WIND LOAD PREDICTING; HOW COULD CFD REPLACED WIND TUNNEL TEST

I Gede Adi Susila
Lecturer of Civil Engineering Department, Faculty of Engineering, Udayana University, Denpasar
E-mail: adiari2004@yahoo.com

Abstract: Computational Fluidal Dynamic (CFD analysis) is highly pointed to solve a number of wind tunnel test problem on the computer simulation. Large-eddy simulation (LES) technique with the Smagorinsky eddy-viscosity model has been applied in order to predict pressure coefficients for 3-D domes and catenoid models.

"Fluent" has been used to analyze the flows. Published data of Maher and the ASCE have been used as the basis guideline to enable wind loading to be applied appropriately. Result of LES computations are compared with those from laminar models as well as those from turbulent models based on Reynolds–average Navier-Stokes equation (RANS model) and those from experiment. The numerical experiment results for all models with various configurations to be exited by the turbulent wind forces were identified. The LES results from 3D computational agreed very well with the experimental or published data. For the dome case of h/d=1/2 ratio, the result can be sort it out into the maximum positive Cp=+0.621 and the maximum negative in the centre of dome is Cp = -1.2. The coefficient offered was quite similar to the published data of Cp=+0.6 and Cp=-1.0, respectively.

CFD has been shown to a reasonable prediction of wind pressure distributions. It conceivably could replace some wind tunnel tests.

Keywords: wind loads, wind tunnel test, published data, CFD method, LES (Large Eddy Simulation) model.

MEMPREDIKSI BEBAN ANGIN; BAGAIMANA CFD MAMPU MENGGANTIKAN TES TEROWONGAN ANGIN

Abstrak : Analisa komputasi fluida dinamis; 'Computational fluid dynamics' (CFD) yang difokuskan untuk memecahkan sejumlah permasalahan tes terowongan angin dalam program computer. Secara teknis 'Large-eddy simulation' (LES) dengan model viscositas Smagorinsky-Eddy diterapkan untuk memprediksi koefisien tekanan angin pada model dome (bangunan setengah lingkaran) dan katenoid (banyak lengkungan) 3 dimensi.

Paket program computer 'Fluent' digunakan untuk menganalisa aliran/hempasan angin tersebut. Sementara data yang sudah di publikasi oleh ASCE menjadi dasar acuan untuk mendapatkan beban angin yang kemudian disesuaikan. Hasil simulasi komputer diperbandingkan antara model aliran Laminar maupun turbulensi dengan dasar model persamaan angka rata-rata Reynolds–Navier-Stokes equation (RANS) dan LES dan perbandingan hasil antara komputerisasi dengan hasil dari percobaan lab (tes terowongan angin). Hasil analisa numerik dari semua model sudah diidentifikasi dengan berbagai konfigurasi yang di hasilkan dari beban angin berupa turbulensi. Hasil dengan simulasi LES dengan model 3 dimensi menunjukkan hasil yang mendekati dan sesuai antara analisa model computer dengan hasil penelitian yang sudah dipublikasikan. Untuk kasus setengah lingkaran dimana rasio h/d=1/2, hasilnya terdiri dari koefisien maksimum tekanan angin positif Cp=+0.621 dan maksimum negatifnya pada pusat/puncak dome sebesar Cp

= -1.2. Koefisien yang dihasilkan sangat sesuai/mirip dengan data yang sudah dipublikasi yakni Cp=+0.6 dan Cp=-1.0.

CFD telah menunjukkan hasil prediksi yang baik dan sangat rasional untuk dapat memprediksi distribusi tekanan angin pada bangunan/struktur. Hal ini sangat dapat dipercaya bahwa CFD bisa menggantikan peran tes terowongan angin.

Kata kunci: beban angin, tes terowongan angin, data publik, metode CFD, LES model.

INTRODUCTION

The large surface area usually projectted by a fabric membrane structure means that wind pressure is a significant load case. It also could be strongly influenced by the basic structural form of the roof. However, a problem exist is the requirement of prediction of the applied loading to membrane roof structures in order to estimate wind loading to complex geometric form. Industry has provided data of CFD (Computational Fluidal Dynamic) and relatively small number of wind tunnel test form the basis of an approach to enable wind loading to be applied appropriately.

The aim of wind tunnel test and CFD is to provide information on local wind patterns, coefficient of wind pressure, and wind-induced structural vibration. The use of wind tunnels is to determine the response of a structure to wind forces and to determine the pattern of wind flow to leeward of a structure. Investigations are carried out on the eddy formation behind model membrane structures to find the frequency and strength of oscillatory forces on the structure of a turbulent air-stream, and on the simulation of natural boundary layer effect, (Fig.1).

The objective of the present research is to develop procedures for accurate and efficient analysis, particularly to estimate wind loading on non-conventional structures and complex geometry with criteria to the design of tent structures or cable suspended roofs. In this study testing and CFD analysis of scale models are used to obtain a better understanding of how these structures behave under wind loading conditions.

A shape of wind tunnel model was also created on Gambit, which the model structure generation was placed in the middle of tunnel model. The mesh generation resulted by Gambit will then exported to the Fluent. The Fluent as a solver then associated a particularly need in which the model appropriately generated.

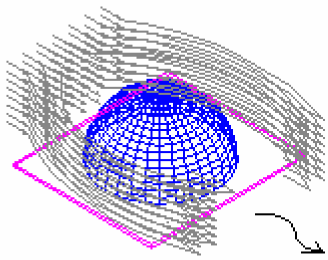


Figure 1. Illustration of wind acting on fabric structure

Several model structures were used in this particular cases such as Sphere, Cooling Tower, China hat model and tandem combination on each model in CFD. The same model was tried to involve in wind tunnel too. In wind tunnel model, there were used lamp shade, small ball, and bowl and fiber-glass resin as basic material to perform the shape model of fabric membrane structure. All models measured approximately in 150mm x 150mm each of plan area. The shape of structures model were constructed look like dome/sphere and China hat model represented cooling tower model. All of them were investigated under low-speed wind tunnel testing.

When the airflow approaches a building, it is impinged around and over the surface of building. The force will create areas of pressure or suction on part of building such as facades, gables and roof. In this study, the material building is fabric membrane structure and cable suspended so that leads to the extraordinary buildings geometric developed. It is the significant requirement to evaluate wind loading by CFD method or wind tunnel test. Once model structure has been examined completely, the result can be combined to the standard method of wind loads in order to analysis a structure related. The important result expected is local mean pressure coefficient *Cp*, where ($Cp = \frac{p}{\frac{1}{2}\rho V^2}$) known as dimensionless pressure, it will then combined with the area pressure coefficient. The area pressure coefficient is integration the local mean pressure coefficient over a surface area such as the roof, gable, etc or part of building faced the wind and then divided by the area yields the area pressure coefficient, which can be used conveniently for determining the wind loads on specific area of building. The pressure coefficient is being indicated by a positive Cp value and a negative value (suction pressure). The pressure counted at any point on the surface is also a fraction of the dynamic pressure (qs). Relationship to the building design is when value of dynamic pressure (qs) combined with pressure coefficient whether it is external or internal. Since the combination between dynamic pressure and coefficient pressure occurred, the wind load can be obtained and then can be applied to the building design.

Letchford and Sarkar (2000) performed wind tunnel test on rough and smooth parabolic domes, which simultaneous pressure measurement involved simulation atmospheric boundary layer flow. Mean and fluctuation pressure distribution have compared with earlier studies for similar shape and Reynolds number. A complication of wind tunnel model studies of these types of structures is because the curved surface, which leads to Reynolds number effects.

Reynolds numbers defined as $\rho\frac{DU}{\mu}$ where ρ and μ are fluid properties, D is the base diameter and U is the mean velocity at the top of model height. The Reynolds numbers were used in the range of each in dealt with Maher used R in the range of 6 x 105- 18x 105, Taylor dealt with R in the range from $1x10^5$ to 3×10^5, and Ogawa also investigated a range of turbulent intensities with Reynolds numbers ranging from 1.2×10^5 to 2.1×10^5. The wind tunnel used is closed circuit, 1.8 m wide with a ceiling adjustable to ~1.8m. There is an upstream fetch of approximately 15m for developing appropriate simulations of the earth's atmospheric boundary layers. A model dome was constructed with a base diameter (D) of 480 mm and height (h) of 150mm. The research estimated pressure coefficient, which all pressure was stated as non-dimensional by the mean of dynamic pressure (1/2 ρ U2) at the top of the dome. Δp is the instantaneous pressure difference between the surface pressure and a reference pressure in the wind tunnel.

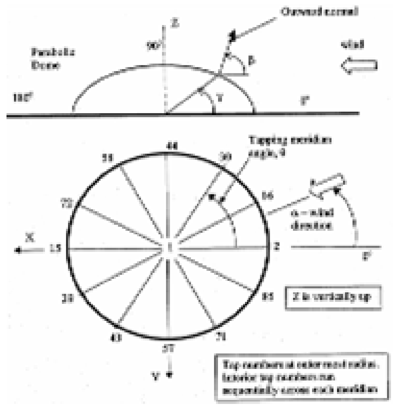


Figure 2. Tapping arrangement and wind di-rection definition for single dome test, by Letchford,cs

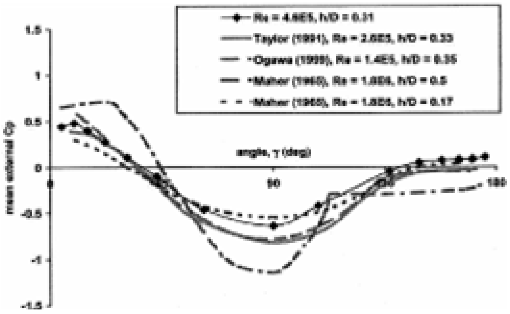


Figure 3. Comparison of mean pressure coef-ficient along centreline of a smooth dome, by Letchford,cs

They governed equation were as follows:

$$\overline{Cp} = \frac{\overline{\Delta p}}{\frac{1}{2}\rho \overline{U}^2}$$ Mean pressure coefficient, ……………………...1

$$Cp_{rms} = \frac{\Delta p_{rms}}{\frac{1}{2}\rho \overline{U}^2}$$ Standard deviation pressure coefficient, ……………….……..2

$$\hat{C}p = \frac{\overline{\Delta \hat{p}}}{\frac{1}{2}\rho \overline{U}^2}$$ Mean peak maximum pressure coefficient, ………………………3

$$\breve{C}p = \frac{\overline{\Delta \breve{p}}}{\frac{1}{2}\rho \overline{U}^2}$$ Mean peak minimum pressure coefficient, ………………………4

$$G = \frac{\hat{U}_3}{\overline{\overline{U}}_{1h}}$$ Gust velocity ratio ………………………5

The corresponding coefficients are

$$\overline{C\hat{p}} = \frac{\hat{C}p}{G^2}$$ Mean maximum pseudo-steady pressure coefficient, ……………………….6

$$\overline{C\breve{p}} = \frac{\breve{C}p}{G^2}$$ Mean minimum pseudo-steady pressure coefficient. ……………………… 7

They specified conclusion that mean and fluctuating pressure distributions are well approximated by a spherical dome of the same height to diameter ratio. The pressure distributions were independent of Reynolds number in the range 2.3 x 105-4.6 x 105 defined by velocity at top of dome and base diameter. In this study, hemisphere is one of the structural geometry on fabric membrane structures under wind force acting investigated. Regarding to the previous research, the result can be referenced to further study to the similar shape geometry with variation on material structure involved.

Published data by Maher's classical study of the dome surface will be compared with the CFD result. Those tables and diagrams involved are based on the work of Maher in 1965 and some by Blessmann in 1971, both of the result of them that arrived in general conclusions. Mean and fluctuation pressure distribution will have compared with those earlier studies.

Experimental Methods

Four models were constructed, all model assumed at a scale of 1:1000 proposed prototypical roof structure. The models represented variations of structures that consist of cable–suspended structure that support fabric membranes. The models structure developed are single sphere, multiple sphere, single cooling tower, and multiple cooling tower. The single sphere structure represented by the model were D=150 m diameter, and H=75 m high, it can be seen in figure 4a-b. The multiple sphere structure represented by the model was 250 m long span, and 75 m high, (figure 5a-b). The single cooling tower structure represented by the model were D1= 205 m diameter (bottom), D2=105 m diameter (top), and H =160 m high, (figure 6.a-b) The multiple cooling tower structure represented by model was 220 m long span, D2=70m of each top diameter, and 130m high, (figure 7.a-b)

Figure 5a. 1:1000 Scale Model of Single Sphere

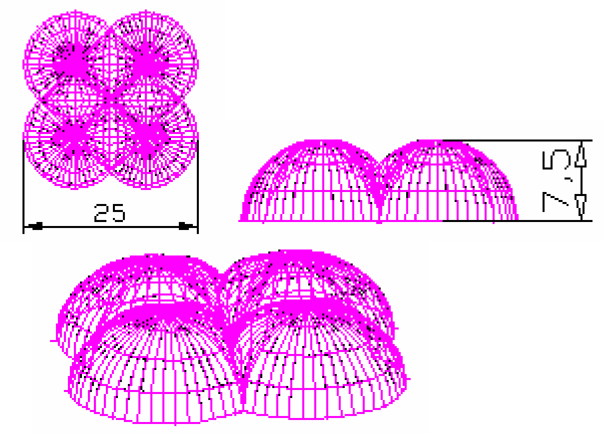


Figure 5b. Sketch Model of Multiple Sphere

Figure 6a. 1:1000 Scale Model of Single Cooling Tower

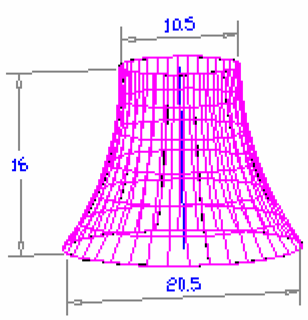


Figure 6b. Sketch Model of Single Cooling Tower

Figure 7a. 1:1000 Scale Model of Multiple Cooling Tower

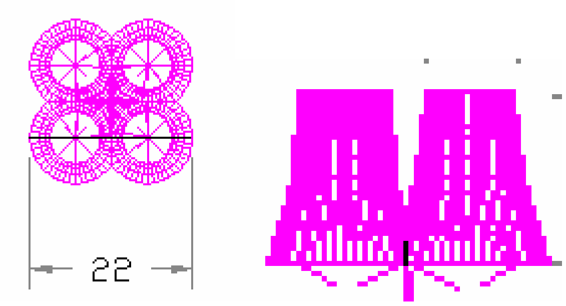


Figure 7b. Sketch Model of Multiple Cooling Tower

The structures of each shape represented by the models were created with rib around the surface that was divided into (8) eight parts of both sphere models and (6) six parts of all cooling tower models.

Experimental Work Procedure 1:1000 Scale Model of Cooling Tower and Sphere Model

The dimension of the 1:1000 scale models was constructed using curvatures orientations as represented structure.

Materials & Construction

Surface of material used is not practically represented in construction of the fabric membrane structures, it is the technical requirement that was needed more hard surface in order to put the pressure tap easily. The curve has already performed by the lamp cap that was made of wire framed to perform shape and covered by material coated. In order to perform harder surface, fibreglass was involved to make no flexible bending on the surface.

The single cooling tower model was formed by hand with fibreglass inside of the put lamp. Pieces by pieces of dry fibre arranged inside the lamp or to be put under the cover of lamp. It is needed a final touch to make a clean and smooth surfaces in order to give an opportunity of the pressure tap fixed. By the authority of wind tunnel test, the model required kind like a hard and smooth surface that is mean to anticipate the possibility of unexpecting result appeared and have an opportunity to deal with their experienced to undertake the model development problem.

Wind Tunnel Testing and Requirement

The wind tunnel planed to use for the 1:1000 scale model of wind loading was small wind tunnel. The taxonomy of wind tunnel introduced in order to know various classification of wind tunnel. In this particular case, according to flow circuit, the small wind tunnel may be classified either as an open-circuit. It is because the wind tunnel machine shape is straight structure. It can be described that the air is drawn into the tunnel from a funnel-shaped intake at one end of the tunnel, while the air exits the tunnel through a funnel-shaped outlet. Sketch of the tunnel can be described in Figure 8.

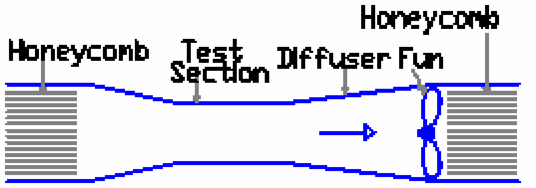


Figure 8. Typical open-circuit wind tunnel

In this particular case, the small wind tunnel available in laboratory has the similarly system to the open-circuit wind tunnel above. However, the fun will created a moving air through the tunnel section and the air exits through the outlet.

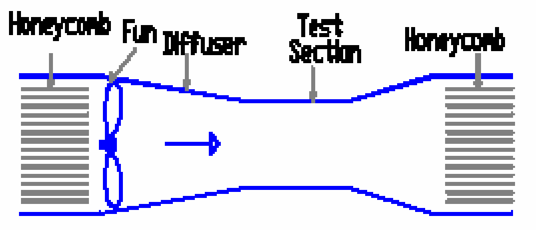


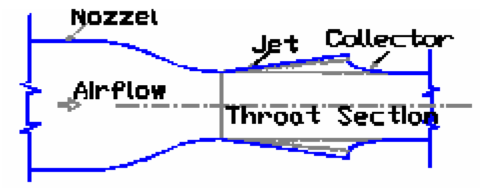
Figure 9. Scheme the open-circuit of small wind tunnel

Figure 10a. Photo Small Wind Tunnel

Figure 10b. Photo Small Wind Tunnel

Figure 10c. Photo Small Wind Tunnel (Department Marine and Technology)

The throat of a wind tunnel is the test section, which the throat condition may either be closed or open to the environment. This tunnel can be arranged as closed and opened tunnel section. However, the open-throat tunnels are suitable only for aeronautic and basic fluid mechanic studies. They are not suitable for testing civil engineering structures such as building and chimneys, which are exposed to turbulent boundary-layer winds. So, testing of most civil engineering structures requires a boundary-layer wind, which can only be generated in a close-throat section.



Open or closed –throat wind tunnel.

For the pressure condition considered, the most wind tunnels are not pressurized. The pressure in a wind tunnel is highest immediately downstream of its fun. Then, it will reach a minimum on suction side of the fan. It is because the speed-up of flow through the throat (test section), the pressure in the throat is normally low - less than atmospheric. In general, mostly non-pressurized tunnels have done for test of buildings and others structures due to the high cost of large-size pressurized tunnels.

Wind speed in the wind tunnel should be considered in order to decide whether having a high–speed or low–speed tunnels. The flow in the tunnel is considered as compressible, which means the density of air changes significantly from place to place in the tunnel or around test objects. In this study, low – speed tunnel has been planed, which is the density of air is essentially constant everywhere, and the flow considered as incompressible. The wind tunnels are commonly referred to as low – speed tunnels that the Mach number involved is less than one-third under the atmospheric condition.

For testing structural models, velocity profile must be generated as vertical distribution of the tunnel test section, which is similar to the logarithmic profile or the power–law profile of the wind encountered by prototypes.

Published Data for Sphere/Domes problem

In this particular case, data of wind tunnel test of sphere represented by the published data available. Fortunately, those models were wind tunnel tested and have already been used for a reference by the researcher. The Maher's classical study was conducted research on large model (600 mm diameter) in 1971 that the result has undertaken to determine wind loads on domes and hemispheres in boundary layer flow.

The study by Maher and some by Blessmann were generally arrived similar conclusions. Study of wind acting to the circular dome rising directly from the ground (Fig.11) presented pressure coefficient, distribution pattern and diagrams that can be described in Figure 11.

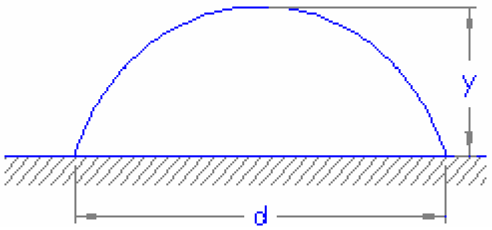


Figure11. Elevation of circular dome rising directly from the ground.

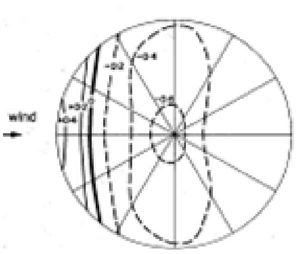


Figure 12. Plan view: pressure coefficients for y/d = ½ (hemisphere)

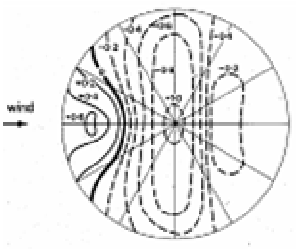


Figure 13. Plan view: pressure coefficient for y/d = ¼

Table 1. Limiting values of Cpe and values CL for domes rising directly from the ground.

Ratio y/d	Maximum positive Cp	Maximum Cp at centre	CL
1/15	+ 0.1	-0.3	0.15
1/10	+0.2	-0.3	0.2
1/8	+0.2	-0.4	0.2
1/6	+0.3	-0.5	0.3
¼	+0.4	-0.6	0.3
½	+0.6	-1.0	0.5

Published Data for Hyperbolic Cooling Tower problem

In this particular case, it is believe that the pressure distribution around a hyper-bolic cooling tower is similar to that around a circular cylinder. In the study of wind load on cooling towers, the common Cp used to represent the difference between the external and the internal pressure coefficients.

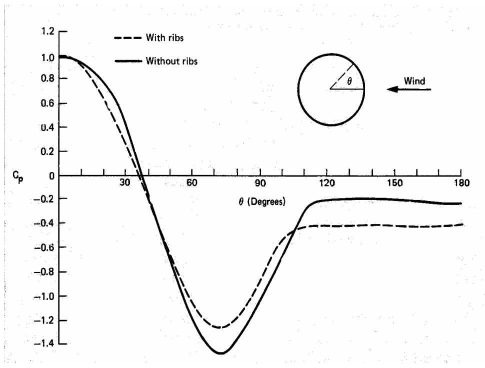


Figure 14. Distribution of local mean pres-sure coefficient around the throat of the cooling tower (ASCE, 1987)

Figure 14 gives the variation of Cp around the throat of typical hyperbolic cooling towers. The solid line represents towers with smooth surface, and the dashed line represents cooling towers with vertical ribs. Vertical ribs are often fitted to the hyperbolic cooling towers in order to reduce dynamic loads on the tower wall.

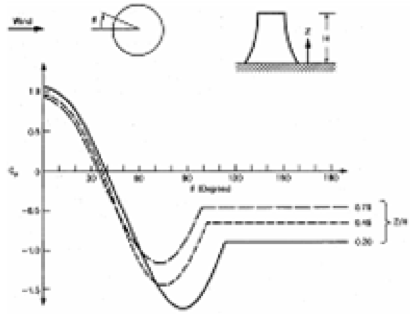


Figure 15. Distribution of local mean pressure coefficient at different height around the hyperbolic throat of the cooling tower (ASCE, 1987)

Figure 15 represents curves average values derived from several field studies (full-scale measurements) described in ASCE (1987). The variation of Cp with height z is given in Figure 14.

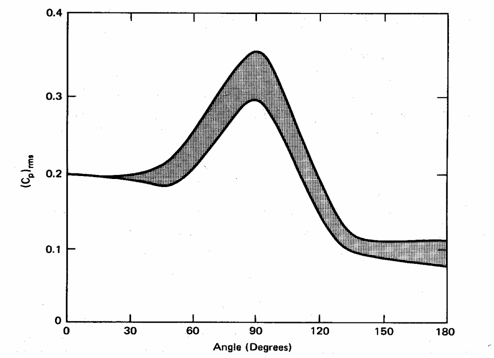


Figure 16. Distribution of root – mean square pressure coefficient around throat of a hyperbolic cooling tower (ASCE, 1987)

In addition of this research, the intensity of pressure fluctuations on a cooling towers can be characterized by the rms pressure coefficient (Cp) rms. Based on limited full-scale test result, the value of (Cp) rms around the throat of hyperbolic cooling towers can be determined from Figure 16. More about wind load on hyperbolic towers can be found in ASCE (1987).

Comparison and Discussion of Published data to the CFD result

The figures include readings taken from the published data, qualitative comments on the visual observations of the behaviour of the models during the test on CFD methods, additional calculations useful for interpreting the investigation and measured data. Comments regarding the observations were made during the CFD method testing and some clarifycations of the notations used in figure are provided following set of figures.

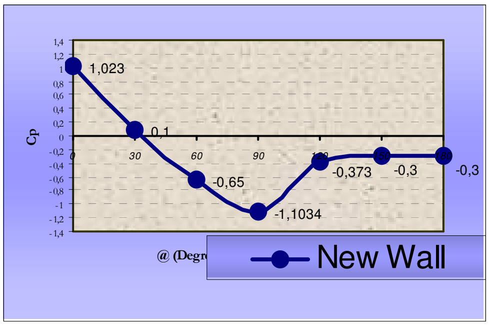


Figure 17. Distribution of local mean pressure coefficient around the hyperbolic Cooling Tower to represent the fabric structure by CFD method under Large Eddy Simulation (LES-Smagorinsky & Lilly)

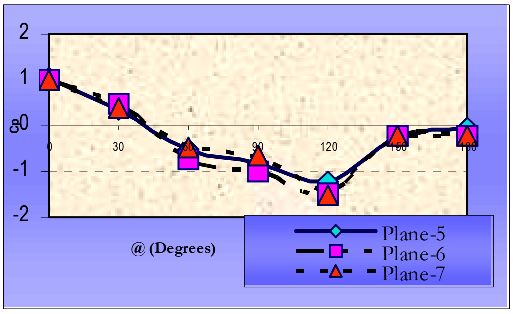


Figure 18. Distribution of local mean pressure coefficient at different heights around the Cooling Tower to represent the fabric structure by CFD method under Large Eddy Simulation (LES-Smagorinsky & Lilly)

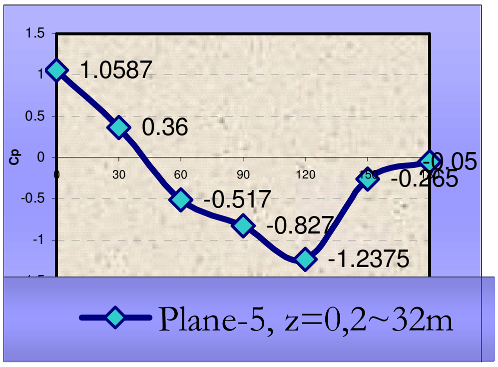


Figure 18a. Distribution of local mean pressure coefficient at different heights around the Cooling Tower to represent the fabric structure by CFD method under Large Eddy Simulation (LES-Smagorinsky & Lilly)

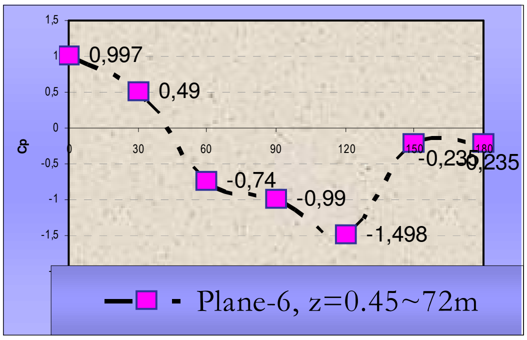


Figure 18b. Distribution of local mean pressure coefficient at different heights around the Cooling Tower to represent the fabric structure by CFD method under Large Eddy Simulation (LES-Smagorinsky & Lilly)

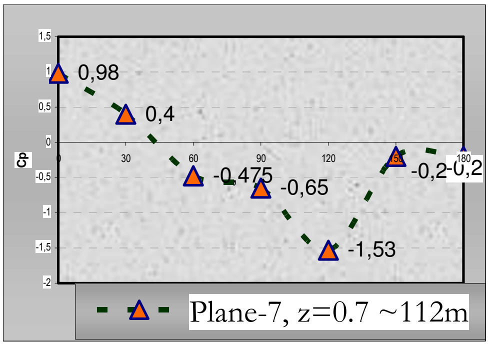


Figure 18c. Distribution of local mean pressure coefficient at different heights around the Cooling Tower to represent the fabric structure by CFD method under Large Eddy Simulation (LES-Smagorinsky & Lilly)

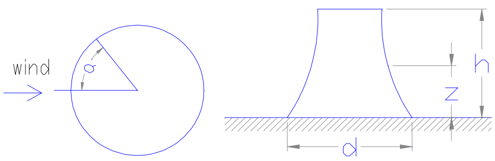


Figure 19. Guiding the angle to describe the pressure coefficient around throat combine with several of different height measurement.

$$\frac{Z1}{H} = \frac{0.32}{1.6} = 0,2 \rightarrow Z1 = 0.26 \sim 32m$$

$$\frac{Z2}{H} = \frac{0.72}{1.6} = 0,45 \rightarrow Z1 = 0.72 \sim 72m$$

$$\frac{Z1}{H} = \frac{1.12}{1.6} = 0,7 \rightarrow Z1 = 1.12 \sim 112m$$

A comparison of the pressure coefficient can be described between the figure 14 and figure17. From the figures, it can be seen the mean and fluctuating pressure distribution compare favourably with earlier study for flow characteristic, similar shape pattern and approaching value occurred on the figure. Contour distribution and pressure coefficient value presented is the result of study by CFD method to the single cooling tower model under LES. In this particular case, relative result from CFD has also been studied such as characteristic pressure under laminar flow problem. For the laminar flow problem, the result obtained is a little bit far from the published data by the ASCE, 1987. However, defining single cooling tower model in CFD under turbulent simulation (LES), the obtained result compared to the published data much more agreed due to the close value of the pressure coefficient presented.

Single Sphere.

Comparing the result of pressure coefficient of single sphere can be described between the figure 12 as the published data by Maher's and the result of CFD method particularly under LES approached. Qualitative comment on the visual observation between the CFD result and the Maher's classic studies lead to the arguing of the compare favourably among them. Visual observation to CFD result and the earlier study can be described below:

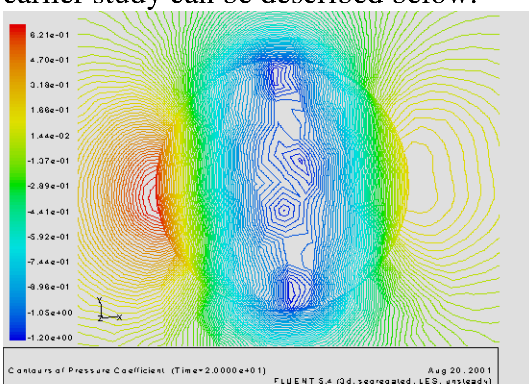


Figure 20. Pressure coefficient contour of the whole body from the top of plan

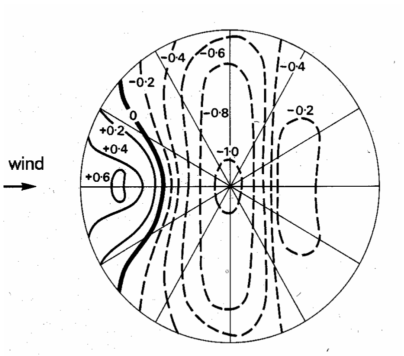


Figure 21. Plan view: pressure coefficients for y/d = ½ (hemisphere)

The result of wind load to the domes conducted by Maher in the current ratio of h/D has also been investigated and compared by Letchford and Sarkar (2000), which can be seen in figure 3, compared to CFD method on sphere model below

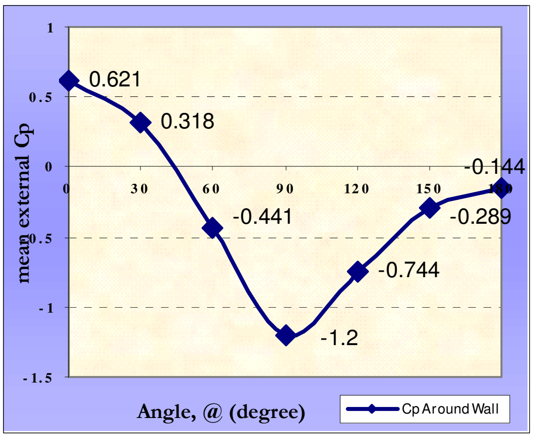


Figure 22. Mean pressure coefficient around the sphere under LES simulation

Comparison of mean pressure coefficient along centreline of a smooth dome, by Letchford, cs (fig.3) indicates well approximated to Letchford and Sarkar investigation. So, it is more confident and promised result performed by the CFD will led more complex problem need to investigate in order to deal with the wind load to the fabric structures.

Single Sphere Model

This model was adopted from the original domes as the arched roof based on the Maher and Blessmann. This model is a circular dome rising directly from the ground with y/d = ½. The sphere model was measured as height, y or H = 7.5 cm, and diameter of bottom structures is D = 15 cm.

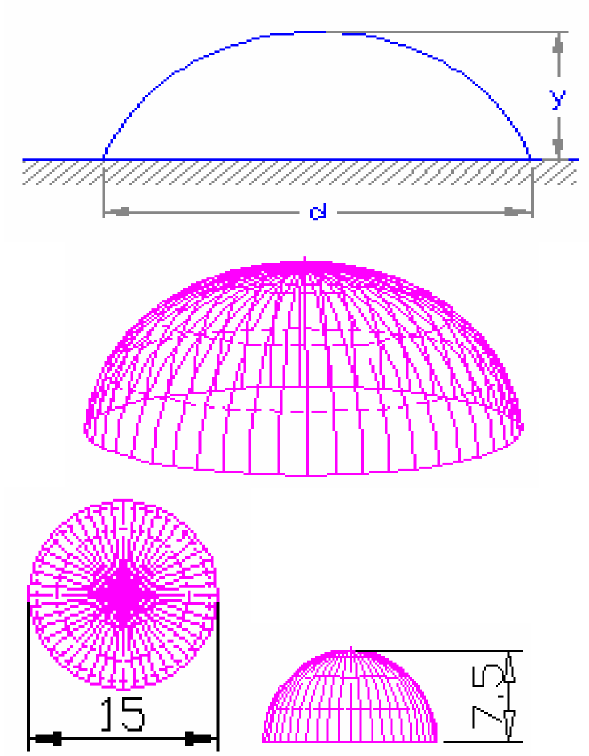


Figure 23' Sketch of Single Sphere (De3.igs of IGES file)

The model geometry was modified in CFD which was selected as a default of measurement in meter. A smaller scale was developed of H=0.75 m, and D=1.5m, respectively. The single sphere has divided into 8 (eight) surfaces with every connection have a rib. The surface of the sphere is known as smooth domes in the field of researcher. In this term, the geometry developed is involved small interval (10 interval meshes) of surfaces will be generated as constrain the mesh generation process. The domain developed was based on the height of model, which is situated in the suitable place.

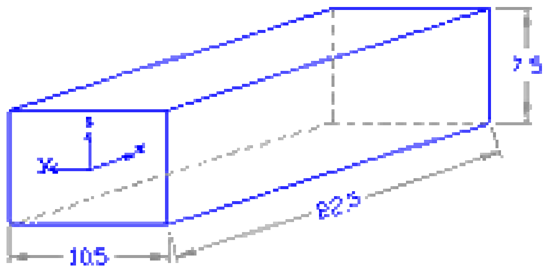


Figure 24. Domain of Single Sphere.

The dimensional of domain developed is 22.5 x 10.5 x 7.5 in unit, which are long x direction as the length, y direction as wide and z direction as the height of domain respectively. Recapitulation to read the grid points and geometry of the single sphere (in Gambit) can be described in figure scheme below with the same procedure in importing and cleaning up model design.

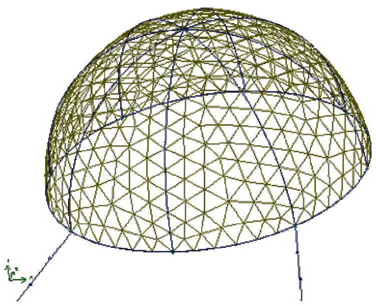


Figure 25. Grid mesh generating

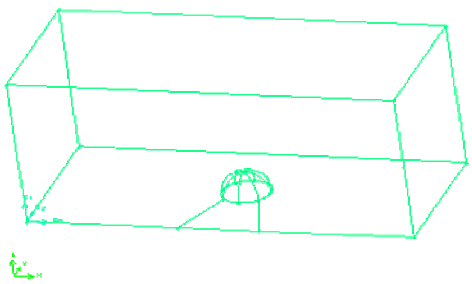


Figure 26. Brick and Sphere

Creating a brick around the multiple cooling tower body of the width (x) =22.5, depth (y) =10.5, and height (z) =7.5, can be described above in figure 26.

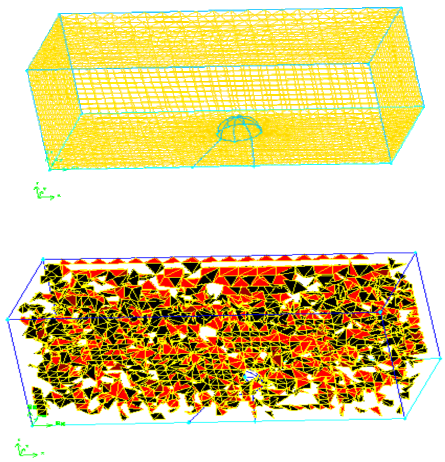


Figure 27. The mesh developed on domain & Elements within a specified quality range of 0.6 upper and 0. 7 lower ratios

To evaluate the reliability of element developed the volume meshes need to be examined. The examining identified the mesh volume created, aspect ratio, how many nodes created and the skew of floating element volume created. The 3D element or mesh volume can be evaluated of 120838 meshes volume and creating 25504 nodes developed with 1: 4 of aspect ratios, which domain developed as 4 wall, 1 velocity inlet and 1 outflow. There is different meshes volume developed when the domain is as 3 walls, 1 velocity inlet and 2 outflows. The meshes volume developed is 25519 elements and 120909 nodes with aspect ratio of 1:4.

Typical process has been applied as well to process iteration prediction set up to 250 for laminar flow and set up 1600 iteration for turbulent problem flows. The result is 88 iteration for laminar and 470 iterations for turbulent problem. Velocity Inlet Magnitude defined at 1 m/**s** and the material air with default the density of 1.225 (kg/m^3) and viscosity of 1.7894 x 10^{-5} (kg/m-s), then the Fluent will process the flow problem.

The Result of the Turbulent Flows under Large Eddy Simulation (LES) of Single Sphere

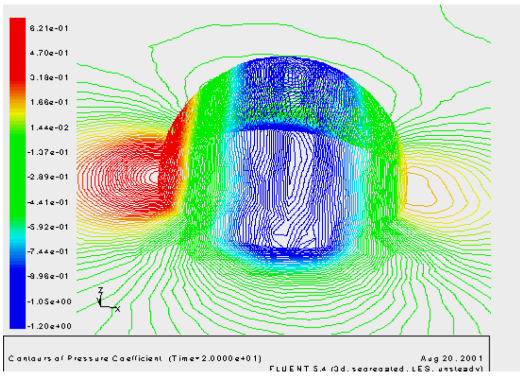


Figure 28. Pressure coefficient contour of the whole body from side elevation

Wind loading test using CFD

The numerical investigation of wind load testing on CDF method indicated that the general nature of the pressure and suction distributions on the model were obtained. In this particular case, the advantages of CFD method applied wind load to fabric membrane structure rather than to solve the dynamic fluid behaviour of wind problems.

General Conclusion and Recommendations

The model developed in CFD is more flexible depend on the ability of user to make it. Many kinds of model structures can be developed easily regarding to the aims of research study. All of them are such as the advantage of CFD methods presented, however it is depend on the capability and availability of computer hardware and the software. As an individual conclusion that this study would probably be replaced the wind tunnel testing to predict pressure coefficient and other parameters intended. It also may be concluded that many advantages can be used for other study related to structural engineering.

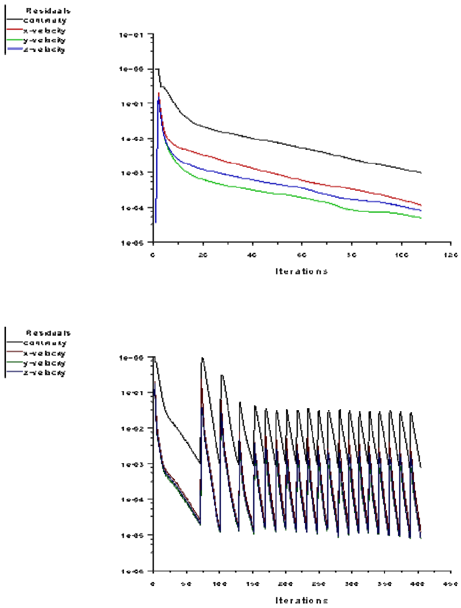


Figure 29. Plot the residual of laminar & turbulent flow

The Result of the Turbulent Flows on Large Eddy Simulation (LES) of Cooling Tower

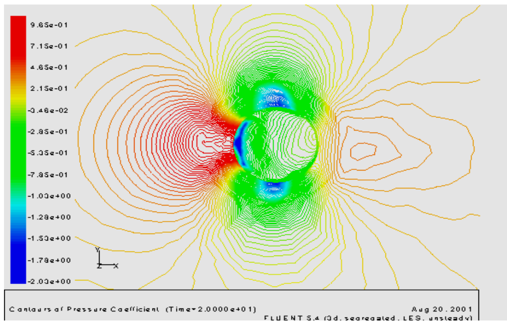


Figure 30. Pressure coefficient contour of the whole body from the top of plan (Coded De11)

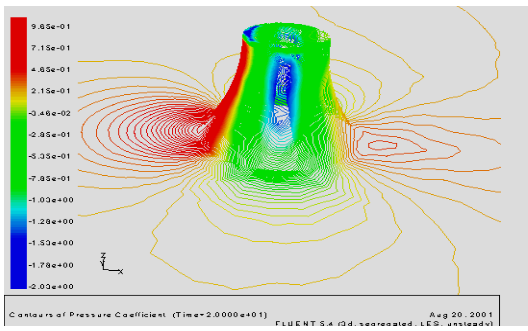


Figure 31. Pressure coefficient contour of the whole body from side elevation

REFERENCES

Ando, K., Ishii A., Suzuki T., Masuda K., and Saito Y., 1999, "Design and construction of a double membrane air-supported structure", *Journal of Wind Engineering and Industrial Aerodynamics, 21, 786-794.*

Barlow, J.B., Rae, W.H.,Jr, Alan Pope, 1999, "Low-Speed Wind Tunnel Testing", Canada, John Wiley & Son.

Birdair, 2001, "Technical Information: Design" www.birdair.com

Campbell, D.M., 2000, The Unique Role of Computing in the Design and Construction of Tensile membrane Structures" Journal Paper, www.yahoo.com

Cook, N.J., 1985, The Designer's guide to wind loading of building structures, London, Butterworths.

Dyrbye, C.and Hansen, S.O., 1997, "Wind Loads on Structures", West Sussex-England, John Wiley & Son.

Ehrhard, J. and Moussiopoulos, N., 2000, "On a new non-linear turbulence model for simulating flows around building-shaped structures", *Journal of Wind engineering and Industrial Aerodynamics*, 88, 1, 91-99.

Frei, O., 1967, "Tensile Structures", M.I.T. Press, Cambridge.

Irwn and Wardlaw, 1973 "A Wind Tunnel Investigation of a Retractable Fabric Roof for the Montreal Olympic Stadium" *Journal of Wind Engineering and Industrial Aerodynamics, 53 & 58, 57-103.*

Kazakevitch, M., 1998, "The aerodynamics of hangar membrane roof", *Journal of Wind Engineering and Industrial Aerodynamics, 77 & 78, 157-169.*

Kim, S.E. and Boysan, F., 1999, "Application of CFD to environmental flows", *Journal of Wind Engineering and Industrial Aerodynamics, 81, 145-158.*

Krisnha, P., 1978, "Cable – Suspended Roofs", United State of America, McGraw-Hill.

Kronenburg, R., 2000, "Portable Architecture", Oxford-England, Architectural Press.

Lakehal, D., 1998, "Application of the k-ε model to flow over a building placed in different roughness sublayers", *Journal of Wind Engineering and Industrial Aerodynamics, 73, 59-77.*

Letchford, C.W. and Sarkar, P.P., 2000, "Mean and fluctuating wind loads on rough and smooth parabolic domes", *Journal of Wind engineering and Industrial Aerodynamics*, 88, 101-117.

Liu, H., 1991, "Wind Engineering A Handbook for Structural Engineers", United State of America, Prentice-Hall, Inc.

Logan, D.L., 1986, "A First Course in the Finite Element Method", Boston-USA, PWS Engineering.

Mikkelsen, A.C. and Livesey, F.M., 1996, "Evaluation of the use of the numerical K-ε model Kameleon II, for predicting wind pressures on building surface", *Journal of Wind Engineering and Industrial Aerodynamics, 57, 375-389.*

Mcllmann, H., 1974, "Analysis of Hanging Roofs by Means of the Displacement Method", Lyngby-Denmark, Poyteknisk Forlag.

Murakami, S., 1998, "Overview of turbulence models applied in CWE-1997", *Journal of Wind Engineering and Industrial Aerodynamics, 74-76, 1-24..*

Oortmerssen, G.V., 1990, "CFD and CAD in Ship design – Proceedings of the International Symposium on CFD and CAD in Ship design Wageningen, The Netherlands 25-26 September 1990", Development in Marine Technology, New York-USA, Elsevier Science. (Voogt, 1990)

Rocha, M.M., Cabral S.V.S., and Riera, J.D., 2000, A comparison of proper orthogonal decomposition and Monte Carlo simulation of wind pressure data", *Journal of Wind engineering and Industrial Aerodynamics*, 84, 329-344.

Russell, B., 2001 update "Mathematics rightly viewed, possesses not only truth, but supreme beauty, a beauty, cold and austere, like that of sculpture", Rudy Enos Design, WWW.-Special Structures.com

Shaeffer, R.E., 1996, "Tensioned Fabric structures - A Practical Introduction", task Committee on Tensioned Fabric Structures, Virginia-USA, American Society of Civil Engineers.

Sachs, P., 1987, "Wind Forces in Engineering", Oxford-England, Pergamon Press Ltd.

Su, M.D., Tang, G.F., and Fu, S., 1999, "Numerical simulation of fluid flow and thermal performance of a dry-cooling tower under cross wind condition", *Journal of Wind engineering and Industrial Aerodynamics*, 79, 289-306.

Szabo, J., Kollar, L., and Pavlovic, M.N., 1984, "Structural Design of cable suspended Roofs" West-Sussex-England, Ellis Horwood Limited.

Tamura, T., Kawai, H., Kawamoto, S., Nozawa, K., Sakamoto, S., and Ohkuma, T., 1997, "Numerical prediction of wind loading on buildings and structures – Activities of AIJ cooperative project on CFD", *Journal of Wind Engineering and Industrial Aerodynamics, 67 & 68, 671-685.*

Uematsu, Y., Yamada, M., Inoue, A., and Hongo, T., 1997, "Wind loads and wind-induced dynamic behavior of a single-layer latticed dome", *Journal of Wind Engineering and Industrial Aerodynamics, 66, 227-248.*

Versteeg, H.K. and Malalasekera, W., 1995, "An Introduction to Computational Fluid Dynamics", New York-USA, John Wiley & Son.

Yasui, H., Marukawa, H., Katagiri, J., Katsumura, A., Tamura, Y., and Watanabe, K., 1999, "Study of wind-induced response of long-span structure", *Journal of Wind Engineering and Industrial Aerodynamics, 83, 277-288.*