standard” *k-ε* model, is one of the most established models for indoor as well as outdoor airflow simulation. *k-ε* Model is able to handle various fluid flow conditions. As it has simple format, strong performance, and also widely validate. In this model various improvisations has been conducted which results in better performance also make it more capable to handle various problem with simplicity. The turbulent eddy viscosity ‘v_t’ in *k-ε* model is find with the help of following relation,

$$v_t = C_\mu \frac{k^2}{\varepsilon} \qquad \text{……..(2)}$$

Where *k* representing kinetic energy, *ε* shows dissipation rate of kinetic energy in flow and C_μ is constant coefficient. Many researchers use Reynolds averaged Naiver-Stokes equation (*k-epsilon model)* to predict turbulent air flow pattern and temperature distribution in cold storage as it has good stability and broad applicability. Research exhibit that Coanda effect (separation and attachment of jet from celling or other surfaces) was not well predicted by k- *ε* model [23]. k- *ε* Model overestimate the Coanda effect thus prediction of separation of jet under adverse pressure gradient is not possible with this model. Studies suggest that results can be improve with changing the multiplier coefficient of the turbulent viscosity from $C_\mu = 0.09$ to $C_\mu = 0.12\ or\ 0.15.$ but no experimental evidence available for this [24]. It was predicted that there is 24.3% average relative error in velocity measurement with respect to experimental results [14].

Low Reynolds number (LRN) k- ε model was use for prediction of turbulent viscosity near walls and stacks but it required very fine grid thus needed high computation time for analysis [25].

M. L. Hoang et al., demonstrate a relative error of 26% for air flow velocity calculation thru *k- ε model* [9]. *k- ε* Model work on an eddy viscosity hypothesis, taking turbulence stress as additional viscous stress.

- ***Renormalization - Group (RNG) k- epsilon model (RNG k- ε model):***

In this model low scaled motion elements are removed systematically through the application of statistical tools. RNG *k – epsilon* model is acquire by a statistical methods, here the governing equation for fluid flow is modified with respect to large motion elements. This model of turbulence consider upper regime with low or zero effects of buffer region. In buffer modified viscosity and large scale motions are considered. Model represent strong anisotropy in turbulent flow through addition of new term in k-epsilon equation. RNG *k- ε* model is fails to predict effect of corners because constraint in use of coarse grid near wall and corners, [3], [4]. RNG K-ε model predict good agreement with large porous medium. With this model hot and cold spots inside the cold storage can be located easily in loaded condition [13]. In some cases RNG models produced better results than standard *k- ε* model while RNG model consider for complex anisotropic conditions [23]. This model predict good results for large space with low height obstacles for air flow like large office space [26].

- ***k* - Omega model (*k- ω* model):**

Omega (ω) is the ratio of ε (rate of dissipation of turbulent kinetic energy) over k (kinetic energy of turbulence). Omega represent the specific dissipation rate of turbulence kinetic energy (k) into internal energy (thermal). Omega shows scale of turbulence. In comparison to k- ε model the k – ω model is predicting better results in where adverse pressure flow conditions are prevailing [27]. Other *k* - **ω** models like Wilcox's *k* - **ω** model, Modified k - **ω** models and shear stress transport (SST) *k* – ω model were developed to analyze specific problems of flow as standard model is not enough for all conditions. (SST) *k* – ω model is use to predict flow near wall boundaries as its produce good result in such cases. (SST) *k* – ω model is equivalent to *k*- ε model in for various flow conditions and produce almost similar results.

2.3 Reynolds Stress Model:

For eddy viscosity model it is assume that turbulence structure is isotropic in nature. RANS eddy viscosity models are face lots of difficulties simulate the problem when heavy swirl and anisotropic conditions are prevailing and it result in poor prediction of flow behavior inside the cold storage chamber or close room. For such conditions eddy viscosity equations are replaced by some other flow transport equations to predict flow in heterogeneous environment. Reynolds Stress Model (RSM) solve transport equations of Reynolds stress and fluxes instead of eddy viscosity. As a result RSM enable to better handle the anisotropic condition then other RANS eddy viscosity model. Many higher order turbulence correlations remains unexplained during the derivation of Reynolds stresses transportation equations for RSM. Reynold stress model is best suited in the case when more than one source of air flow available inside the chamber or unsymmetrical cold room geometry. It's provide slightly better results than *k*- ε model but takes higher computing capacity and time. When RSM compared with K-epsilon model with low Reynolds number, Author found that RSM provide good results for separation of jets from the wall and general behavior of air flow pattern but required high coarse grid [28]. RSM provide good prediction with high coarse grid (238560 cell), it tends to high computational requirements with lots of time and skills.

2.4 Large Eddy Simulation (LES) & Detached Eddy Simulation (DES) Models:

LES required higher computer capacity and user skills. LES solve NS equation for large eddy with sub grid scale eddies filtering. Tian et al. compare LES, K-ε and RNG k-ε and for room environment for air circulation analysis and demonstrated that used all three CFD models predict good with respects to experimental setup. He also concluded that LES provide better result in comparison to RANS models and results are close to actual conditions of experimental setup. Because of its capability to solve complex flow problem and it also consider wall function for its calculation thus LES can be used to validate K -ε model [29].

Some researcher use DES model for complicated enclose air flow analysis but no significant role seen in study of cold storage air flow analysis DES model suitable for very high unsteadiness of flow as RANS model are not good to predict massive separation in free shear flows.

DES is performance as well as cost wise fall between LES and RANS models. For cold storage analysis where steady state conditions is desired in chamber and once flow is properly established then very few disturbing elements presents for such condition use of LES or DES is costly and time taking affair.

3. Concluding Remarks:

This review discussed various aspects and use of CFD models with reference to cold storage application. Although LES, DES & DNS are provide good results and best agreement with experimental values but required large computational capacity and time with lots of user skills. Most of the author prefer RANS model for simulation of cold storage. RNG K-ε and k-ε model provide results with approximately 24% error are easy to use for modeling, as these model required less user skill and low computing capacity. It is also surmised that LES can be used to validate RANS models. 3D geometrical models help in better visualization turbulence inside the cold storage in loaded conditions. Review suggest that no author in past explain basis of selection of particular CFD model for cold storage air flow analysis. This review conclude that RANS eddy viscosity models particularly two equation turbulence models with 3D geometrical modeling are capable to predicts air flow condition in well-structured cold storage chamber.

4. References

[1] Boon C R 1978 Airflow patterns and temperature distribution in an experimental piggery, *Journal of Agriculture Engineering & Research*. **23** 123

[2] Wang H, Touber S, 1990; Distributed dynamic modelling of a refrigerated room, *International Journal of Refrigeration.* **13** 214

[3] Yakhot V and Orszag S A 1986 Renormalization group analysis of turbulence, *Journal of Scientific Computing.* **1** 3

[4] Yakhot V, Orszag S A, Thangam S, Gatski T B, Speziale C G 1992 Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids.* **4(7),** 1510

[5] Chen Q 1995 Comparison of different k-ε models for indoor air flow computations, *Numerical Heat Transfer*. B **28**, 353

[6] Stamou A, Katsiris I 2006 Verification of a CFD model for indoor airflow and heat transfer, *Building and Environment.* **41,** 1171

[7] Mirade P S 2003 Prediction of the air velocity field in modern meat dryers using unsteady computational fluid dynamics (CFD) models, *Journal of Food Engineering.* **60** (**1**), 41

[8] Mirade P S and Daudin J D 1998 Numerical simulation and validation of the air velocity field in a meat chiller, *International Journal of Appl. Sci. Compute.r* **5 (1),** 11

[9] Hoang M L, Verboven P, De Baerdemaeker J & Nicolai B M 2000 Analysis of airflow in a cool store by means of computational fluid dynamics, *International Journal of Refrigeration.* **23 (2),** 127

[10] Chourasia M K & Goswami T K 2001 Losses of potatoes in cold storage vis-à-vis types, mechanism and influential factors. *Journal of Food Science and Technology*. **38(4),** 301

[11] Chourasia M K & Goswami T K 2007 (a) CFD simulation of effects of operating parameters and product on heat transfer and moisture loss in the stack of bagged potatoes, *Journal of Food Engineering.* **80,** 947

[12] Chourasia M K & Goswami T K 2007 (b) Three dimensional modeling on airflow, heat and mass transfer in partially impermeable enclosure containing agricultural produce during natural convective cooling, *Energy Conversion and Management.* **48,** 2136

[13] Chourasia M K & Goswami T K 2007 (c) Steady state CFD modelling of air flow, heat transfer and moisture loss in a commercial potato cold store, *International Journal of Refrigeration.* **30,** 672

[14] Delele M A, Schenk A, Ramon H, Nicola B M, Verboven P 2009 Evaluation of a chicory root cold store humidification system using computational fluid dynamics, *Journal of Food Engineering.* **94,** 110.

[15] Xie J, QuX H, shi J Y, Sun D W 2006 Effects of design parameters on flow and temperature fields of a cold store by CFD simulation, *J Food Eng*. **77,** 355.

[16] Smale N J, Moureh J, Cortella G 2006 A review of Numerical Models of refrigerated food applications, *Int. Journal of Refrigeration.* **29,** 911.

[17] Toma's Norton, Da-Wen Sun, Jim Grant, Richard Fallon, Vincent Dodd 2007 Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review ; *Bioresource Technology* **98** 2386

[18] Ambaw, Delele M A, Defraeye T, Ho Q T, Opara L U, Nicolai B M, Verboven P 2013 The use of CFD to characterize and design post-harvest storage facilities: Past, present and future; *Computers and Electronics in Agriculture*. **93,** 184.

[19] Nahor H B, Hoang M L, Verboven P, Baelmans M, Nicolai B M 2005 CFD model of the airflow, heat and mass transfer in cool Stores, *International Journal of Refrigeration.* **28,** 368.

[20] Zhai Z, Zhang Z, Zhang W, and Chen Q 2007 Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-1: summary of prevent turbulence models, *HVAC&R Research.* **13,** (6).

[21] Xu Y, Burfoot D 1999 Simulating the Bulk Storage of food stuff. *Journal of Food engineering.* **39,** 23.

[22] Nieuwstadt F T M, 1990; Direct and large-eddy simulation of free convection. *Proc. 9th Int. Heat Transfer Conf*. (Jerusalem) 37.

[23] Moureh J, Flick D, 2005; Airflow characteristics within a slot ventilated enclosure, *International Journal of Heat and Fluid Flow*. **26,** 12.

[24] Choi H L, Albright L D, Timmons M B, 1990; An application of the ke3 turbulence model to predict how a rectangular obstacle in a slot-ventilated enclosure affects air flow, *Transactions in Agriculture*. **33,** 274.

[25] Davidson L 1989 Ventilation by displacement in a three-dimensional room: a numerical study, Building and Environment, 24, 263-272.

[26] Zhang L, Chow T T, Wang Q, Fong K F, Chan L S 2005 Validation of CFD model for research into displacement ventilation. *Architectural Science Review*. **48**(4), 305.

[27] Huang P G, Bradshaw P and Coakley T J 1992 Assessment of closure coefficients for compressible flow turbulence models. *NASA TM-103882*.

[28] Moureh J, Menia N, Flick D 2002 Numerical and experimental study of airflow in a typical refrigerated truck configuration loaded with pallets, *Computer and Electronics in agriculture.* **34**, 25.

[29] Tian Z F, Tu Y U, Yeoh G H and Yuen R K K 2006 On the numerical study of contaminant particle concentration in indoor air flow, *Building & Environment.* **41(11),** 1504.