Nineteenth Australasian Wind Engineering Society Workshop, April 4-6, 2018, Torquay, Victoria

A comparison of total design wind force estimates between traditional methods and CFD for open-framed mining structures

Robert Reis[1]
[1]*School of Engineering, Griffith University, Gold Coast, Qld, Australia*
robert.reis@griffithuni.edu.au

ABSTRACT

Mining related structures often differ to the general structural forms regularly encountered by designers and investigated by researchers. They are often of irregular form and are mostly open framed with high levels of blockage within the structural framing due to the mechanical equipment supported within. Currently, the author is researching the establishment of guidance on drag factors for mining structures, utilizing both CFD and wind tunnel testing of 3D printed models. This paper provides an overview of the work being undertaken to develop CFD benchmarks using the ANSYS Fluent Shear Stress Transport (SST) k-ω Model, prior to future wind tunnel assessment. The preliminary results derived from the CFD simulations here are compared with values obtained from AS1170.2.

1. Introduction

Open framed mining structures, such as conveyors, materials handling transfer and screening plant are common structures in Australia's mining regions. The design of such structures is complicated by the difficulty in understanding the total loads generated by wind on these structures. Although guidance is provided by AS/NZS 1170.2 for Australian design practice, the code is not clear on the effects of dense blockages which contribute to the overall structural loading. The code presents a methodology for multiple open frames using solidity ratios and shielding, but do not provide mechanisms for addressing the potential increases or decreases due to funneling or different angles of attack.

Significant work has been undertaken internationally by the petrochemical industry in the last decade, including wind tunnel testing, to provide guidance on petrochemical offshore platforms, however most of this research is not public domain. Petrochemical structures are the closest analogues to mining related structures, but differ primarily in that they contain cylindrical vessels such as boilers, piping and heat exchangers versus the square components used in mining.

This paper presents a summary of a portion of work being undertaken as an overall research project investigating wind loads on such structures, using CFD and wind tunnel testing, with the goal of providing improved means of assessing the load on such structures. The work described here relates to some preliminary results from CFD studies on a generic transfer tower structure.

2. Structural Model

The structures shown in

Figure 1 are typical of the high solidity open frame structures encountered on a mine site. This paper presents preliminary analysis of a conceptual transfer tower structure. Typically, these are tower-like structures at the elevated ends of conveyors. The towers support the elevated end of the conveyor and chute-work internally for the transfer of granular material from one system to another. Unlike buildings for human occupancy, the primary role of these structures is to provide support for the interconnected structures (conveyors) and the internal chute-work. Flooring is secondary, and only provided for the purposes of maintenance access.

Figure 1. Typical iron ore handling plant showing conveyor and transfer tower

Typical dimensions for such structures comprise braced frames with columns on 3m - 6m square grids. Floor heights typically range from 2.5m to 6m. For the conceptual model being investigated (Refer Figure 2), a square grid of 6m has been selected, with a uniform floor to floor height of 4.5m. Columns are 250UC72 and have been selected with typical universal beam section flooring. A solid floor plate has been included covering the floor support members.

Typically chute-work is of the order of 1.0m to 1.5m square. For the purposes of this study a 2m square chute has been used to provide a significant blockage (1/3 of face area). The model has been split into horizontal and vertical 2D planar sections for the analysis (Refer Figure 3).

3. Wind Model

The variation of drag with wind velocity is of practical importance. For many types of structure where the dynamic response can vary with the velocity and vortex shedding, it is important to understand the applied force and dynamic response through the whole velocity range. The structures considered here are different. They are typically very stiff braced structures with high natural frequencies. They are unlikely to have a significant dynamic or harmonic response in a global sense. As such, the designer is more concerned with the maximum, or ultimate limit state load for both the strength and stability cases.

Considering limit state winds exceeding velocities of 50m/s, and with cross sections of the individual structural elements around 200 – 300mm, the Reynolds Numbers are likely to be in excess of 8×10^5. At this scale, the momentum forces on bluff bodies will be dominant over viscous, so force will be approximately linear with the square of velocity. On this basis it is assumed that the velocity can be fixed to the upper bound design value, and the assumption re-assessed at a later date.

For Australian wind conditions and applying AS/NZS1170.2:2011, Standards Australia 2011, a design wind environment based on Region A, Importance Level 2, 50 Year Design Life, Terrain Category 2 and a 30m maximum structure height has been adopted. This gives a design ultimate wind speed of 50.4m/s as the basis of the simulations and calculations.

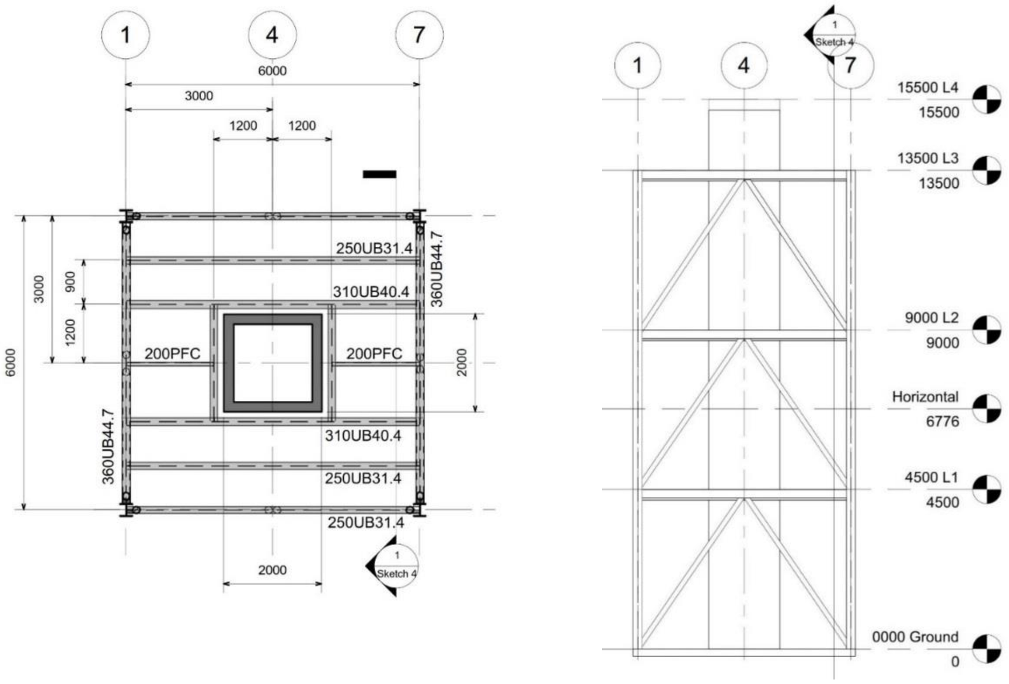


Figure 2. 3D view and elevation of conceptual transfer structure

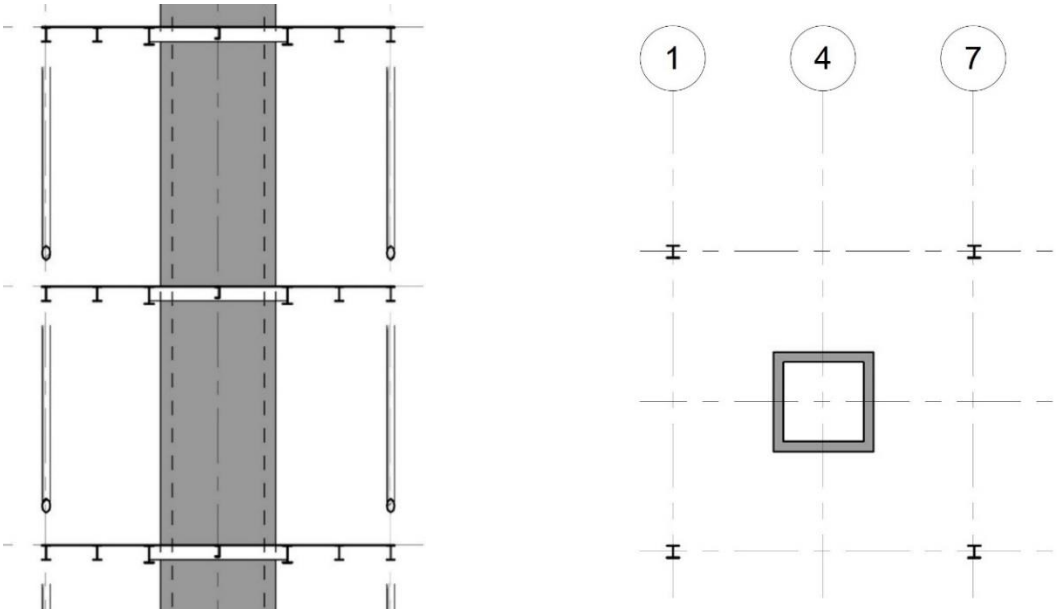


Figure 3. Vertical and horizontal tower cross sections used for 2D CFD flow analysis

4. CFD Model

The choice of CFD turbulence model is a significant topic in its own right. For the purposes of this work, a short survey of recent relevant papers has been undertaken as shown in Table 1. The papers listed were chosen as a reference as they have direct comparison with wind tunnel testing to confirm the CFD results. Based on evidence from the above references, and the inherent robustness in the two equation hybrid model, the Shear Stress Transport k-ω turbulence model has been selected Four cases were simulated as shown diagrammatically in Figure 4, where the orientation was varied with respect to the wind. Cases 1 – 3 took advantage of symmetry to reduce solution time. Case 4 was a variant of Case 3 where the angle of attack of the wind was offset by 15°. Symmetry was not used in this case.

CFD Turbulence Model Survey Summary		
Reference	**Turbulence Model**	
	RNG k-ε	**SST k-ω**
Zhang, Gu (2008)	X	
Yang, Dang, Niu, Zhang, Zhu (2016)		X
Mamou, Cooper, Benmeddour, Khalid, Fitzsimmons, Sengupta (2008)	X	
Hubova, Macak, Konecna, Ciglan (2017)		X

Table 1. Survey results of a selection of recent papers utilising CFD for wind load studies

For all models a flow domain of 30D upwind and side-wind, and 70D downwind was used, with "D" being the column face-width. The simulations utilised the SimpleC pressure-velocity coupling algorithm and run using 0.05 second time steps for a total of 5 seconds. Refer Figure 4 for case layouts.

5. Calculation of loading estimates from AS/NZS 1170.2:2011

Calculation of the wind forces has been undertaken using the equation from Section 2, Clause 2.5(3) and the aerodynamic shape factors, C_{fig} and shielding factor K_{sh} from Appendix E. From above the pressure has been calculated based on the ultimate wind speed of 50.4 m/s as follows:

- For the square columns of Case 3 and 4, and the chute of all cases, C_{fig} = 2.2 from Table E4. For the 250UC72 column in Cases 1 and 2, C_{fig} has been taken as 1.6 and 1.9 respectively from Table E5.
- The area of all elements, A_z, has been calculated using the windward face width given a unit frame height. It is assumed that for the horizontal section under consideration (being a unit height slice) that the flow is not influenced by the floors.
- The "solid" area of the windward frame has been taken as 2 x column width x unit height, or 0.508m^2 for the 250UC72 columns. The gross area has been taken as 6m column spacing x unit height, or 6m^2. This gives a solidity ratio of approximately 0.1.
- The frame spacing ratio, λ, for the chute was taken as 0.3, giving K_{sh} of 0.9.
- The frame spacing ratio, λ, for the leeward columns was taken as 1.0, giving K_{sh} of 1.0

The force on each element was then calculated using:

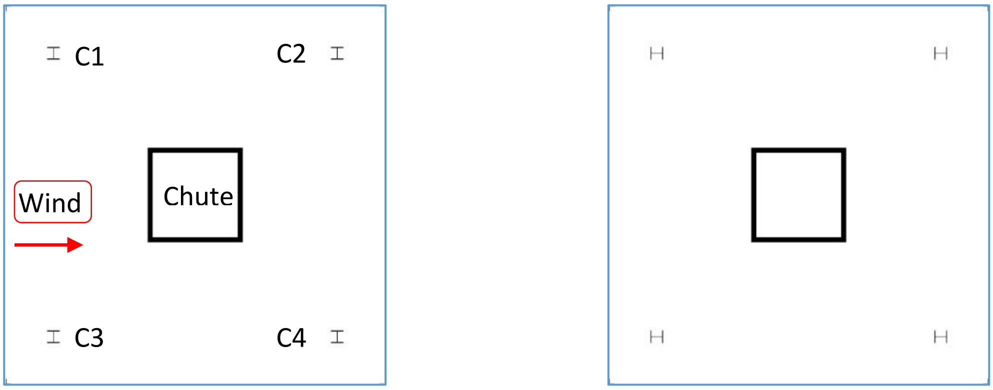
$$F = (0.5\,\rho_{air})[V_{des}]^2\, C_{fig}\, C_{dyn}\, K_{sh} A_z \qquad (1)$$

where ρ_{air} = 1.2kg/m^3 and C_{fig} = 1.0.

6. Preliminary CFD results and comparison to calculations

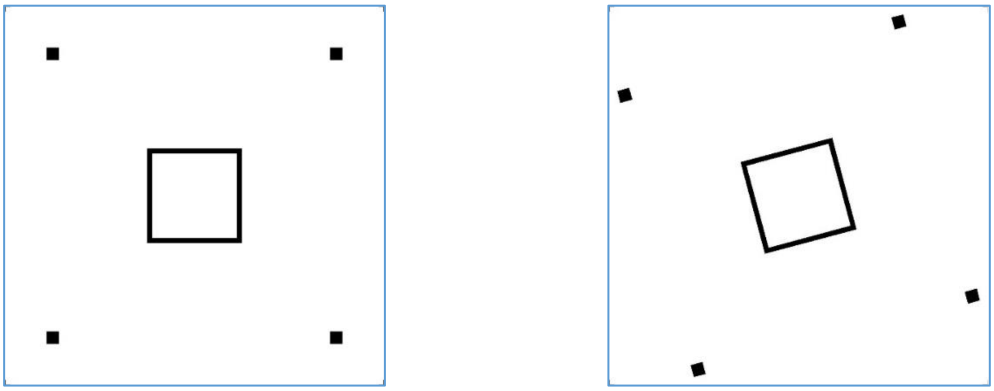
The vertical section model produced a maximum floor segment load of 564 N, occurring on the second level.

AS1170.2 is difficult to apply in this situation, and the contour plot (refer Figure 5) provides some interesting insight. The load on a 250 mm deep open section alone is around 780 N based on a drag factor, C_{fx} = 2.05. Although the CFD result is 28% lower than code, which could be a function of mesh, turbulence model and model step size, the qualitative result is informative. There would appear to be little increase of cumulative load across the plated diaphragm with significant stagnation between the floor members. Additionally, the stagnation effect is very localised to the floor diaphragm and does not extend deeply between the floors. This can be seen by the velocity profile between the floors being maintained at around 53 m/s.



Case 1 – 250UC72 Direction A

Case 2 – 250UC72 Direction B



Case 3 – 250UC72 Replaced with Square

Case 4 – Case 3 Rotated 15 degrees

Figure 4. CFD Simulation Cases

For cases 1 – 3 (refer Table 2), the sum of total load for the model is within good agreement with the shielded estimates from AS1170.2, ranging from 3.5 to 7% difference. The surprise is the distribution of load, which initially is non-intuitive with higher loads on the leeward columns. This starts to make sense once the velocity contour plots are examined (refer Figure 6) which shows an acceleration zone after the windward column, where the airflow accelerates to the side of the chute to re-join the main airflow.

Case 4 is an extension of the effects of cases 1 – 3, due to the windward exposure of all faces and the angle of attack, the total load is increased in this scenario where no floor boundaries exist. Once again, the accuracy of results from CFD are to be further examined, but the overall trend is the point of interest here in understanding how these effects can be reflected in design guidelines.

7. Conclusions

Although difficult to interpret in some instances, AS1170.2 Appendix E guidelines can be used to produce valid estimates for wind loading for design of open frame mining structures. Historically the modelling for the methodology has had a strong wind tunnel foundation. However, CFD is now able to be used to visualize the internal flow fields inside complex structures to gain some better insights as to the various components and distribution of loading.

Results from CFD compared with AS1170.2								
Case	**Description**	**Load for each reference location [N]** (Refer Figure 4 for locations)						
		C1	**C2**	**C3**	**C4**	**Chute**	**Combined**	**Variance**
1	AS – Shielded	619	619	619	619	6035	8513	CFD 3.5% > AS Shielded
	CFD	623	1046	623	1046	5476	8813	
2	AS – Shielded	736	736	736	736	6035	8979	CFD 3.5% < AS Shielded
	CFD	524	1008	524	1008	5594	8658	
3	AS – Shielded	852	852	852	852	6035	9443	CFD 7% < AS Shielded
	CFD	576	989	576	989	5663	8792	
4	AS – No Shield	1039	1039	1039	1039	8181	12337	CFD 16% lower
	CFD	520	683	644	600	7959	10407	
Notes	AS – No Shield = Calculated from AS1170.2:2011 with K_{sh} = 1.0 AS – Shielded = Calculated from AS1170.2 with K_{sh} as noted in Section 5. CFD – CFD simulation results Combined = sum of C1 + C2 + C3 + C4 + Chute Case 4 does not have a shielded version due to the rotation angle							

Table 2. Results from CFD compared with AS1170.2

The preliminary CFD modelling described above has shown the potential for lower shielding effects through the depth of the structure than previously thought for floor framing in the horizontal plane, and higher loads on leeward vertical elements due to acceleration of flow.

Although the accuracy of the CFD simulations is to be investigated through further work, it appears that the published guidelines coupled with CFD will provide a useful tool in understanding the loading distribution for limit state loads for structural design.

8. Future Work

There is still significant work to be undertaken to full form views on appropriate design values. Topics to be investigated include (i) work on varying the grid spacing to look at internal velocity relationships; (ii) variation of wind speed to assess effects on drag relationship; (iii) further investigate drag with various angles of attack; (iv) comparison with the ASCE Publication "*Wind Loads for Petrochemical and Other Industrial Facilities*"; (v) further review of the turbulence model selection, and sensitivity testing on grid and step size; (vi) extension of 2D CFD simulations to 3D simulations to include bracing, handrails and other attachments.

The most significant work planned for the near future will be wind tunnel validation of the CFD simulations.

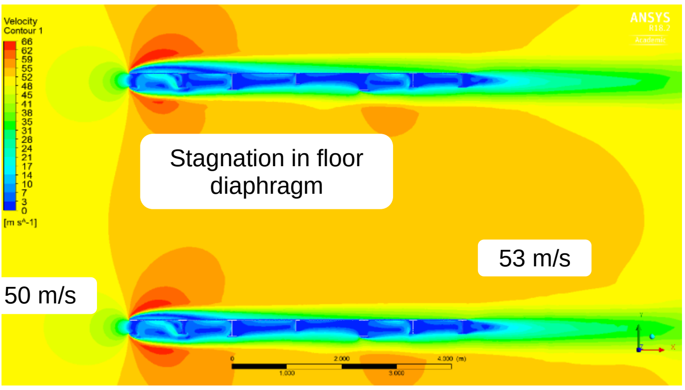


Figure 5. Velocity Contours for Vertical Section

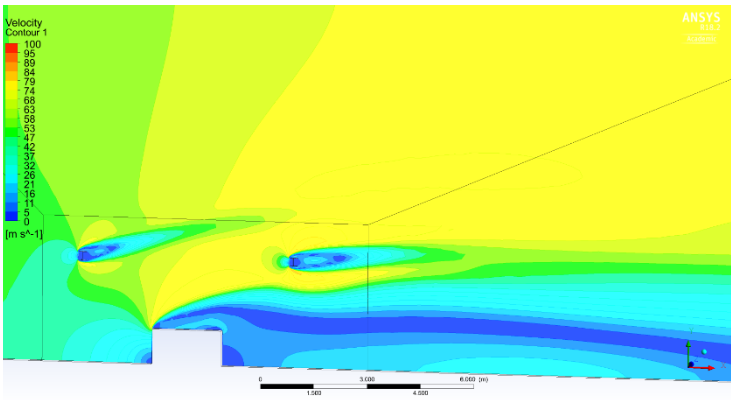


Figure 6. Case 1 Horizontal Section Velocity Contours

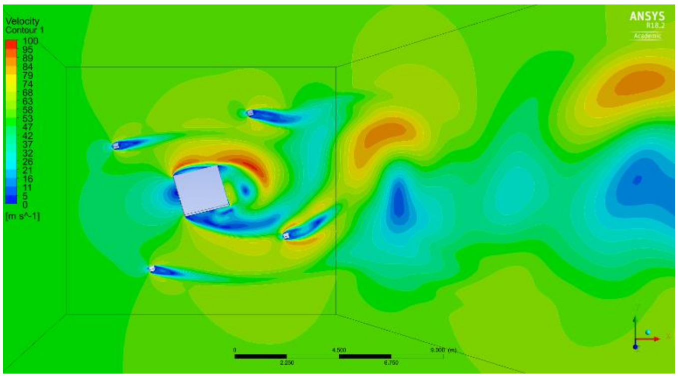


Figure 7. Velocity contour plot for case 4

9. References

Standards Australia, (2011), "Structural design actions. Part 2 Wind actions", Australian/New Zealand Standard, AS/NZS 1170.2:2011.

Zhang, A., Gu, M., (2008), “Wind tunnel tests and numerical simulations of wind pressures on buildings in staggered arrangement”, *Journal of Wind Engineering and Industrial Aerodynamics*, 96: 2067-2079.

Yang, F., Dang, H., Niu, H., Zhang, H., Zhu, B., (2016), “Wind tunnel tests on wind loads acting on an angled steel triangular transmission tower”, *Journal of Wind Engineering and Industrial Aerodynamics*, 156: 93-103.

Mamou, M., Cooper, K.R., Benmeddour, A., Khalid, M., Fitzsimmons, J., Sengupta, R., (2008), “Correlation of CFD predictions and wind tunnel measurement of mean and unsteady wind loads on a large optical telescope”, *Journal of Wind Engineering and Industrial Aerodynamics*, 96: 793-806.Hubova, O., Macak, M., Konecna, L., Ciglan, G., (2017), “External Pressure Coefficients on the Atypical High-Rise Building – Computing Simulation and Measurements in Wind Tunnel”, *Structural and Physical Aspects of Construction Engineering*, Procedia Engineering 190: 488-495.