Sources of errors for indoor air CFD simulations (Part 1)

Kemal Gungor, M.AIRAH   Norman Disney & Young

ABSTRACT

Computational fluid dynamics (CFD) has emerged as a viable design tool in the industry over the past decade. CFD is a powerful technique for estimating air motions within buildings; however, the accuracy of CFD simulations strongly depends on the appropriate setting of boundary conditions and numerical simulation parameters. Thermal comfort and indoor air quality simulations are the most common CFD applications within the construction industry. Lack of any common guideline/protocol causes diverse settings for the CFD simulations. This also causes difference in simulation outcomes for indoor air CFD simulations. This study summarises the factors affecting the indoor air flow simulation and demonstrates the effect of using simple settings upon the results.

BRIEF HISTORY

Computational Fluid Dynamics, or simply CFD, solves and analyses fluid-flow and heat-transfer problems by using numerical methods and algorithms. One of the earliest applications of CFD was actualised by Lewis Fry Richardson. He attempted to calculate the weather for a single six-hour period starting from 7am on May 20,1910. Richardson divided a map of Europe into 200km squares from north to south. The six-hour forecast took him six weeks to calculate, and when he had finished, the forecast was drastically incorrect. Kumar and Philominathan [1] review the growth of CFD as a discipline and discuss its contemporary methodology in details.

Computational fluid dynamics has traditionally been one of the most demanding computational applications. Until the 1980s, CFD was performed using academic research and in-house codes. When one wanted to perform a CFD calculation, you had to write a program. The advent of high-speed and large-memory computers has enabled CFD to obtain solutions to many flow problems. CFD is now a standard part of the toolkit used both in scientific studies and engineering predictions.

In the past decades, CFD has been studied intensively as a tool for evaluating the indoor environment of buildings and its interaction with the building envelope, as well as for analysing the outdoor environment around buildings.

In the past few years, CFD has played an increasingly important role in building design, assessment of HVAC system performance, indoor air quality, air-change effectiveness, fire safety and thermal comfort. The CFD models account for 70% of the ventilation performance studies published in 2007 [2].

During the past few years, there has been a substantial increase of CFD simulations in the construction industry, predominately for indoor air quality and/or thermal comfort. There is anecdotal evidence that not all of the simulations are performed to the same standards/guidelines/detail/accuracy level. Trust and quality issues in CFD have become important, as it is widely used in design and analysis.

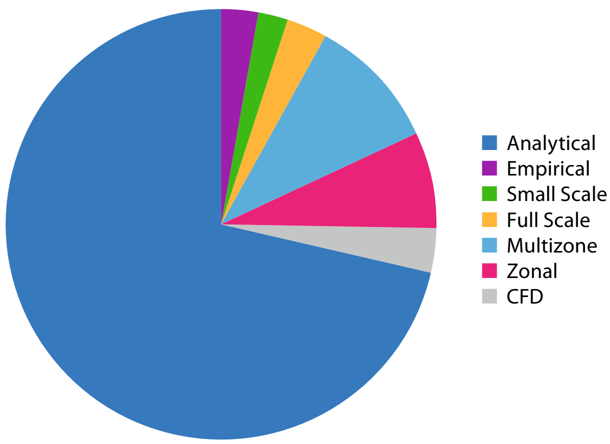

Figure 1: The share of different models in 2007 for predicting ventilation performance in buildings [2].

This study is an overview of the CFD simulation parameters affecting quality/accuracy of CFD analysis in general and more specifically the thermal comfort and air-change effectiveness analysis.

GENERAL UNCERTAINTIES AND ERRORS IN CFD SIMULATIONS

It is relatively easy to get results from commercially available CFD packages, but how do we ensure that the results make sense and can be trusted? In order to answer these questions, we need to have a good understanding of where errors/inaccuracies and uncertainties can arise in the modelling and solution process.

Uncertainty and error can be considered as the broad categories that are normally associated with loss of accuracy in modelling and simulation. American Institute of Aeronautics and Astronautics, AIAA, [3] provides useful definitions of error and uncertainty in CFD as follows:

Error: A recognisable deficiency in any phase or activity of modelling and simulation that is not due to lack of knowledge.

Uncertainty: A potential deficiency in any phase or activity of the modelling process that is due to lack of knowledge.

There is no universally excepted means of identifying or classifying errors, which can range from human or user errors to inadequacies in the modelling strategy and model equations. However, the European Research Community on Flow, Turbulence and Combustion, ERCOFTAC, Best Practice Guidelines [4], adopts the following classification based on seven different sources of error and uncertainty:

- **Model error and uncertainties:** The difference between the actual flow and the mathematical model we are using. Most well publicised errors in this category are the errors from turbulence modelling;
- **Discretisation or numerical error:** The difference between the exact solution of the mathematical model and numerical solution, with a limited resolution in time and space;
- **Iteration or convergence error:** Relating to the difference between the exact and iterative solutions of the discretised equations;
- **Round-off error:** This is caused by the limited number of computer digits available for storage of a given physical value;
- **Code errors:** Errors due to bugs in the software, unintended programming errors in the implementation of models or compiler errors;
- **Application uncertainties:** Inaccuracy is also introduced because the application is complex, and precise data needed for the simulation is not always available;
- **User errors:** Usage errors are due to the application of the code in a less-than-accurate or improper manner. Usage errors may actually show up as modelling and discretisation errors. Using incorrect parameter values, badly chosen model or boundary conditions are among the typical user errors. Usage errors can exist in the CAD geometry, grid generation, and post-processing software, in addition to the CFD set-up.

It would be impossible to adequately address all these factors in this short paper. Consequently we will mainly concentrate on variables and parameters whose effects would be considered under the last two categories: application uncertainties and user errors. This study will further focus on indoor ventilation CFD simulation and the common user errors, as well as demonstrate the effect of some of commonly made modelling simplifications.

EFFECT OF THE SOFTWARE PACKAGE

Before considering any other variables, CFD software selection will be briefly considered. When the CFD simulations are considered, selecting a CFD solver can sometimes be a difficult and daunting task. There is an overwhelming variety of choices. A list of common CFD packages, both commercial and public domain, can be found on cfd-online [5] and TenLinks [6].

Not every CFD software package is suitable for indoor air simulations. CFD packages should be capable of creating models that range from a momentum source scale to at least a building scale, if it is being used for a mechanically ventilated indoor environment simulation. This is directly linked to the meshing capability of the software chosen.

Meshing needs to be fine enough to capture flow characteristics of the air terminal units. Depending on the diffuser type and modelling technique used, down to 10mm or smaller mesh size may be required. Unless it is purely an academic exercise, it is obvious that this mesh size cannot be used for the whole simulation domain for a typical office building that is several hundred square meters, if not thousands.

For mechanically ventilated indoor simulations, CFD packages should be capable of generating grids that can capture the diffuser-flow characteristics and maintain the total number of grid elements within a limit. That limit should allow the solution to be delivered in a commercially acceptable time frame. Unstructured meshes allow greater freedom in providing fine resolution to one region, but having course resolution in other areas.

Generally speaking, non-conformal or unstructured meshing will capture flow characteristics with smaller grid numbers when compared to structured meshing. For a specific case, Shewchuk [26] states that "structured mesh has five times as many nodes, but yields the same accuracy in numerical simulation as the unstructured mesh"

USER ERRORS IN CFD SIMULATION

Computational fluid dynamics is a powerful technique for estimating fluid dynamics, but the classic computer science saying of "garbage in, garbage out" applies to CFD. User-friendly CFD packages make it relatively easy to set up a model and provide improvement to the productivity. However, at the same time, it has a potential to produce ColourFul Diagrams that have minimal resemblance to the modelled system behaviour. CFD user errors can be listed under four groups:

- CAD-related errors
- Errors due to improper grid generation
- Inappropriate setting of solver and boundary conditions
- Post-processing user errors.

CAD-RELATED ERRORS

An effective simulation starts with good CAD techniques, both in terms of model integrity and proper creation of the flow domain. Following is a list of potential CAD-related user errors:

- To save time and computer resources, it is common practice to simplify the model. Over-simplification can, however, alter the flow pattern within the domain.
- If the floor plate is too large for a CFD simulation, the floor can be subdivided into several CFD models. But extreme care is needed to determine the boundaries of each section. Maintaining an overlapping section between two adjacent sections is considered best practice.
- For external flow and wind comfort analysis, incorrect domain size is among the common user errors. Architectural Institute of Japan [7] and COST Action 732 [8] suggest that the blockage ratio should be below 3%, based on knowledge of wind tunnel experiments.
- Imposing symmetries that are not present in the real flow.

CAD-related user errors are among the easiest to recognise.

ERRORS DUE TO IMPROPER GRID GENERATION

The primary role of a mesh is to enable an accurate simulation to be performed on a computer. It is appropriate to consider mesh quality in terms of error analyses. Meshing quality is a direct result of the user input.

Following is a list of the most common grid errors caused by the users:

- Too coarse meshing. (Not capturing the curvature of solid within the critical area)
- High skewness. (Large skewness compromises the accuracy of the interpolated regions)
- Large aspect ratios. (Having a large aspect ratio can result in an interpolation error of unacceptable magnitude)
- Large jumps in volume between adjacent cells (Ideally, the maximum change in grid spacing should be <20%:)
- Inappropriate boundary layer mesh
- Inappropriate non-conformal meshing (Interpolation errors occur at non-conformal interfaces).

Poorly constructed grids can introduce significant discretisation errors and can lead to poor or very slow convergence.

Steve Owen [9] gives an overview about meshing algorithms and meshing techniques. Figure 2 shows typical meshing algorithms. Dimitri J. Mavriplis [10] from NASA Langley Research Center summarises the unstructured mesh-related issues and compares the CFD results by experimental results. His study demonstrates the effect of grid quality on results.

Depending on the grid generation techniques used and software package limitations, meshing choices vary widely. Meshing options for a simple circle can provide an idea about the possibilities for meshing a real model and its possible variations. Grid independence checks should always be performed for any CFD simulations. Improper meshing will have a significant impact on the solution, and lead to incorrect CFD simulation results.

INAPPROPRIATE SETTING OF SOLVER AND BOUNDARY CONDITIONS

The accuracy of CFD simulations strongly depends on the appropriate setting of the solver and boundary conditions. CFD solutions rely upon physical models of real-world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.) The CFD solutions can only be as accurate as the physical models on which they are based, and consequently, turbulence is one of the most intensively studied CFD fields.

Meshing Algorithms

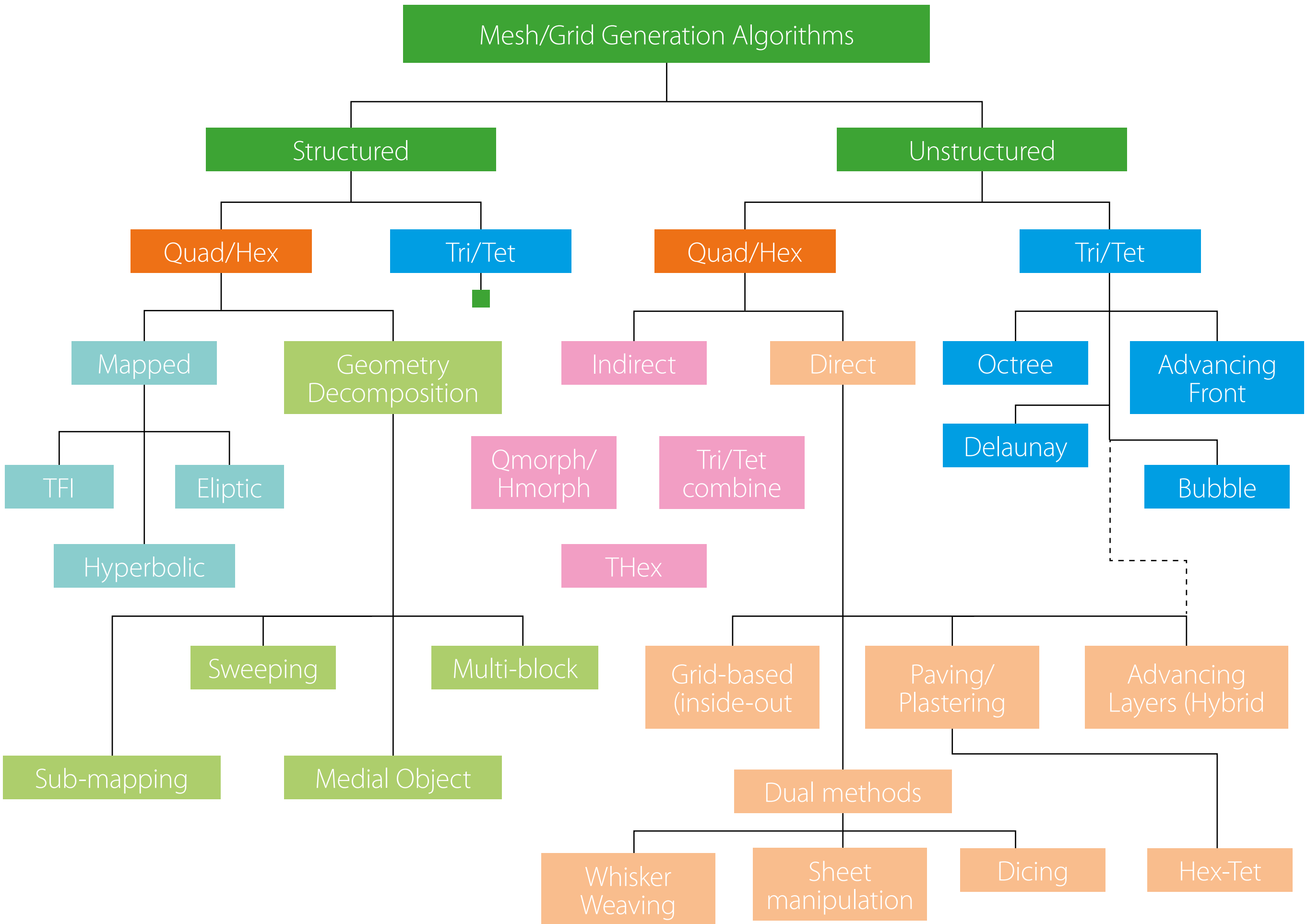

Figure 2: Typical meshing algorithms from [9].

Hundreds of turbulence models have been suggested in literature over the past 50 years. Nichols [12] provides the basic information about most of the currently available turbulence models, their limitations and application tips. Roy and Blottner [13] have conducted a review and assessment of turbulence models, concentrating on the hypersonic flows, and have included new experimental data since 1994. NASA Langley Research Center [14] has created a website that documents RANS turbulence models. The objective is to provide accurate and up-to-date information on RANS turbulence models as well, to verify that models are implemented correctly. The site will be updated regularly, as new models and/or verification/validation cases are incorporated and tested. NASA adopts the following classification of Turbulence models.

- One-equation models;
  - Spalart-Allmaras
  - Nut-92
- Two-equation models;
  - Menter k-omega SST
  - Menter k-omega BSL
  - Wilcox k-omega
  - Chien k-epsilon
  - K-kL
  - Explicit Algebraic Stress k-omega
- Three-equation models:
  - K-e-Rt
- Seven-equation omega-based full Reynolds stress models:
  - Wilcox Stress-omega
  - SSG/LRR
- Seven-equation Epsilon-based full Reynolds stress models:
  - GLVY Stress-epsiln

Picking the best turbulence model for a particular application is not a simple matter, if done inappropriately, erroneous simulation results will be obtained.

Inadequate set-up of boundary conditions are another source for the potential user errors. In principle there are four boundary conditions in CFD:

- Inlet/outlet
- Opening
- Wall
- Symmetry

Among the four boundary conditions, the most challenging one is setting the inlet/outlet boundary condition for an indoor ventilation CFD simulation. As there exists many different methods of how to describe the BCs for an air diffuser, it can be very confusing for the CFD user. There is no consensus as to how to model a supply air diffuser. Modelling the diffuser with simplified boundary conditions is a common approach. Such simplified diffuser models neglect the geometric details of the diffuser, and instead aim to capture and describe the velocity field and jet penetration. The box method and the momentum method are widely accepted, and use diffuser modelling methods.

With the momentum method, the supply diffuser is considered as a free opening. Accordingly, the actual discharge velocity is determined based on the airflow rate and the effective face area. In order to apply the momentum method correctly, diffuser discharge velocity or the diffuser effective area is needed, as well as the discharge flow direction. This information – especially the discharge direction – is not provided in every product catalogue.

The box method does not explicitly model the jet behaviour in the immediate vicinity of the supply diffuser, instead it specifies flow boundary conditions at the surfaces of an imaginary box around the diffuser. The box method requires to specify the distributions of air velocity, air temperature, and contaminant concentrations in the box surface through which the flow is discharged. This information may not be available in the product catalogues of diffuser manufacturers.

Srebric and Chen [15] demonstrate how to use simplified box and momentum methods to simulate diffusers in room airflow modelling by CFD. They propose a method to determine minimum box size. For the momentum method, Srebric and Chen recommend a procedure on how to determine the discharge velocity.

They also state that the flow direction can be estimated through smoke visualisation. In order to achieve valid/realistic CFD simulations, the diffuser boundary conditions should be calibrated against the product flow pattern.

Rusly and Piechowski [16] conclude their study by stating that "accurate calibration and representation of swirl diffusers with proper modelling technique incorporating diffuser performance data need to be implemented in CFD modelling".

Gery Einberg [17] conducted full-scale measurements to validate the diffuser models during his doctoral thesis. Figure 3 shows one of the validated diffuser boundary conditions.

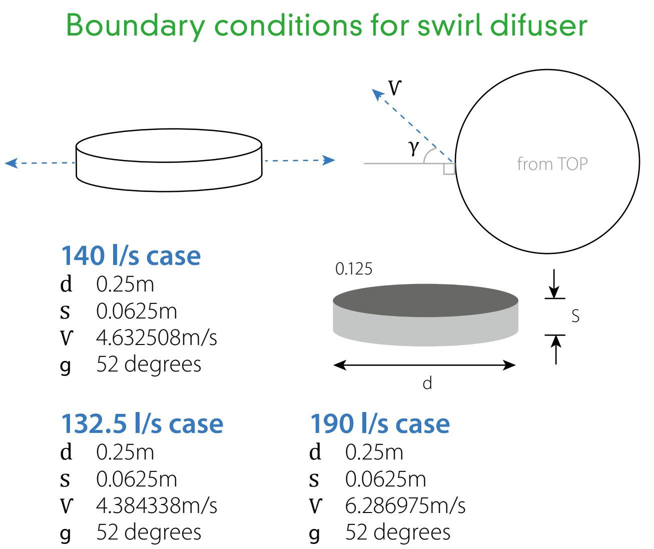

Figure 3: A sample of boundary condition set up for a swirl diffuser from [17].

Three CFD simulations were carried out in order to demonstrate the effect of the boundary condition settings. A floor swirl diffuser supplying 100 l/s air at 14°C was modelled in a 10mx10mx2.7m room. The same geometrical model with an identical mesh and solver set-up was used. Non-conformal meshing techniques have been used. A maximum 5mm mesh face size is used at the discharges surface, with a 5% growth rate. Maximum element size is 50mm within 2m radius of the diffuser.

The only difference between the three simulations was the angle between the axial and radial velocity components of the diffuser supply velocity, shown on Figure 4. As β angle changes up to 30 degrees, the flow pattern changes very little, then a small increase (1 degree in this example) of β, alters the flow pattern significantly. Entrained air influences the flow pattern and causes sudden changes. Any further increase of βangle causes small changes in the flow pattern.

Figure 5 shows the effect of varying βangle and changing flow patterns. Plan view vectors are taken 20mm above the diffuser surface. The βangles can be observed on section plots of the vectors.

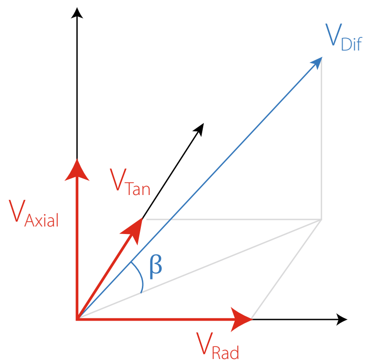

Figure 4: Diffuser supply velocity vector components.

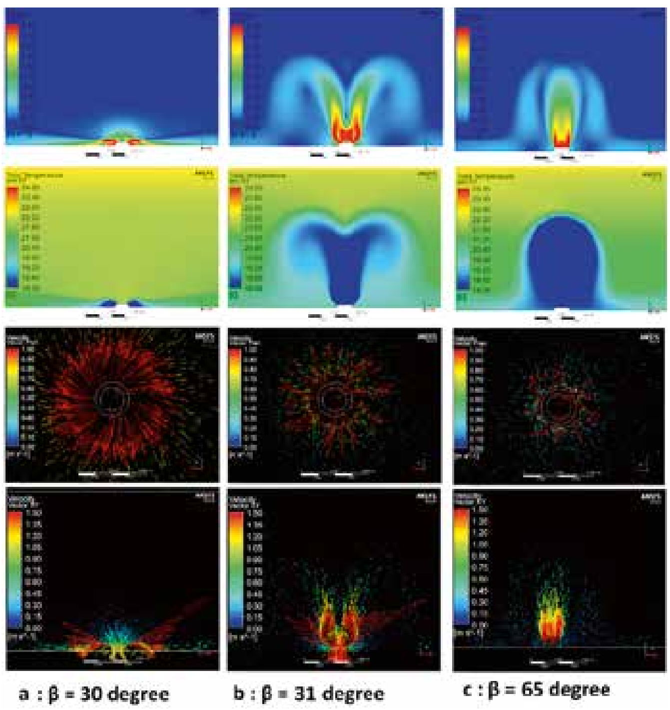

Figure 5: Velocity, temperature contours and vector plot of changing flow patterns.

Further flow patterns of some floor swirl diffusers can be found in Badenhorst's [18] work published in Ecolibrium. In order to have healthy CFD simulation results for indoor ventilation, the diffusers must be calibrated at the simulated flow rate to the manufacturer's experimental flow characteristics.

Wall boundary conditions set-up also has a significant effect on the outcome of the indoor CFD simulation. It would be a pragmatic and preferred approach to obtain the building fabric heat gains/losses from a building thermal simulation (assuming CFD package used does not provide such information). It is recommended that window solar gains should be applied to the perimeter zone floor area as a uniform heat flux. Window conduction and convection heat loads/losses should apply only to the glazing area.

POST-PROCESSING USER ERRORS

CFD simulations don't finish with obtaining the solver results. Benefits from a simulation requires post-processing of the results. Post-processing is the process to examine and analyse the flow field solutions, including contours, vectors, streamlines, iso-surfaces, animations, and CFD uncertainty analysis.

It is rare, but still possible to have post-processing user errors. Post-processing errors usually happen as a result of insufficient experience with the software package used for the CFD simulation. If the user of the CFD package is not fully familiar with the CFD package background and its terminology, it is very easy to make a mistake and plot erroneous results. A good example of this is highlighted and discussed by Leap Australia [19]. ▮

REFERENCES

1 Manickam Siva Kumar, Pichai Philominathan "Bringing out Fluids Experiments from Laboratory to in Silico – A Journey of Hundred Years", American Journal of Computational Mathematics, 2011, 1, 271–280

2 Qingyan Chen "Ventilation performance prediction for buildings: A method overview and recent applications", Building and Environment 2009, 44(4), 848–858.

3 "Guide for the Verification and Validation of Computational Fluid Dynamics Simulations", American Institute of Aeronautics and Astronautics, AIAA G-077-1998,

4 Casey, M., and Wintergerste, T., (ed.), European Research Community on Flow, Turbulance and Cumbastion, ERCOFTAC, Special Interest Group on Quality and Trust in Industrial CFD "Best Practice Guidelines", Version 1.0, January 2000

5 CFD Online, http://www.cfd-online.com/Wiki/Codes

6 TenLinks, http://www.tenlinks.com/cae/products/cfd.HTM

7 Yoshihide Tominaga et al, "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings", Journal of Wind Engineering and Industrial Aerodynamics 96 (2008) 1749–1761

8 Jörg Franke, Antti Hellsten, Heinke Schlünzen, Bertrand Carissimo "Best Practise Guideline For the CFD Simulation of flows in the Urban Environment" COST Action 732, 1 May 2007

9 Steve Owen, "An Introduction to Mesh Generation Algorithms", 14th International Meshing Roundtable presentation, San Diego, California, USA, September 11–14, 2005

10 Dimitri J. Mavriplis, NASA Langley Research Center, "Unstructured Mesh Related Issues In Computational Fluid Dynamics, (CFD) – Based Analysis And Design", 11th International Meshing Roundtable, Ithaca New York, USA, September 15–18, 2002,

11 Stanford University, ME469B/2/GI "Geometry Modeling & Grid Generation" hand-out, http://web.stanford.edu/class/me469b/handouts/geoandgrid.pdf , last accessed, 3/3/2015

12 R. H. Nichols, "Turbulence Models and Their Application to Complex Flows", University of Alabama at Birmingham, Revision 4.01

13 Christopher J. Roy and Frederick G. Blottner, "Review and Assessment of Turbulence Models for Hypersonic Flows: 2D/Axisymmetric Cases", 44th AIAA Aerospace Sciences Meeting and Exhibit, 9¬–12 January 2006, Reno, Nevada

14 NASA, Langley Research Centre, Turbulence Modeling Resource, http://turbmodels.larc.nasa.gov/index.html last accesses on 7/3/2015.

15 Srebric, J. and Chen, Q. 2001. "A method of test to obtain diffuser data for CFD modeling of room airflow," ASHRAE Transactions, 107(2), 108–116.

16 Eddy Rusly and Mirek Piechowski, "CFD Modelling For Swirl Diffuser and its Implications on Air Change Effectiveness Assessment to Green Star's IEQ-2", Proceedings of Building Simulation 2011: 12th Conference of International Building Performance Simulation Association, Sydney, 14–16 November.

17 Gery Einberg "Air Diffusion and Solid Contaminant Behaviour in Room Ventilation – a CFD Based Integrated Approach", Doctoral Thesis, Kungliga Tekniska Högskolan, June 2005, ISBN 91-7178-037-8

18 Sean Badenhorst, "Floor swirl diffusers: types and applications", Ecolibrium, 40–50, October 2013,

19 "Tips & Tricks: How to interpret results for multiphase & porous domains using true velocity and superficial velocity", Posted By LEAP CFD Team on Nov 22, 2012, –www.computationalfluiddynamics.com.au/interpreting-results-for-superficial-and-true-velocity/, last accessed on 16/3/2015.

20 Pieter G. Buning, Reynaldo J. Gomez, "20+ Years of Chimera Grid Development for the Space Shuttle", 10th Symposium on Overset Composite Grid and Solution Technology September 20–23, 2010, Moffett Field, CA

21 W. L. Oberkampf and T. G. Trucano, "Validation Methodology in Computational Fluid Dynamics", American Institute of Aeronautics and Astronautics, AIAA 2000-2549, Fluids 2000, 19–22 June 2000, Denver, CO

22 www.grc.nasa.gov/WWW/wind/valid/tutorial/bibliog.html, last accesses 25/3/2015

23 Nuclear Energy Agency and The Committee on the Safety of Nuclear Installations (CSNI) "Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications", NEA/CSNI/R(2014)11, February 2015

24 ASME, "Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer", V V 20 – 2009.

25 Kemal Gungor, "Guide t o air change effectiveness", Ecolibrium, March 2013.

26 J. Shewchuk, "Lecture Notes on Delaunay Mesh Generation", The University of Iowa, 58:110 Computer-Aided Engineering, Lectures 12 and 13: Mesh Generation Notes, 1999

ABOUT THE AUTHOR

Kemal Gungor (PhD), M.AIRAH, is an ESD consultant at Norman Disney & Young. Kemal's expertise lies in heat transfer, thermodynamic analysis, design and computer modelling of thermal systems (i.e. energy modelling, facade analysis). He is also specialised in computational fluid dynamics analysis. These skills are being utilised to search for, develop and implement ecologically sustainable design solutions within the building industry.