Control of flow around a circular cylinder by the use of surface roughness: A computational and experimental approach

Ryan Merrick [1)] and Girma Bitsuamlak [2)]

[1)] *RWDI USA LLC, Miramar, Florida, 33025, email: Ryan.Merrick@rwdi.com*
[2)]*Department of Civil and Environmental Engineering/International Hurricane Research Center,Florida International University, Miami, Florida 33174, USA, bitsuamg@fiu.edu*

ABSTRACT

Some difficulties are encountered when simulating super-critical Reynolds number (R_e) flow over curved surfaces of a building in a low speed boundary layer wind tunnel (BLWT) due to the sensitivity of the flow to R_e. Surface roughness on the façade of the cylinder can affect the location of the separation point and the extent of the wake on the leeward face, upon which the wind-induced responses are dependent. This study attempts to control the flow around a circular cylinder through the implementation of artificial surface roughness across the exterior of the cylinder. Subsequently, the size of the artificial roughness will be correlated with R_e variations through a computational fluid dynamics wind flow simulations. These correlations will assist in selecting the appropriate artificial roughness that will cause boundary layer transition at points similar to super-critical R_e simulations. Several roughness patterns were tested on the circular cylinders, which were subjected to wind flows with varying turbulence intensity. Measurements of the pressure distribution across the façade have been obtained over the R_e range of $1\text{x}10^4$ to $2\text{x}10^5$ in a BLWT. Results from the experiments are compared with previously published experimental data, as well as computational simulations. The results have direct application in the testing of scale models in low speed BLWTs where super-critical flow characteristics are desired.

Key words: circular cylinder, bluff body, surface roughness, Reynolds number, boundary layer wind tunnel, drag coefficient, pressure distribution, pressure coefficient.

INTRODUCTION

Many obstacles are encountered when trying to produce high R_e flow effects over curved surfaces at relatively low wind speeds in a boundary layer wind tunnel. Simulating the effects of full scale winds over curved surfaces on scale models is made difficult due to the contrast in R_e between the model scale and full scale structures. R_e dictates the laminar boundary layer separation point, which was of particular interest in this study. The purpose of this study was to design a surface roughness pattern that would produce an apposite amount boundary layer turbulence across a curved surface and simulate super-critical R_e flow effects at a sub-critical R_e. Experiments were done with various roughness patterns on circulars cylinders for a range of wind speeds. The results from the experiments were then compared with computational simulations at super-critical R_e to determine which roughness pattern appropriately altered the

[1] Senior Technical Coordinator
[2] Assistant Professor

wind flow around the test model.

Numerous experiments have been conducted to investigate the flow around circular cylinders. Roshko (1961) defined the range of critical R_e, which poses the fundamental problem for the scale model testing of curved structure in low speed wind tunnels. For instance, overly conservative results may be obtained by model testing a curved structure at sub-critical R_e. In contrast, wind loads can be underestimated if a building is model tested at critical R_e. Thus, laboratories should be diligent in simulating the flow pattern that the structure would experience at full scale.

The concept of controlling the flow over circular cylinders is not novel, as many researchers have studied the effect of surface roughness on cylinders in the past. Dimpled surfaces were examined by Bearman and Harvey (1993), while roughness on a cylinder was studied by Szechenyi (1975). Both of these studies showed that the pressure distribution around the cylinder could be altered through the addition of a roughness pattern. The impact of roughness strips and engineered grooves on circular cylinders were investigated by Nakamura and Tomonari (1982) and Kimura and Tsutuhara (1991), respectively. While Nakamura and Tomonari noted that super-critical flows could only be obtained by roughness strips and not roughness patterns, Szechenyi concluded that the outlook was favourable for finding a surface roughness pattern that could simulate super-critical R_e at sub-critical R_e. The study by Kimura and Tsutuhara added that the orientation of the groove with respect to wind direction was critical in predicting the flow around the cylinder. Since this paper addresses the application of surface roughness for the model testing of structures, most commonly at 1:300 and 1:400 scales, it is not practical to consider aerodynamic trips that will produce varying flows when the model is rotated, such as grooves and strips. Very limited research has been done looking at the application of surface roughness to increase precision in model testing. Thus, this study investigated whether a roughness pattern could simulate super-critical flows and sub-critical R_e and hopes to open the discussion on this topic.

In the present study, different roughness patterns have been investigated inside a BLWT. In order to assess the effectiveness of the roughness patterns, the experimental pressure distributions must be measured against super-critical R_e distributions. In the absence of super-critical R_e experimental data, computational simulations were performed as benchmarks to correlate the patterns with R_e. Computational fluid dynamics (CFD) simulations of high R_e wind flows past a circular cylinder at critical R_e were carried out by Selvam (1997), and showed good agreement with experiments. The simulations performed for comparison purposes in this study, as well as those in Selvam's simulations, were conducted using a large eddy simulation (LES). Commercial CFD software Fluent (Fluent 2003) was used in the present study.

BACKGROUND

When simulating the wind flow over a scale model comprised of curved surfaces, discrepancies are present between the model scale data and full scale winds experienced by the structure. Since the R_e corresponding to the curved surface is a function of the radius of curvature, there is an inconsistency between the model scale R_e and the full scale R_e. For a circular cylinder, R_e number can be computed via Equation 1.

$$R_e = \frac{2ur}{\upsilon} \tag{1}$$

In Equation 1, R_e is the Reynolds number of the shape, u is the wind velocity in unobstructed flow, r is the radius of the curvature and υ is the kinematic viscosity of air. Sub-critical flow over a smooth cylinder generally occurs at R_e less than $2\text{x}10^5$. Sub-critical flows are characterized by laminar flow over the windward surface of the cylinder with the flow separating on the upwind face. Super-critical flow occurs at R_e greater than $4\text{x}10^6$ and is evident by the turbulent boundary layer that forms over the surface of the cylinder. The turbulent wind separates from the cylinder on the leeward face and results in a lower drag coefficient. The critical region is defined as flow resulting at R_e between those of sub-critical and super-critical flows. The relationship between R_e and drag coefficient is illustrated in Figure 1. Sub-critical and super-critical flows around a circular cylinder are illustrated in Figure 2.

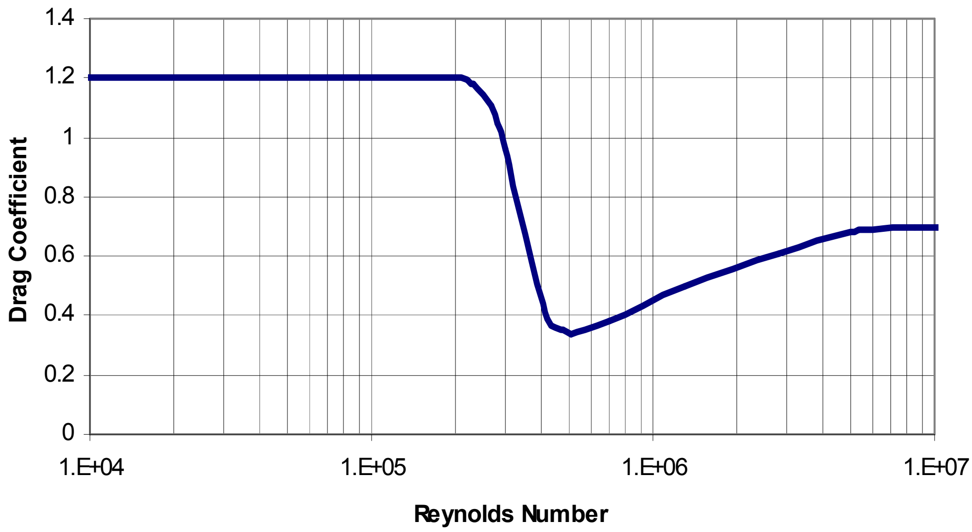


Figure 1 – Drag coefficient vs. R_e number for a circular cylinder (Scruton and Rogers, 1971).

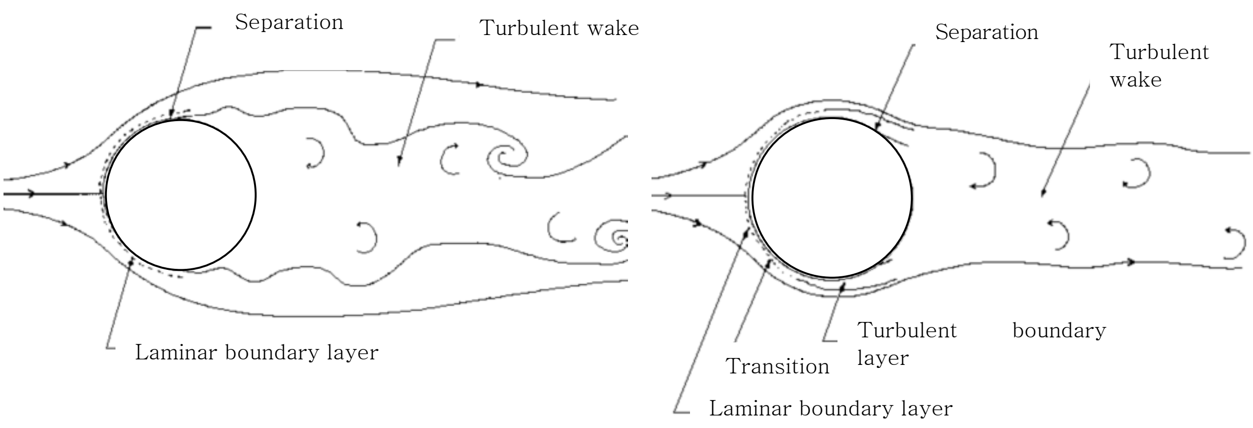


Figure 2 - Two-dimensional sub-critical (left) and super-critical (right) flow around a circular cylinder (Davenport 2003).

Factors such as surface roughness and turbulence in the free stream flow can cause the boundary layer transition to occur at lower R_e. The purpose of this study was to evaluate the effectiveness of aerodynamic trips in causing the transition from sub-critical to super-critical flow to occur at a low R_e. An appropriately designed aerodynamic trip would have direct applications in the testing of scale models in low speed BLWTs, where super-critical flow characteristics over curved surfaces are desired. The addition of groves on the windward face of the cylinder studied by Kimura and Tsutahara (1991) indicated that the drag loading on the cylinder could be reduced by the inclusion of a groove. They also reported that the location of the groove on the windward face was critical in determining the reduction in drag experienced. Similarly, Bearman and Harvey (1993) demonstrated that the drag coefficient could be reduced through the use of dimples. The current study attempted to build on previous research and develop a roughness pattern that can be utilized for scale model studies in BLWTs to compensate the effect of super-critical R_e.

EXPERIMENTAL MODEL DEVELOPMENT

The first step in the model development stage was to choose the orientation of the test apparatus. It was determined that two vertical circular cylinders mounted side by side would be the most efficient way to obtain the necessary data. Circular cylinders of 57 mm (2.25”) and 203 mm (8”) outside diameters were placed 610 mm apart in a BLWT as shown in Figure 3. The separation between the two cylinders ensured that the wakes produced from each cylinder during testing would not interfere. The top and bottom of each test cylinder were constructed of solid acrylic pipe. Resin inserts pieces were grown using a stereo lithography (SLA) machine and inserted between the acrylic sections, as seen in Figure 3. The models were mounted on the tunnel floor and fastened from above to prevent oscillation.

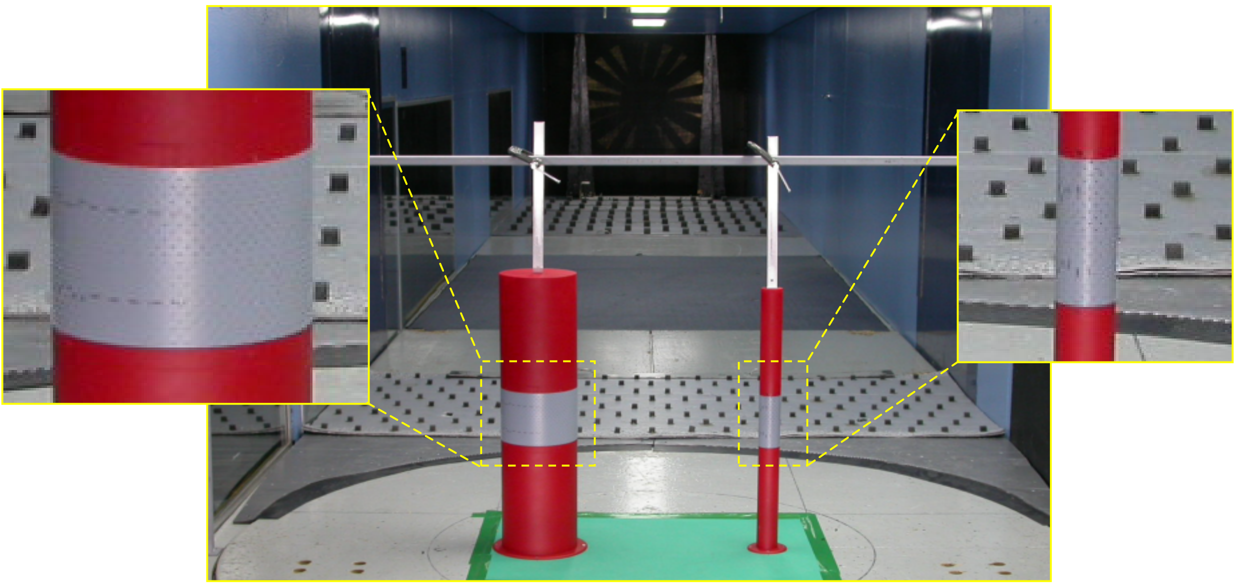


Figure 3 - Study model as tested.

Issues such as size, durability and effectiveness were considered in determining the shape of the surface roughness. Hexagonal prisms were chosen as the shape for the aerodynamic trips, as the sharp corners were desired to generate boundary layer turbulence across the surface. The hexagonal prisms were constructed with a 2 mm base to avoid damage due to handling prior to the test. The height of each prism is directly related to how effective the trip is in altering the flow pattern around the cylinder. Research by Szechenyi (1975) showed that a trip with a roughness R_e of 200 is required to modify the flow over the surface, although the turbulent boundary layer may not be totally developed until that number reaches 1000. The roughness R_e is defined in Equation 2.

$$R_{\mathrm{Re}} = \frac{uh}{\upsilon} \tag{2}$$

The variables in Equation 2 are identical to those defined in Equation 1, with the addition of the height of the trip, h. Roughness heights of 0.5 mm, 1.0 mm and 1.5 mm were chosen to achieve a wide range of roughness R_e. The distribution of the trips was selected to sufficiently cover the curved surface. Trips were distributed across the surface of the cylinder in a diamond pattern, 12.5 mm (1/2") apart, as seen in Figure 4. Roughness patterns with hexagonal prisms protruding both inwards and outwards were tested. Pressure taps were distributed across the surface of the cylinder to capture the pressure distribution across the façade. Pressure taps were more heavily concentrated in areas where high pressure gradients were anticipated.

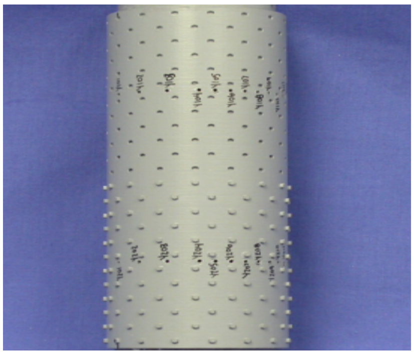


Figure 4 – Roughness pattern on the resin insert.

BOUNDARY LAYER WIND TUNNEL TESTING

Tests of the circular cylinders were performed in a BLWT facility. A synchronous pressure acquisition was enlisted to extract the pressure coefficients during testing. In order to obtain results for a range of R_e, data were acquired for both the 57 mm and 203 mm cylinders at five different wind speeds. Table 1 summarizes the wind speed, diameter and R_e for each of the wind tunnel tests.

To investigate the influence of turbulence on the effectiveness of the surface roughness, data were collected for each test cylinder in the presence of four varying turbulence intensities. The turbulence was generated by roughness elements on floor of the BLWT, as well as flow conditioning spires placed upstream of the test section. A crosswire probe was instrumented directly in front of the study cylinders prior to testing to determine the mean wind velocity and turbulence intensity generated by the working section of the tunnel floor and the spires. The resulting turbulence intensities for each test are summarized in Table 2. The crosswire probe was set at a height between the two rows of taps and placed directly in front of the cylinders to obtain a representative mean velocity and turbulence intensity for both rings of pressure taps. Figure 5 shows the crosswire probe set up in the BLWT.

Table 1 – Wind speed, diameter and *Re* relationships for wind tunnel tests.

Case	Diameter (mm)	Wind Speed (m/s)	R_e
1	57	3.0	1.17E+04
2	57	6.1	2.34E+04
3	57	9.1	3.52E+04
4	57	12.2	4.69E+04
5	57	15.2	5.86E+04
6	203	3.0	4.17E+04
7	203	6.1	8.33E+04
8	203	9.1	1.25E+05
9	203	12.2	1.67E+05
10	203	15.2	2.08E+05

Table 2 - Profile Turbulence Intensity.

Profile	Turbulence Intensity (%)
1	0.8
2	8.2
3	19
4	23

The pressure coefficients obtained through the wind tunnel experiments were calculated using the Equation 3.

$$C_p = \frac{P}{P_{st}} \tag{3}$$

In Equation 3, C_p represents the recorded pressure coefficient, P is the dynamic pressure measured at the tap location and P_{st} is the dynamic pressure measured at the stagnation point. Pressure coefficients representing the maximum, minimum, root-mean-squared and mean pressures were recorded for each pressure tap on the study cylinders.

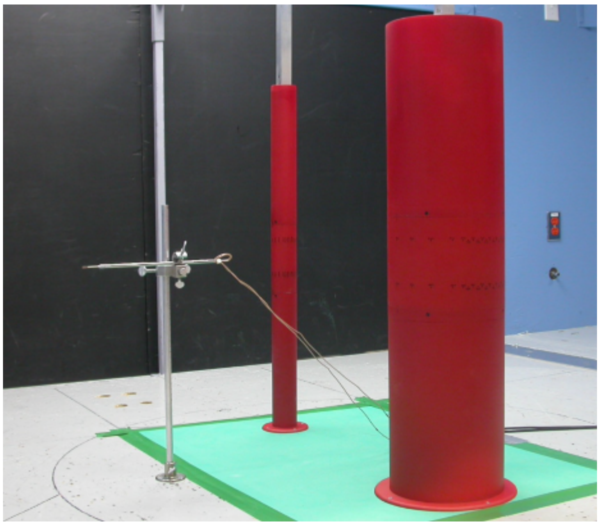


Figure 5 – Crosswire probe setup.

COMPUTATIONAL FLUID DYNAMIC (CFD) SIMULATION

A computational model similar to the BWLT test model was developed. Simulations were carried out for sub-critical and super-critical R_e cases. A computational domain (CD) having similar cross-section with the BLWT test facility was used (i.e. 2 m by 2.4 m). The size of the CFD test model was similar to the larger experimental model, with a diameter (D) of 200 mm and a height (H) of 500 mm. The commercial software FLUENT 6.2, was utilized for this simulation. Large eddy simulations (LES) were employed due to their relative accuracy for predicting the flow field around bluff bodies compared to other turbulence models (Bitsuamlak et al., 2000). The inlet, top, outlet, and two sides of the CD were set to 6D, 10D, 26D, and 6D from the centre of the base of the cylinder, respectively, as shown in Figure 6. Body fitted hexagonal grids were used near the surface of the cylinder, as shown in Figure 7. The body fitted grids used are advantageous for accurate surface boundary application and representation of the cylinder with limited grid points. In all present simulations $y^+ < 5$ ($y^+ = u^* y_P / \upsilon$) was applied, where u^* is friction velocity, y_p is the location of the first grid from the cylinder surface and υ is kinematic viscosity of air. In addition, the turbulence intensity was set to 8.2% to match the experimental profile 1. The grid details used in the present study at mid cylinder height, where the C_p values were extracted, is illustrated in Figure 7.

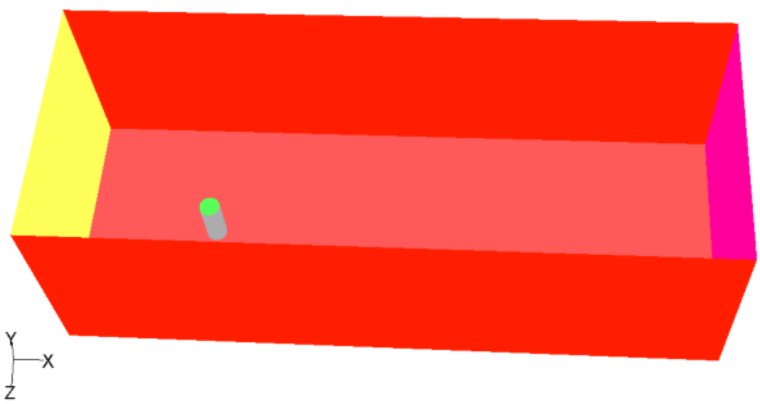


Figure 6 - Computational domain.

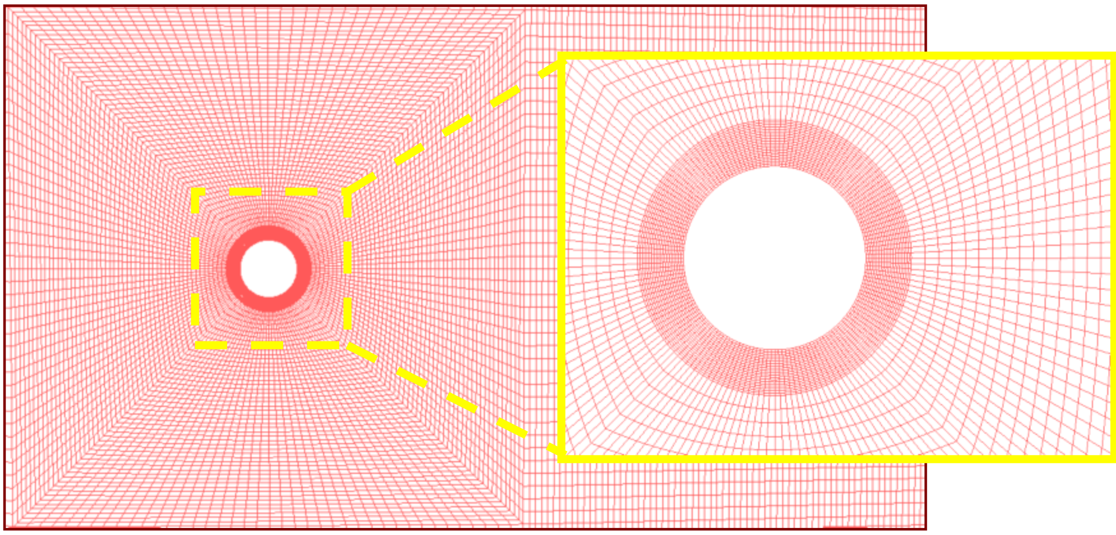


Figure 7 – Grid at mid cylinder height from the ground.

RESULTS AND DISCUSSION

Nearly 300 pressure distributions were collected in the BLWT, comprised of the various permutations of wind speed, cylinder diameter, surface roughness and turbulence. The experiments involving the smooth cylinders exhibited the classical effect of R_e on the flow around the cylinder. Figure 8 shows the mean pressure distribution around the cylinders without surface roughness in profile 1 for the indicated wind speed. In Figure 8, 0° points directly into the wind flow, while 180° is located on the leeward-most point of the cylinder.

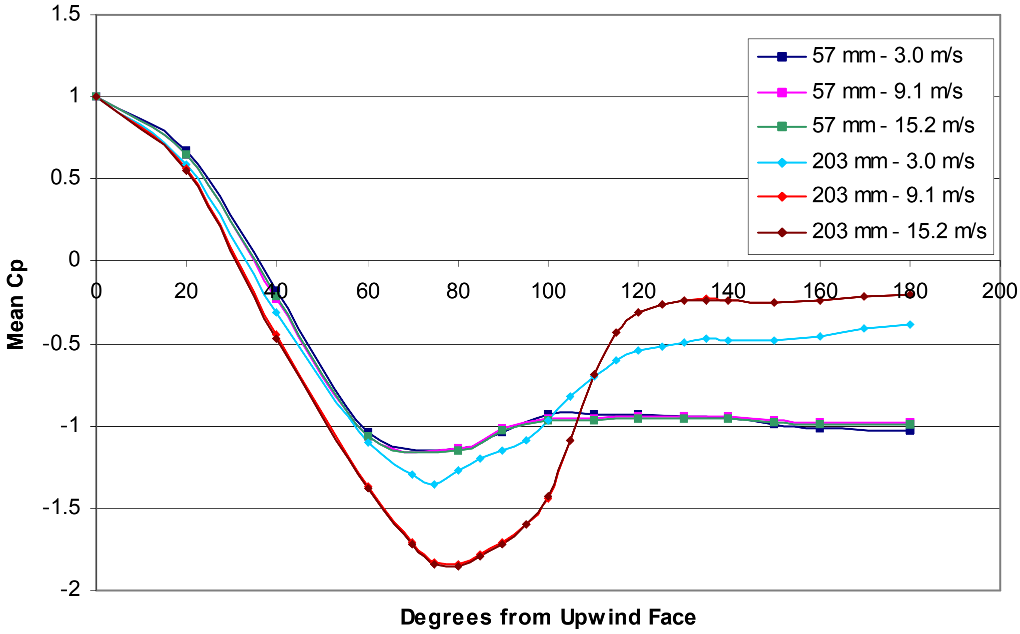
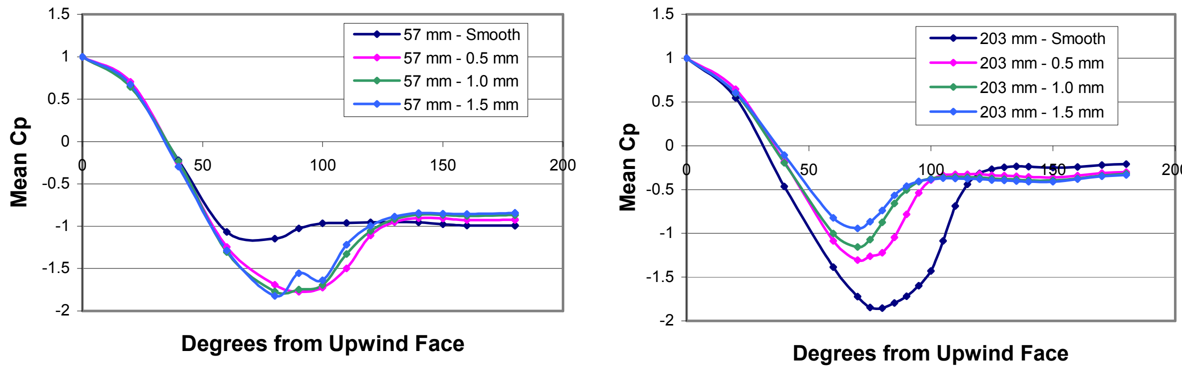


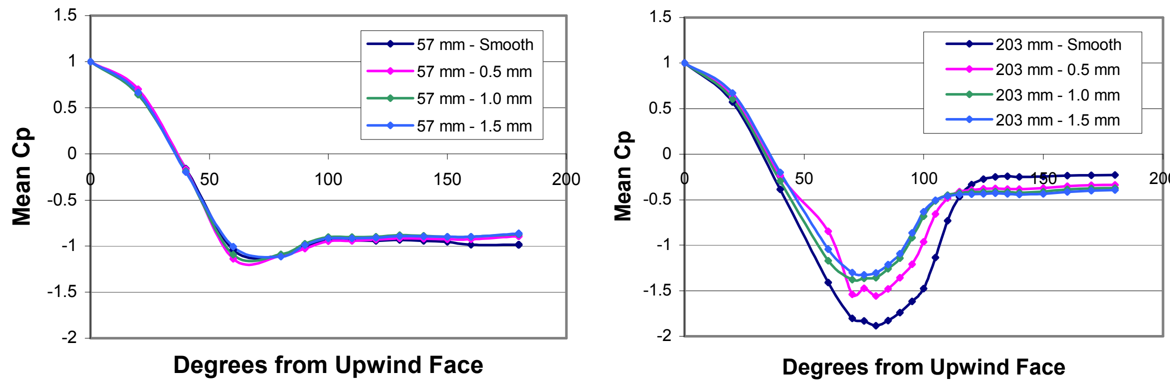
Figure 8 – Mean pressure distributions around circular cylinders in profile 1 without surface roughness.

The 57 mm cylinder exhibited sub-critical *Re* flow, characterized by the large suction zone on the leeward face and the similarity in pressure coefficients on the side and rear of the cylinder. Super-critical R_e flows were experienced around the 203 mm cylinder at wind speeds greater than 9.1 m/s. These flows are evident by the reduction in suction from the side to the rear of the cylinder and the noticeably smaller wake on the leeward face. At 3.0 m/s, the 203 mm cylinder appeared to be in the critical R_e transition zone, as the distribution resembles a combination of the sub and super-critical flow, as described early. This experiment further enforces the fact that R_e plays a crucial role in a model testing.

It was intended that through the addition of surface roughness, super-critical R_e flows could be simulated at sub-critical R_e for the purposes of model testing in a BLWT. As mentioned earlier, six roughness configurations were tested, in addition to a control cylinder without roughness. Figure 9 illustrates the effect of the surface roughness patterns on the flow around both test cylinders. In the legend for each plot, the first parameter given is the cylinder diameter and the second variable is the height of the aerodynamic trip. The upper plots in Figure 9 show the collected data with the roughness protruding out of the surface, whereas the lower graphs depict the data for the dimpled surface.



(a) Outward protrusions



(b) Inward protrusions

Figure 9 – Mean pressure distributions around circular cylinders with surface roughness at 15.2 m/s in profile 1.

The left plot in Figure 9a shows that the outward protruding roughness for all roughness heights on the 57 mm cylinder is effective in altering the flow to show super-critical characteristics. The same cannot be said for the inward protruding dimples, which had very little effect of the wind flow around the 57 mm cylinder, as seen in left plot of Figure 9b. The plots on the right side of Figures 9a and 9b show that the surface roughness can have an effect on reducing the peak negative pressure coefficient experienced and that the degree its influence on the wind flow is dependent on the height of the surface roughness. Thus, is it critical that these experimental results are compared with full scale simulations to determine most appropriate roughness pattern. Evidently, the outward protrusions are most effective than their inward counterparts, but the height of the roughness plays a critical role the simulation.

The effect of the surface roughness elements with varying levels of turbulence in the wind flow was also examined. It was hypothesized that larger roughness elements may be required in the absence of turbulence in the free stream flow. The four plots Figure 10 show the effect of the 1.0 mm outwards roughness pattern on the 57 mm cylinder at a wind speed of 15.2 m/s in the 4 test profiles, each with a unique turbulence intensity.

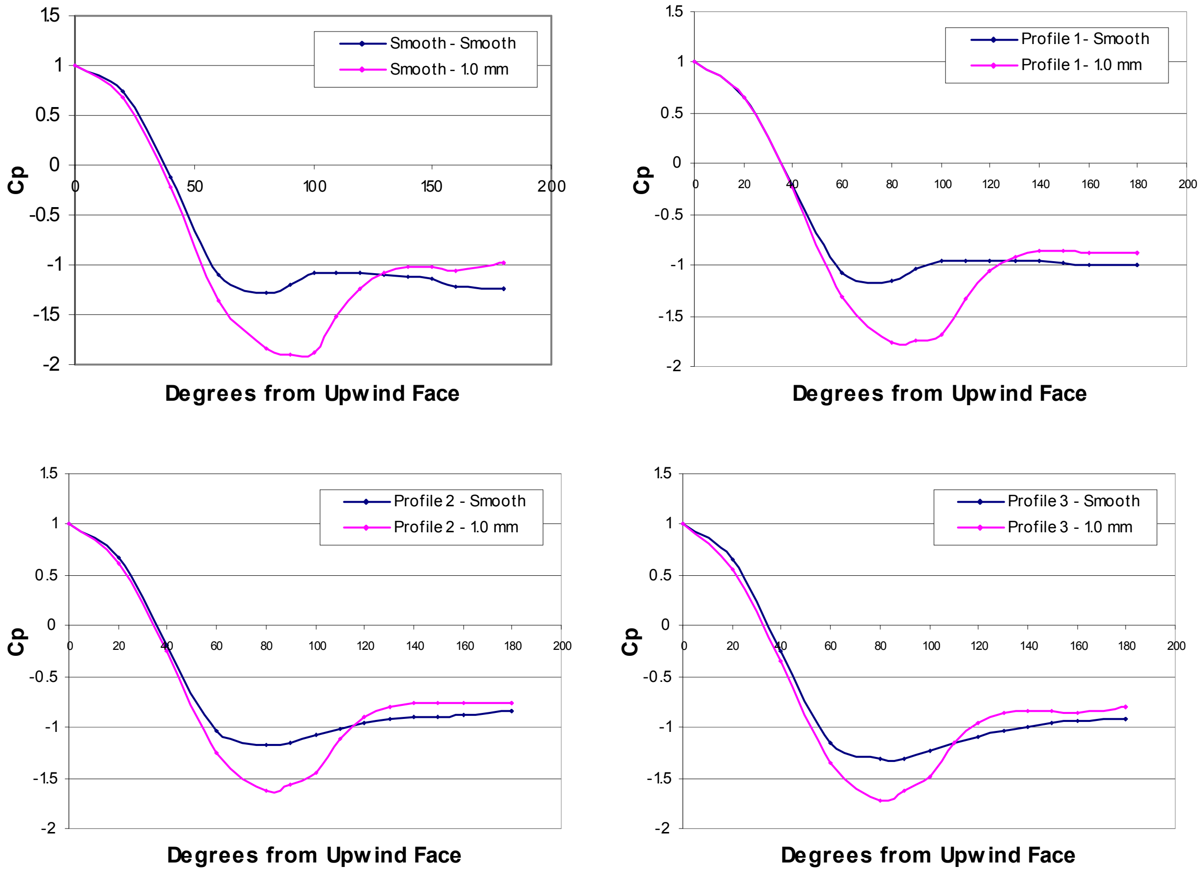


Figure 10 – Mean pressure distributions around 57mm cylinders with 1.0 mm surface roughness pattern at 15.2 m/s in with varying turbulence levels.

The four graphs in Figure 10 show that the surface roughness had a near equal effect on the pressure distribution in each of the test profiles. The 1.0 mm trip generated sufficient turbulence over the surface of the cylinder to reduce the turbulent wake region. For this trip height and roughness R_e, the pressure distribution was independent of the turbulence intensity. This is a favourable result, since model tests are generally conducted with turbulence varying over height and wind direction. However, it is still imperative that the roughness produce wind patterns that mimic the full scale effects as well.

As part of an effort to correlate high R_e effect with the artificial roughness applied on the wind tunnel test models, CFD simulations for high R_e ($1.5\text{x}10^6$) and medium R_e ($1\text{x}10^5$) and low R_e ($3.0\text{x}10^4$) simulation were carried out, details of the computation is described in the previous section. The mean C_p values extracted at mid height of the cylinder are shown in Figure 11. Notably the low R_e simulation case provided higher C_p values behind the cylinder (contributes to an increase in the drag forces) compared to the medium and high R_e cases as expected. It is note worthy to point out that the plots for the velocity vectors indicated a relatively narrow wake (Simiu and Scanlan 1996) for high R_e case compared to the medium and low R_e cases as shown in Figure 12.

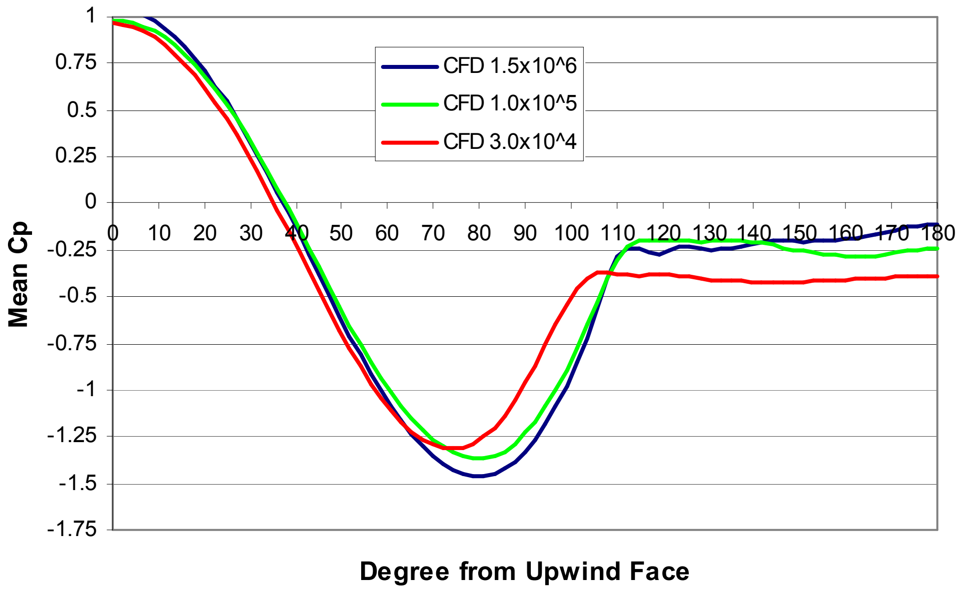
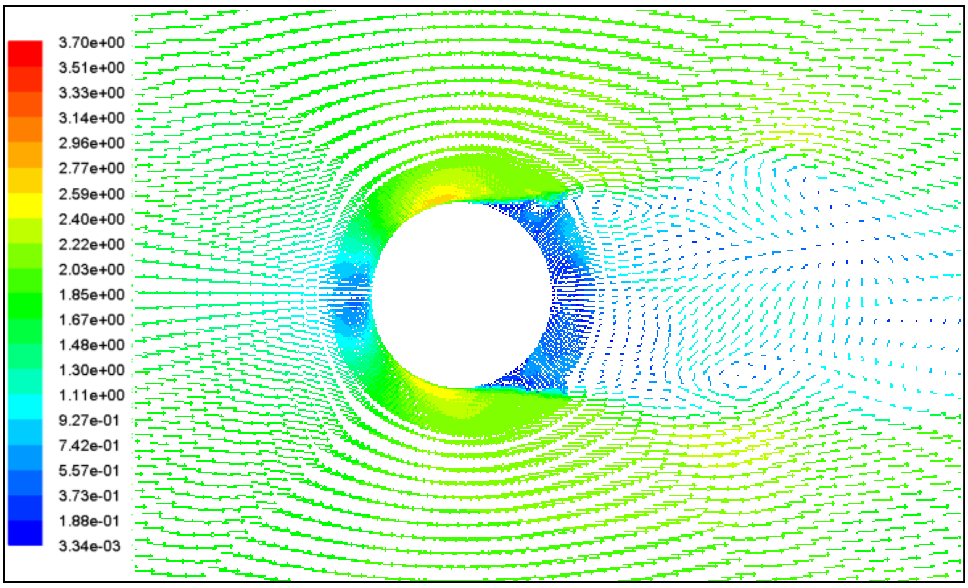


Figure 11 – Numerically (LES) obtained C_p values for High, medium and Low R_e.

(a)



(b)

(c)

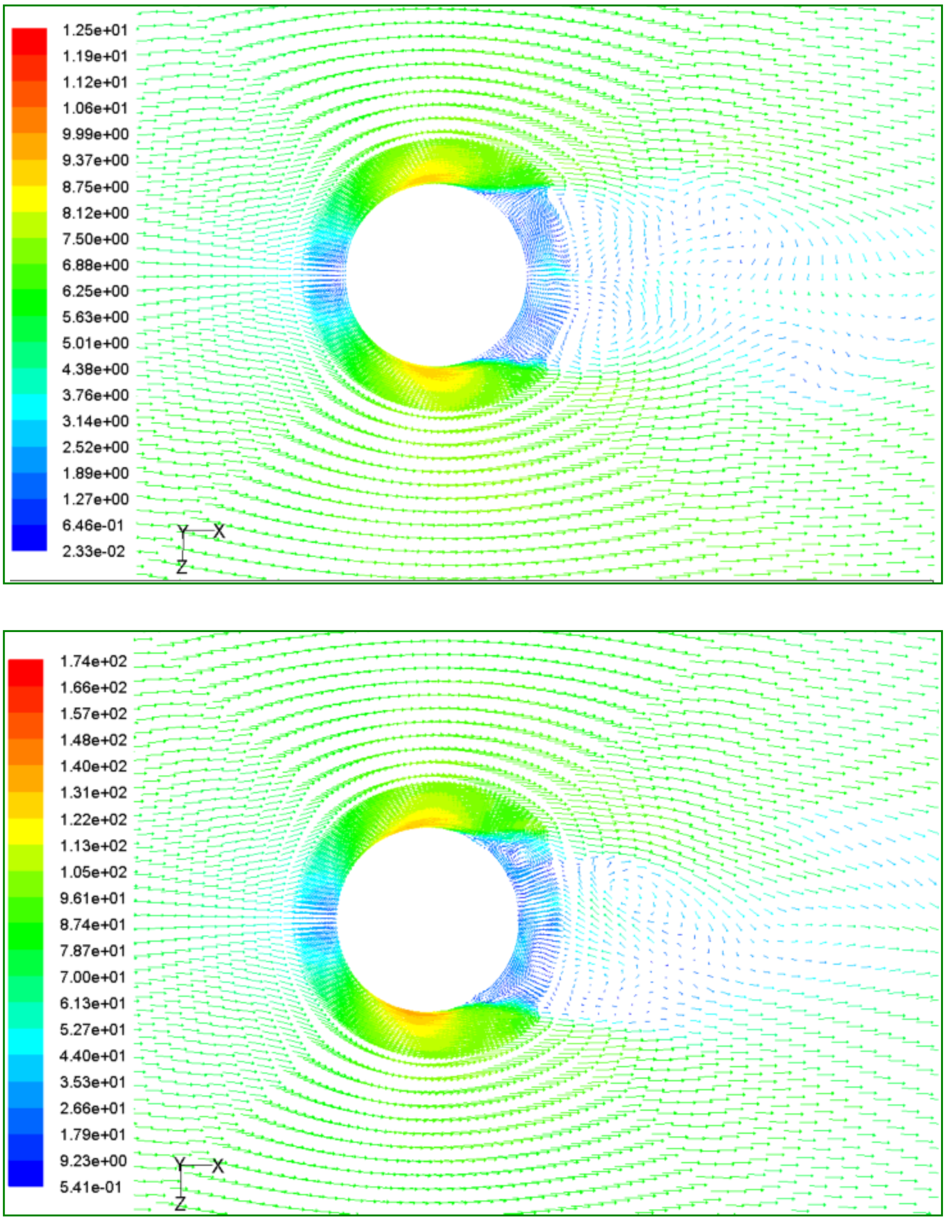


Figure 12 - Velocity vector field, (a) Low R_e case, (b) Medium R_e case and (c) High R_e case. Note that the wind speeds are indicated on each figures.

Comparisons were also drawn between the collected data and previously published experimental data. Scruton and Rogers (1971), summarized works with circular cylinders by several authors, including Roshko (1961). The ESDU 80025 standard also outlines theoretical guidelines for the formulation of mean pressure coefficients (C_p) across a circular cylinder. Figures 13 and 14 compare the pressure distributions on the smooth test cylinders with other literature values in both the sub-critical and super-critical flow regimes. The wind tunnel and CFD data collected in this study show good agreement with the published data.

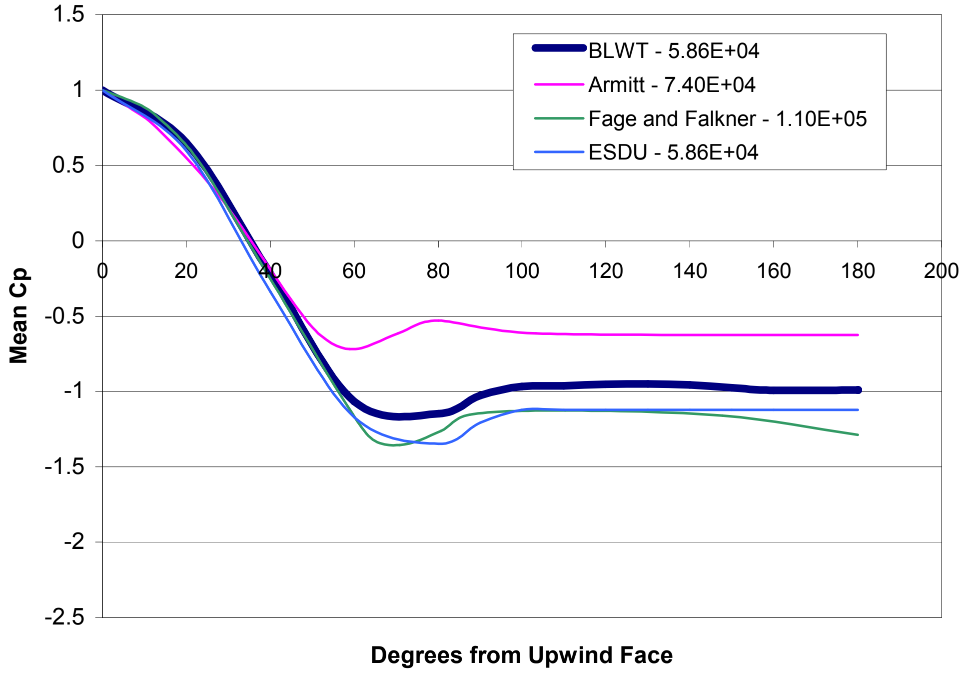


Figure 13 – Mean sub-critical pressure distribution over circular cylinder at various R_e.

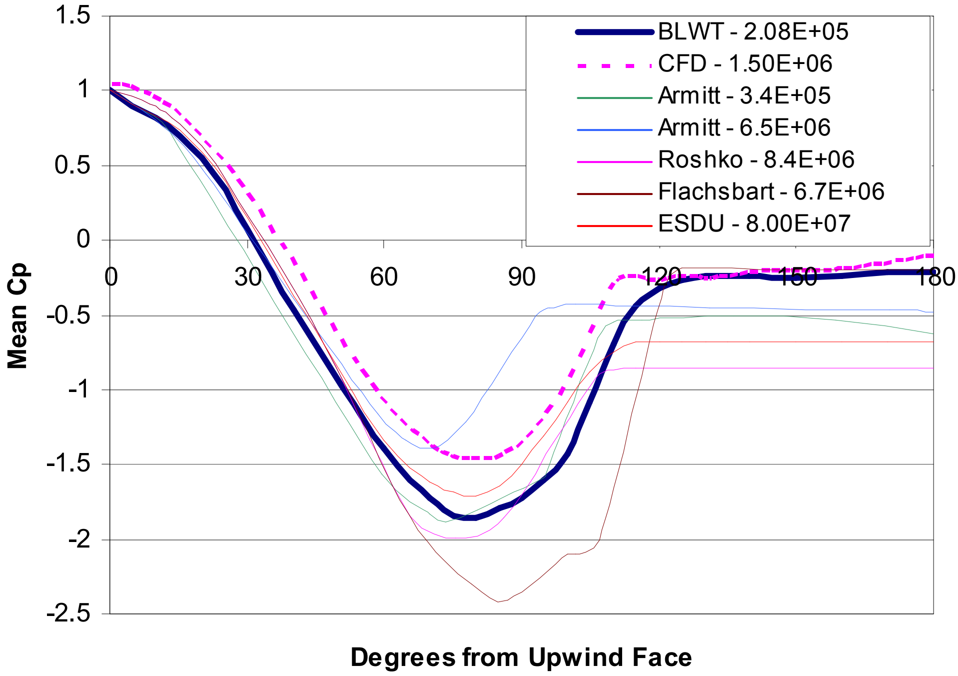


Figure 14 – Mean super-critical pressure distribution over circular cylinder at various R_e.

To validate the effectiveness of the surface roughness pattern, it must be benchmarked against the available super-critical flow distributions. Figure 15 shows the wind tunnel test data for the 57 mm cylinder in profile 1 with 1.0 mm roughness and no roughness. The CFD pressure distribution and the ESDU calculation for a very high R_e are also included.

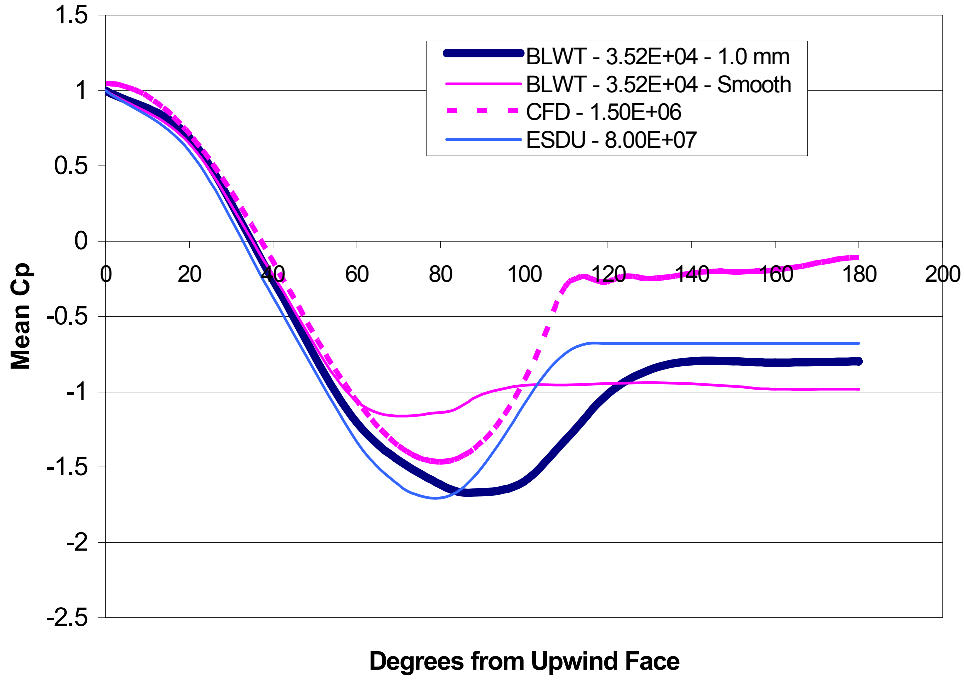


Figure 15 – Comparison of mean pressure distribution over circular cylinders

The pressure distribution with the roughness correlates very closely with the ESDU distribution, whereas the smooth cylinder test yielded results very different than the CFD and ESDU values. Thus, it is evident that the surface roughness can be effective in producing super-critical flow characteristics at sub-critical R_e.

Although this study indicates the potential use of numerical simulation in correlating the high R_e number effects with artificial roughness length, these results need to be validated further with full scale experiments. As a part of these efforts, the authors are undergoing a large scale cylinder testing using the Wall of Wind full scale facility (Gan Chowdury et al. 2007, Leatherman et al, 2008).

CONLCUSIONS AND FUTURE RESEARCH

Use of artificially created surface roughness to compensate for high R_e number effects for circular building testing in low wind speed wind tunnel has been described. An attempt to address the challenge of correlating the high R_e number flow characteristics and surface roughness has been made through numerical simulation with LES turbulence modeling. Previously published research was also presented to support the validity of the results. The data illustrated that the surfaces roughness patterns tested were capable of simulating super-critical flow characteristics at sub-critical R_e. Although no conclusions were drawn about which surface roughness pattern is optimal, research is ongoing to validate the use of surface roughness for scale model testing.

This study opens various doors for future research. Full scale testing of a large circular cylinder at high wind speeds is planned at the Wall of Wind testing facility. In addition, other quantities, such as the drag coefficient and Strouhal number, could be extracted from the experimental data and compared with CFD simulations and full scale tests. Finally, the applicability of adding surface roughness to test models that are not perfect cylinders is necessary. This would include the applicability of surface roughness on curved and rough-walled surfaces for the purposes of scale model testing.

REFERENCES

Bearman, P.W. and J.K. Harvey. (1993), "Control of Circular Cylinder Flow by the Use of Dimples", *AIAA Journal,* **31**, 10, 1753-1756.

Bitsuamlak, G.T., Stathopoulos, T., Bédard, C. (2000),"Progress on numerical simulation of wind loads on buildings", *8th annual conference of the CFD society of Canada (CFD2K)*, Montreal, Canada.

Gan Chowdhury, A., Simiu, E., Leatherman, S.P. (2007), "Hurricane Damage Mitigation of Coastal Houses," Proceedings of the 12th International Conference on Wind Engineering, Cairns, Australia, p. 1975-1982

Davenport, W.J. (2003), "Flow Past a Circular Cylinder", *Virginia Tech University. Online. Internet.* 9 Oct. 2003. Available: www.aoe.vt.edu

Engineering Sciences Data Unit, ESDU, (2007), Data Item 80025. "Mean Forces, Pressures and Flow Field Velocities for Circular Cylindrical Structures: Single Cylinder with Two-Dimensional Flow", *Engineering Sciences Data Unit.*

Fluent Inc., 2003. Fluent 6.2 User's Guide. Lebanon, New Hampshire.

Kimura, T. and M. Tsutuhara. (1991), "Fluid Dynamic Effects of Grooves on Circular Cylinder Surface", *AIAA Journal,* **29**, 12, 2062-2068.

Lakehal, D. (1999), "Computation of Turbulent Shear Flows Over Rough-Walled Circular Cylinders", *Journal of Wind Engineering and Industrial Aerodynamics,* **80**, 47-68.

Leatherman, S., Roberston, C., Simu, E., Choudhury, G., Bitsuamlak, G., Huang, P. (2008),"Full-Scale Destructive Testing of Houses to Hurricane-Force Wind and Rain", Solutions to Coastal Disasters Conference, April 13-16, Oahu, Hawaii.

Nakamura, Y. and Y. Tomonari. (1982), "The Effects of Surface Roughness on the Flow Past Circular Cylinders at High Res", *Journal of Fluid Mechanics,* **123**, 363-378.

Roshko, A. (1961), "Experiments on the Flow Past a Circular Cylinder at Very High Reynolds", Number. *Journal of Fluid Mechanics*. **10**, 345-356.

Scruton, C., and E.W.E. Rogers. (1971), "Steady and Unsteady Wind Loading of Buildings and Structures [and Discussion]", *Philosophical Transactions of the Royal Society of London,* **269**, 1199, 353-383.

Simiu, E., and R. Scanlan. 1996. Wind Effects on Structures. John Wiley & Sons, New York.

Selvam, R.P. (1997), "Finite Modelling of Flow Around a Circular Cylinder Using LES", *Journal of Wind Engineering and Industrial Aerodynamics*. **67** & 68, 129-139.

Szechenyi, E. 1975, "Supercritical *Re* Simulation for Two-Dimensional Flow Over Circular Cylinders", *Journal of Fluid Mechanics,* **70**, 529-542.