THE BeVERLI HILL THREE-DIMENSIONAL SEPARATING FLOW CASE: CROSS-FACILITY COMPARISONS OF TURBULENCE MODELING VALIDATION EXPERIMENT RESULTS

Julie E. Duetsch-Patel[1], Daniel MacGregor[2], Yngve L. Jenssen[3], Pierre-Yves Henry[3], Chittiappa Muthanna[3], Luca Savio[3], Philippe Lavoie[2], Aldo Gargiulo[1], Vignesh Sundarraj[1], Thomas Ozoroski[1], Christopher J. Roy[1], William J. Devenport[1], Aurelien Borgoltz[1], and K. Todd Lowe[1]

Advisors: Todd Lowe and William Devenport

[1]Virginia Polytechnic Institute and State University (Virginia Tech), Blacksburg, VA, USA
[2]University of Toronto, Toronto, ON, Canada
[3]SINTEF Ocean and Norwegian University of Science and Technology, Trondheim, Norway

The BeVERLI Hill project seeks to create a detailed database of the turbulent separated flow over a three-dimensional bump. This project has expanded as part of a NATO collaboration through AVT-349 to study the same model at multiple facilities (Virginia Tech, USA; University of Toronto Institute of Aerospace Sciences (UTIAS), Canada; SINTEF Ocean, Norway) in order to reduce uncertainties in results due to bias effects from a specific facility. Initial qualitative comparison of results across facilities has proven that the unexpected flow phenomena observed, including a bimodal unsteady wake and a steady mean asymmetric wake at two different symmetric angles, are fundamental flow features and not a result of facility effects influencing the flow. Continuing efforts are in place for quantitative comparison of results across facilities.

I. Introduction

The BeVERLI Hill project is an ongoing turbulence modeling validation experiment with the goal of creating a detailed experimental database of the flow over a fifth-degree polynomial profile bump. This project began at Virginia Tech and initial results from small-scale and the first two full-scales test have been previously published (Gargiulo et al., 2020, 2021; Lowe et al., 2020) This project has recently expanded to include collaborative experimental efforts at SINTEF Ocean in Trondheim, Norway and at the University of Toronto Institute for Aerospace Studies (UTIAS) in Toronto, Canada. These cross-facility efforts have come about as part of a collaboration through the Advanced Vehicle Technology (AVT) Panel of the NATO Science and Technology Organization, specifically under AVT-349, *Non-Equilibrium Turbulent Boundary Layers in High Reynolds Number Flow at Incompressible Conditions*.

The flow over a bump serves as a valuable test case for assessing and improving turbulence models, as the simple geometry generates a flow field with pressure gradients, curvature effects, and three-dimensional boundary layer separation and reattachment. All these features are generally challenging for computational fluid dynamics (CFD) and are very common in practical applications. The findings from this research will be particularly relevant for commercial, research, and military applications, as enhanced computational capabilities will reduce testing and design time and support certification-by-analysis efforts to reduce the costs of aircraft development (Bolds-Moorehead & Shikany, 2018).

The flow over variously shaped axisymmetric bumps have been studied for at least the past four decades. Initial studies by Hunt & Snyder (1980), Pearse (1982), and Arya & Gadiyaram (1986) focused upon civil engineering applications, with an emphasis on the effects of local topography on wind patterns close to the ground. Later studies pivoted their focus to CFD validation. One of the most in-depth of these was conducted at Virginia Tech in the 2000s. Byun, Simpson, and colleagues developed a detailed database of flow over a cosine-squared cross-section, axisymmetric bump (Byun et al., 2003, 2004; Byun & Simpson, 2006; Simpson et al., 2002, among others). Bell and

Unabridged paper presented at the 2022 AIAA SciTech Forum as AIAA Paper 2022-0698.

colleagues at NASA conducted further studies on the flow over another axisymmetric bump, called the FAITH hill model (Bell et al., 2012, 2016).

While all these studies generated detailed data describing the flow over their respective models, no database has yet been developed that includes extremely accurate boundary conditions, as-built model and facility geometry, and other key qualitative and quantitative attributes that fully define the physics of a flow. Detailed quantification of these attributes is critical, as any assumptions regarding boundary conditions or inflow conditions propagate through simulations as unknown uncertainties and directly influence the quality of the validation results. Oberkampf & Smith (2017) have proposed a strict documentation process for model validation experiments, describing a set of criteria for experiments designed to rigorously validate computational models and calculations. Based upon the level of documentation, a completeness level ranging from 0 (minimal) to 3 (extremely detailed) is assigned to six attributes and then to the overall project.

No turbulence modeling validation experiment on the flow over a bump has thus far achieved Completeness Level 3, although the ongoing experiments of the flow over the "speed bump" geometry of Williams, Robbins, and colleagues have been recently assessed to reach Completeness Level 2 in some categories (Robbins et al., 2021). As a result, to facilitate landmark CFD validation studies, it is necessary to expand upon these early databases and design a new experimental study to create a second dataset that includes this crucial background information to the highest completeness level. Recent efforts at Virginia Tech (as discussed) and at the University of Washington (Robbins et al., 2021; Williams et al., 2020; Williams et al., 2021) are continuing and expanding upon previous efforts to generate an extremely detailed experimental database of flow over bump profiles.

As noted by Aeschliman & Oberkampf (1998), a recommended procedure in a CFD validation experiment is to conduct the same experiment in different facilities with the same model and same personnel, as this lends confidence that there are no hidden facility-related bias errors in the data. Ideally, this would be conducted using the same model in each facility. Early comparisons of the BeVERLI Hill are not able to use the same model across facilities due to the different facility scales. However, this collaboration across facilities will still allow for comparison of non-dimensional results across at a variety of different Reynolds numbers. The experimental results will also allow for an assessment of the sensitivity of the flow to inflow conditions and model manufacturing error through the different facilities and models.

The primary goal of this study is to understand the impact of facility bias in the flow over the BeVERLI Hill model and demonstrate the benefits of testing the same geometry at multiple facilities and scales. This will be completed by measuring the flow over the BeVERLI hill model at Virginia Tech and SINTEF Ocean and qualitative comparing the results to identify any differences in the flow physics that could be due to facility biases. Future work will include quantitative comparison with wind tunnel data captured by UTIAS with their model. The results from these different facilities will capture additional flow physics on the model and allow for quantification of any facility-related bias errors in results that will be included in the BeVERLI Hill experiment database.

II. Model, Facilities, and Instrumentation

2.1 Model

The nominal geometry of the BeVERLI Hill model is defined by Gargiulo et al. (2020) and shown in Figure 1. Models at all three facilities were manufactured using CNC machining and were subsequently scanned to compare the as-manufactured geometry to design. Due to the different scales of each facility, the bumps were sized differently for each test location, with scales shown in Table 1.

The bump exhibits rotational symmetry every 90° in yaw from a given orientation. The nominal 0° orientation is defined with the $P_5(x)$ profile, shown in Figure 1, aligned with the freestream flow direction. A rotation to a new angle is defined as a rotation about y, such that the 45° orientation is the result of a 45° rotation about the y-axis.

Table 1: Model heights, widths, and height-based Reynolds numbers of study at the three collaboration facilities.

Facility	Height	Width	Re_H
Virginia Tech	7.36 in (0.19 m)	36.80 in (0.91 m)	250,000 325,000 650,000
SINTEF	3.86 in (0.098 m)	17.99 in (0.46 m)	270,000 498,000 605,000
UTIAS	4.72 m (0.12 m)	23.60 in (0.60 m)	81,000 162,000 250,000

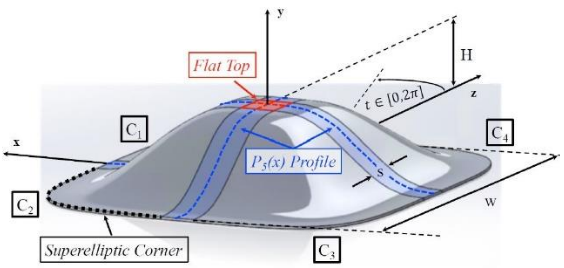


Figure 1: The BeVERLI Hill design geometry.

2.2 Virginia Tech Stability Wind Tunnel

The Virginia Tech Stability Wind Tunnel (SWT) is a continuous, single-return, subsonic wind tunnel, with interchangeable aerodynamic and hybrid-anechoic test sections. Both test sections are 7.32 m long, with a 1.85 x 1.85 m cross section. The maximum flow speed in the test section is approximately 80 m/s, corresponding to a Reynolds number per meter of 5×10^6. The freestream turbulence levels range from 0.0103% at 20 m/s to 0.0229% at 70 m/s. A schematic of the wind tunnel circuit is shown in Figure 2.

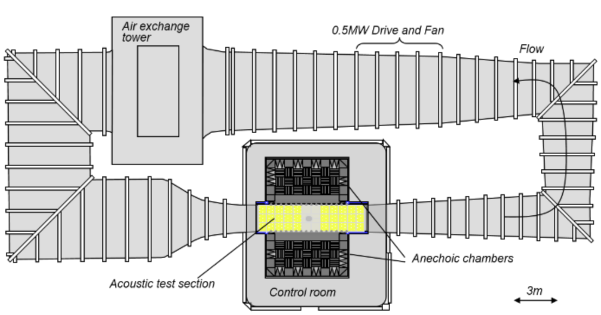


Figure 2: Virginia Tech Stability Wind Tunnel (SWT)

Two BeVERLI hill models were CNC-milled out of tooling foam at SINTEF Ocean for the experimental entries at Virginia Tech: one with 135 pressure taps and one with slots for clear acrylic windows for laser Doppler velocimetry (LDV) measurements. Both models were then painted with black paint and covered with a glossy clear coat for flexibility in locations for oil film interfereomtry (OFI) measurements. These models have been scanned using multiple laser scanning techniques and systems to allow for the geometry to be assessed using several points of comparison. The bump configuration in the SWT is shown in Figure 3.

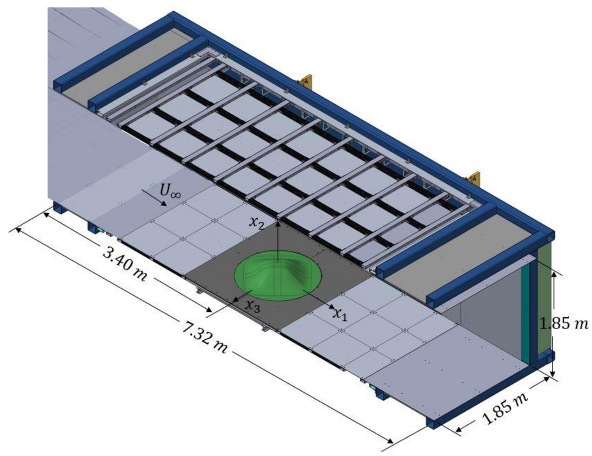


Figure 3: The BeVERLI Hill model mounted in the SWT. Note that to properly show the coordinate system, the side wall on which the bump is mounted is shown as the "floor."

Pressure taps were used throughout the SWT, as well as in one of the BeVERLI models tested, to take detailed static pressure data during each run. A total of ~350 pressure taps were used to collect pressure data throughout the test section and on the bump model. One half of the bump was heavily instrumented. The model was then rotated and measurements across symmetric angles were used to create contours of the full pressure distribution on the model. The pressure coefficient was computed by the data acquisition script as $C_p = (p - p_\infty)/(p_0 - p_\infty)$.

2.3 SINTEF Ocean Cavitation Tunnel

The cavitation tunnel at SINTEF Ocean, shown in Figure 4, is a vertical loop water channel with adjustable working pressure between 10 kPa to 250 kPa. The tunnel features a speed range from 1 to 10 m/s with turbulence levels of approximately

0.5%. The test section is 6 m in length with a cross section of 1.3 x 1.2 m. The bump configuration in the cavitation tunnel is shown in Figure 5. The bump was installed on the false ceiling of the tunnel that was extended downwards to improve optical access to the model for Particle Image Velocimetry (PIV) measurements. A hole in the ceiling extension with a gap of approximately 1 mm enabled connecting the bump to a six-component force balance.

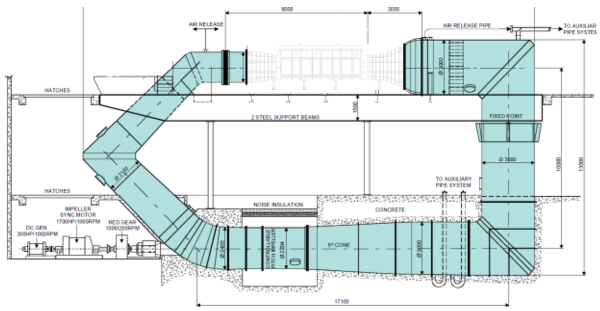


Figure 4: Cavitation Tunnel at SINTEF Ocean.

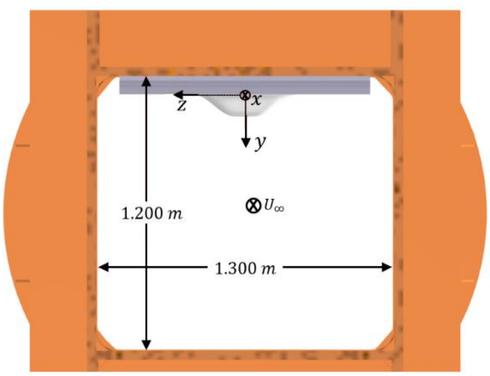


Figure 5: The BeVERLI Hill model mounted in the Cavitation Tunnel at SINTEF Ocean.

The forces acting on the bump were recorded using a six-component balance whose coordinate system is shown in Figure 5. The sampling frequency on the force channels was set to 200 Hz and low-pass filtered at 20 Hz as for all the channels that monitored the status of the tunnel. Forces on the bump were continuously recorded and a feedback signal logged at 9600 Hz from the PIV system enabled synchronizing the two measurements.

One PIV plane in the adverse gradient region of the bump and two optical configurations were investigated at SINTEF Ocean's cavitation tunnel. Seeding was provided by Vestosint silver 50 µm particles added to the water. A LaVision system was used for the PIV tests in the cavitation tunnel and the images analyzed with DaVis software.

2.4 UTIAS Wind Tunnel

The wind tunnel at UTIAS can operate at speeds up to 35 m/s with a freestream turbulence intensity of approximately 0.05% up to 13 m/s, monotonically growing to 0.08% at full speed (configuration shown in Figure 6). The test section is 5 m long with an octagonal cross section that is 0.8 m high and 1.2 m wide. The bump model is mounted 2.5 m downstream from the leading edge of a flat plate that spans the full width and length of the working section and is used to grow a new turbulent boundary layer. The test section with the bump installed is shown in Figure 7.

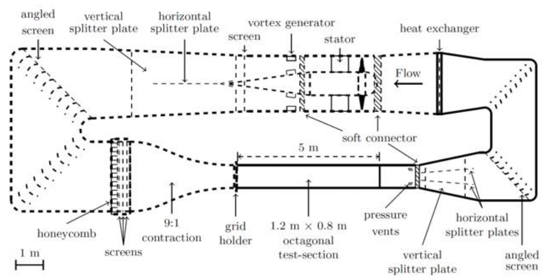


Figure 6: Wind Tunnel at UTIAS.

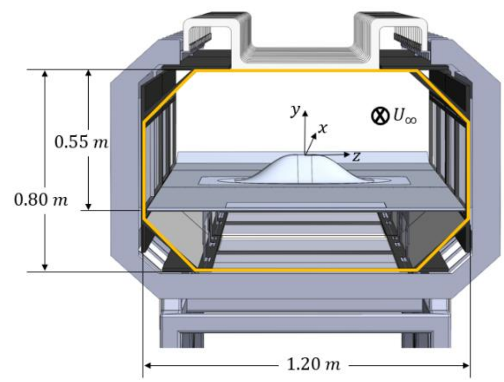


Figure 7: The BeVERLI Hill model mounted in the UTIAS wind tunnel.

The UTIAS BeVERLI model was manufactured from a single block of 6061 aluminum and then acid etch anodized in a matte black finish. The as-manufactured model matches well with as-design geometry, with deviations less than ±0.2 mm.

III. Results

Experiments and comparison efforts are ongoing, and future publications will share additional updates as work continues. Quantitative overlapping measurements across facilities at the same non-dimensionalized locations are still ongoing and will be a primary focus of future publications. Results and insights from facilities which have completed recent wind tunnel tests (Virginia Tech and SINTEF Ocean), which impacted actions and priorities at collaboration facilities, are shared below.

3.1 Virginia Tech

A total of three full-scale wind tunnel experiments have been conducted at Virginia Tech with the goal of documenting the flow over the bump to create a database for CFD validation. As part of this goal, flow visualization using oil mixtures were conducted, in addition to key quantitative measurements of the inflow boundary layer, static pressures over the model, and diagnostic methods, including PIV, LDV, and OFI. The final entry, which concluded at the beginning of November 2021, focused on these final three techniques, and data is still in the processing stage. Preliminary results will be shared here, and final results will be published in future articles and made available through the NASA Turbulence Modeling database.

3.1.1 Unsteady Wake Measurements

The initial planned primary orientation for the BeVERLI Hill experiments, and the primary focus of Entry 1 (February 2020), was the 0^{o} bump orientation. Initial measurements showed asymmetry in the wake pressures, but this was shown to be sensitive to the bump orientation $(0^{\text{o}}, 90^{\text{o}}, 180^{\text{o}}, 270^{\text{o}})$ due to the perturbations in the as-manufactured geometry in the preliminary models used (Gargiulo et al., 2021). Experiments at SINTEF Ocean during the interim period between tests suggested that the wake in this configuration was not stably asymmetric, but rather bimodal.

This was evaluated in the SWT in Entry 2 (May 2021) through the use of unsteady pressure measurements, taken using a DTC ESP 32HD scanner. Unsteady pressures were collected at the 0^{o} and 45^{o} orientations over a period of approximately 60 seconds and compared at symmetric measurements across the centerline in the wake. Because a calibration was not completed before the taps were disconnected at the end of the test, these results are qualitative in nature.

The results for the 0^{o} case clearly indicate in Figure 8a that the asymmetric wake at 0^{o} is not stable, but rather changes sides chaotically as the unsteady pressure on either side of the bump (the

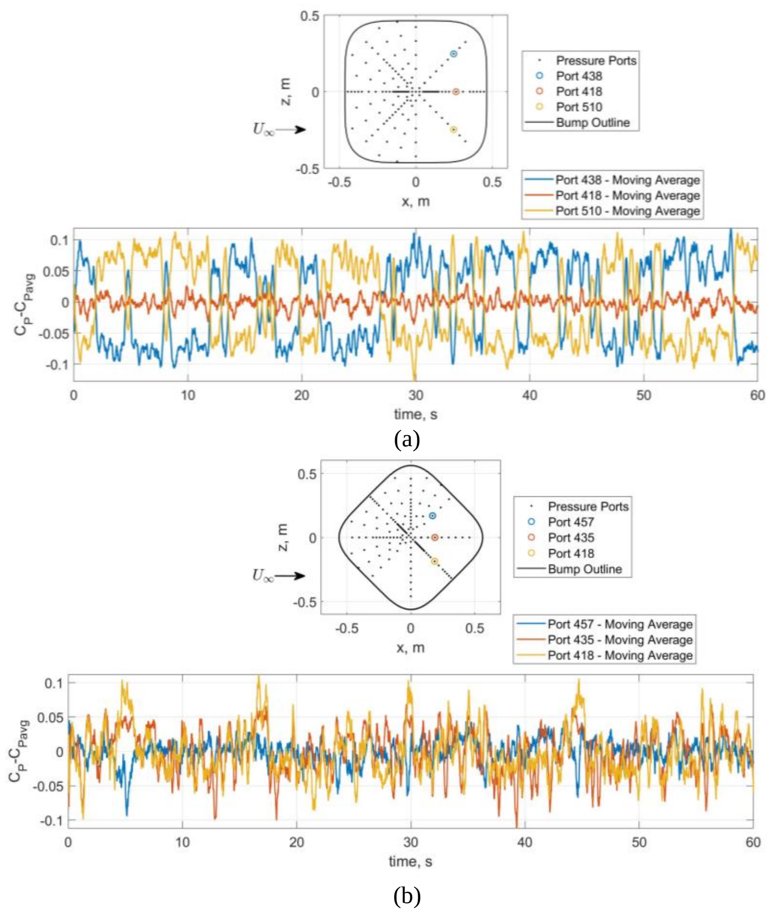
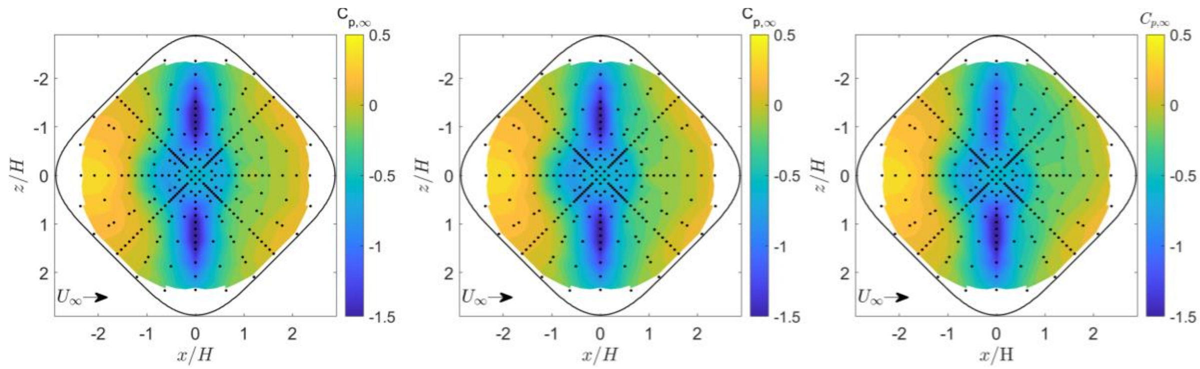


Figure 8: Qualitative (uncalibrated) unsteady pressures at 0º (a) and 45º (b) orientations on the leeward side of the BeVERLI Hill.

blue and yellow curves in Figure 8a) can be seen to be almost exactly out of phase. It was not

As shown, all three Reynolds numbers experience very similar pressure distributions, but a notable



(a) $Re_H = 250K$ (b) $Re_H = 325K$ (c) $Re_H = 650K$

Figure 9: Interpolated static pressure contours over the VT BeVERLI Hill at the three height-based Reynolds numbers studied.

initially clear if this was a facility-bias effect or a real feature of the geometry, but comparisons with SINTEF Ocean's results, detailed in Section 3.2 confirmed that this was a real feature of the geometry.

In contrast, the 45^o case shown in Figure 8b does not display this behavior and appeared more steady, though more asymmetric. Because of this, the 0^o case was de-prioritized due to the importance of time-averaged simulations for this project, and 45^o was chosen as the primary case.

3.1.2 Key Observations

While the 45^o case was shown to be more stable, oil flow visualization images show a clear asymmetry in the wake, as seen in Figure 10. Streamlines in the wake also are slanted down towards the floor of the test section. However, force measurements from SINTEF Ocean also suggested the presence of asymmetry even in the 45^o configuration, suggesting that it is a feature of the bump and not a facility or model bias.

This asymmetry is also visible in the static pressure distributions over the bump surface in Figure 9. The pressure distributions largely matched expectations, with a high-pressure region at the front of the model, a strong favorable pressure gradient along the windward slope and low pressure regions along the center span, and a subsequent region of alternating pressure gradient sign on the top of the bump due to the flat region.

asymmetry develops across the centerspan as the Reynolds number is increased. The pressure distribution across the centerline ($z = 0$) and the centerspan ($x = 0$) are shown in Figure 9. As shown, there are clear Reynolds number effects, particularly above $Re_H = 325{,}000$. There is a noticeable effect on the centerline in the wake at the highest Reynolds number, as well as a sharp change in the centerspan pressure distribution. The pressure along the centerline beyond $x/H \approx 1$ as clearly decreased when $Re_H = 650{,}000$. In addition, clear asymmetry in the region of $-2 \leq x/H \leq -1$ is seen at $Re_H = 650{,}000$.

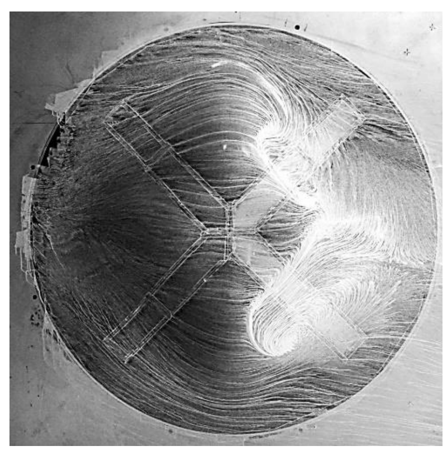


Figure 10: Oil flow visualization in the SWT at $Re_H = 650K$.

LDV, PIV, and OFI data have also been collected on the VT BeVERLI Hill, with the largest batch of data still in the processing phase after a recent 5-

week wind tunnel entry in October 2021. Detailed results will be published in the future.

3.2 SINTEF Ocean

The primary aim of the tests performed at SINTEF Ocean was to identify whether the flow separation in the wake of the bump was stable in the range of Reynolds numbers chosen for the cross-facility comparison. Early tests and simulations revealed that for the rotation angle of 0^{o}, the flow may be bi-stable for high Reynolds numbers. In order to check the hypothesis of bi-stability, the bump was tested in the cavitation tunnel at the angles of 0^{o}, 2^{o}, and 45^{o}. At 0^{o} and 45^{o}, the bump is symmetrically oriented, while the tests at 2^{o} were added to check the resilience of the bistable flow to the incoming flow direction. The bump was mounted on a false ceiling to the top of the tunnel through a force balance that measured the forces and moments acting on the model. The force balance allowed for changing the orientation of the bump without changing the orientation of the force balance with respect to the cavitation tunnel.

For any given orientation of the bump the water speed was varied in steps so that the range of height-based Reynolds number between 186,000 and 614,000 was covered both with positive and negative speed steps. Every time a new water velocity was set, sufficient time was allowed for the flow in the tunnel stabilize and the data recorded over a period of time that would lead to statistically reliable values.

Data was continuously recorded throughout each single test run, leading to time series for the forces that were experienced by the bump at different Reynolds numbers for the drag (x-direction) and lateral force (z-direction). Given the symmetry of the bump at 0^{o}, the lateral force would be expected to average to zero in the case of a stable flow. However, as the speed increases, a larger variability of the force in z-direction can be observed. A closer look to the signal for a constant water speed reveals how the forces in z-direction periodically switched sign. Such switching of the force was not observed in any other direction beside the moment about the x-axis, which is again relative to the switching in of the of the force in z-direction.

In Figure 11, the time trace of the force in x- and z-direction while the water speed was kept constant is shown along with the respective probability distribution functions. The corresponding Gaussian distribution is also plotted for reference. The probability distribution function of the force in the x-direction is well represented by a Gaussian curve, while the force in z-direction

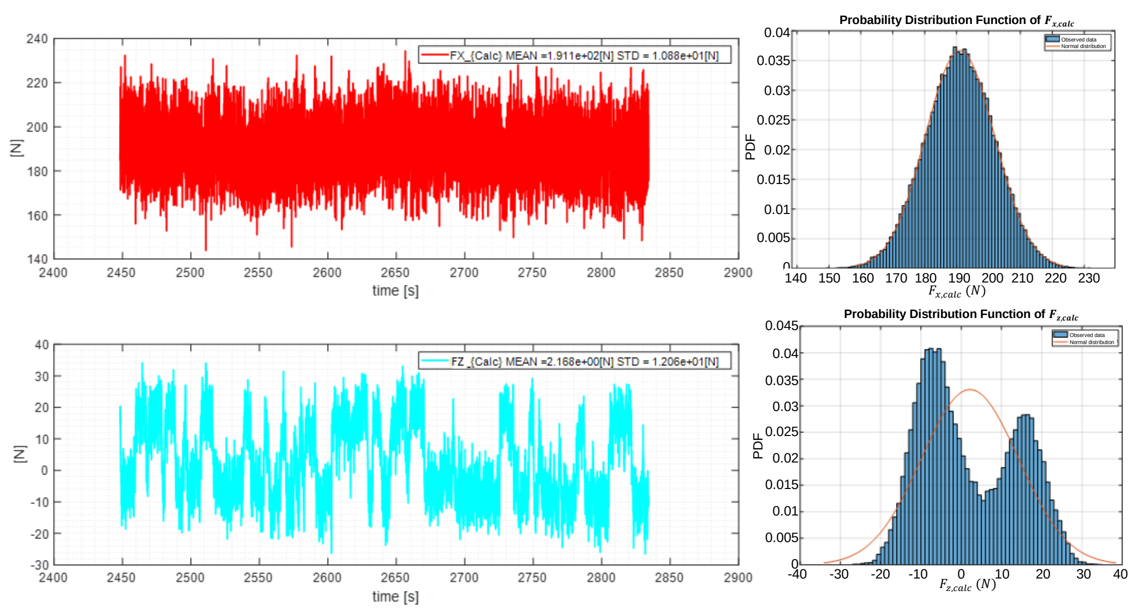


Figure 11: Forces in the x-direction (top) and z-direction (bottom) at $Re_H = 506K$.

shows a bi-modal distribution of probability. Given the low frequency at which the phenomenon of switching happens, the time window over which the water speed was kept constant may have not been long enough to conclude whether the bimodal distribution is symmetric. However, there seems to be a dependency on the Reynolds number in the probability distribution function, where the bimodal behavior is more prominent at high Reynolds numbers.

To confirm that the oscillation of the force in z-direction are to be attributed to a bimodal flow, PIV measurements were carried out on the bump at the 0^{o} orientation. During these confirmatory tests, attention was paid to timing the PIV measurements so that the lateral force was consistently either positive or negative. In Figure 13, the out of plane components (V_z) of the velocity during two opposite oscillations of the flow are shown.

The oscillations of the flow were observed also when the bump was rotated by $+2^{\text{o}}$. The two 2^{o} rotations were originally planned to qualitatively check the sensitivity of the flow to a possible misalignment of the model. These tests proved to be useful to confirm that the flow oscillations are a resilient feature of the 0^{o} orientation.

These results match qualitatively with the results from Virginia Tech - the wake asymmetry for the 0^{o} configuration is not steady and flips orientations chaotically throughout the sampling time. The Strouhal numbers from both experiments were estimated using the width of the bump models, the freestream flow velocity, and the approximate frequency of the oscillations. The Strouhal number for SINTEF was estimated to be approximately 0.003 based on an approximate frequency of 0.05 Hz, while the Strouhal number for Virginia Tech was estimated to be 0.003 based on an approximate frequency of 0.2 Hz.

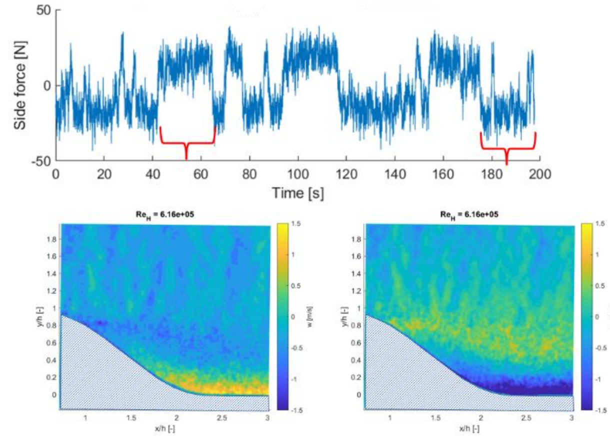


Figure 13: Out-of-plane velocity (V_z) for $Re_H = 616K$.

Measurements were also performed at 45^{o} which gives again a symmetrical inflow to the bump. Instabilities in the forces were observed also at this inflow angle for the higher velocities as shown in Figure 12. However, the instabilities are different in nature from those seen at 0^{o} and $+2^{\text{o}}$ rotation angles. The slow and periodic switching of the forces in z-direction from positive to negative is not present but instead the force is seen experiencing short lived bursts. Further, while the periodic oscillation was seen only in z-direction, and consequently the corresponding moment about the x-axis, for 0^{o} and $+2^{\text{o}}$, the measured force experiences simultaneous bursts in both x- and z-directions as illustrated in Figure 12.

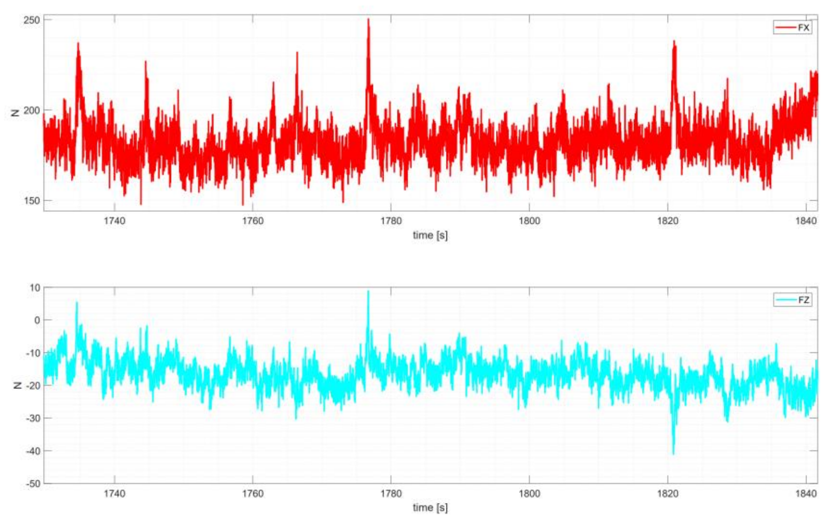


Figure 12: Forces in the x- (top) and z-direction (bottom) for the bump at 45^o.

3.3 UTIAS

Experiments are being conducted at UTIAS to expand the Reynolds number range of the test cases to include 80,000, 165,000, and 250,000. The primary hill orientations being tested are 45^{o} and 0^{o}, which match with the test cases from Virginia Tech and SINTEF Ocean. The highest Reynolds number of 250,000 overlaps with the lowest test case conducted by Virginia Tech, which will enable quantitative comparison of flow features between the two facilities. Planar and stereo particle image velocimetry are being collected to compare against the mean flow statistics measured by Virginia Tech. In place of laser Doppler velocimetry, the UTIAS campaign is using hot-wire anemometry (HWA) and PIV to collect the near-wall velocity profiles. Flow visualization tests are also being conducted to generate surface streamline mappings over the model surface. These vector fields will also be used in the analysis of oil-film interferometry data to calculate local skin friction results.

The UTIAS experimental campaign is also exploring the flow asymmetry observed by Virginia Tech and SINTEF in more detail. The presence of the asymmetry will be verified by monitoring the dynamic pressure fluctuations on the leeward region of the model using ENDEVCO piezoresistive pressure transducers. Mean pressure data from the surface taps of the model will be integrated to determine the forces experienced by the model and compared against those measured by SINTEF Ocean and Virginia Tech.

IV. Conclusion and Future Work

A CFD validation effort studying the flow over a 3D bump that began at Virginia Tech has expanded to become an international collaboration across continents and experimental facilities, with collaborators at SINTEF Ocean in Norway and UTIAS in Canada. This collaboration was undertaken for more rigorous uncertainty quantification efforts, specifically to quantify facility-bias impacts in flow measurements, but the initial collaboration efforts have also benefited the research effort by yielding new information about the flow physics taking place over the bump geometry. This new information has directly influenced decisions on the project level, with one key decision being to focus upon the 45^{o} case as a primary focus due to the more stable asymmetric wake at this orientation. This decision was directly due collaborative efforts between VT and SINTEF Ocean to analyze the bimodal wake of the hill in the 0^{o} configuration, utilizing force measurements, PIV, and unsteady pressure measurements. Another key flow phenomenon identified is the stable asymmetry in the wake at 45^{o} as a function of increasing Reynolds number (through oil flow visualization and wall-static pressures on the bump surface at Virginia Tech and force measurements at SINTEF Ocean).

Efforts to directly compare data across facilities will continue as experimental entries are completed and data is processed. Future goals include comparing measurements at the same (x/H, y/H, and z/H) locations across facilities to directly evaluate any facility-bias effects upon the measured results, and continuing collaborations and discussions to identify key flow phenomena taking place over this geometry. A large quantity of data on the BeVERLI Hill model flow in the Virginia Tech has been collected, including LDV at 13 locations as close as 50 micrometers to the surface of the model, 30 TB of time-resolved PIV data, and direct measurements of skin friction via oil film interferometry. This data will quantify the flow over the model in more detail, allowing for comparison with data collected at other facilities.

In addition, testing is ongoing using the UTIAS model in the Virginia Tech SWT to evaluate the flow over the same geometry and model in different facilities. This will primarily focus on collecting mean and unsteady pressures at symmetric rotations for direct comparison with Virginia Tech data.

Acknowledgements

This work was supported by NASA under grant 80NSSC18M0146.

<u>Literature Cited</u>

Aeschliman, D. P., & Oberkampf, W. L. (1998). Experimental Methodology for Computational Fluid Dynamics Code Validation. *AIAA Journal*, *36*(5), 733–741. https://doi.org/10.2514/2.461

Arya, S. P. S., & Gadiyaram, P. S. (1986). An experimental study of flow and dispersion in the wakes of three-dimensional low hills. *Atmospheric Environment*, *20*(4), 729–740. https://doi.org/10.1016/0004-6981(86)90187-3

Bell, J. H., Heineck, J. T., Zilliac, G., Mehta, R. D., & Long, K. R. (2012). Surface and flow field measurements on the FAITH hill model. *50th AIAA Aerospace Sciences Meeting*, AIAA 2012-0704. https://doi.org/10.2514/6.2012-704

Bell, J. H., Heineck, J. T., Zilliac, G., Mehta, R. D., & Long, K. R. (2016). *Experimental Investigation of Subsonic Turbulent Boundary Layer Flow Over a Wall-Mounted Axisymmetric Hill*.

Bolds-Moorehead, P., & Shikany, D. (2018). Aircraft Certification by Simulation. *Royal Aeronautical Society Flight Simulation Conference*.

Byun, G., & Simpson, R. L. (2006). Structure of Three-Dimensional Separated Flow on an Axisymmetric Bump. *AIAA Journal*, *44*(5), 999–1008. https://doi.org/10.2514/1.17002

Byun, G., Simpson, R. L., & Long, C. H. (2003). A study of vortical separation from three-dimensional symmetric bumps. *41st Aerospace Sciences Meeting and Exhibit*, AIAA 2003-651.

Byun, G., Simpson, R. L., & Long, C. H. (2004). Study of Vortical Separation from Three-Dimensional Symmetric Bumps. *AIAA Journal*, *42*(4), 754–765. https://doi.org/10.2514/1.1829

Gargiulo, A., Duetsch-Patel, J. E., Ozoroski, T. A., Beardsley, C. T., Vishwanathan, V., Fritsch, D. J., Borgoltz, A., Devenport, W. J., Roy, C. J., & Lowe, K. T. (2021). Flow Field Features of the BeVERLI Hill Model. *SciTech 2021 Forum*, AIAA 2021-1741.

Gargiulo, A., Vishwanathan, V., Fritsch, D. J., Duetsch-Patel, J. E., Szoke, M., Borgoltz, A., Devenport, W. J., Roy, C. J., & Lowe, K. T. (2020). Examination of Flow Sensitivities in Turbulence Model Validation Experiments. *AIAA SciTech 2020 Forum*, AIAA 2020-1583. https://doi.org/10.2514/6.2020-1583

Hunt, J. C. R., & Snyder, W. H. (1980). Experiments on stably and neutrally stratified flow over a model three-dimensional hill. *Journal of Fluid Mechanics*, *96*(4), 671–704. https://doi.org/10.1017/S0022112080002303

Lowe, T., Beardsley, C., Borgoltz, A., Devenport, W. J., Duetsch-Patel, J. E., Fritsch, D. J., Gargiulo, A., Roy, C. J., Szoke, M., & Vishwanathan, V. (2020). Status of the NASA/Virginia Tech Benchmark Experiments for CFD Validation. *AIAA SciTech 2020 Forum*, AIAA 2020-1584. https://doi.org/10.2514/6.2020-1584

Oberkampf, W. L., & Smith, B. L. (2017). Assessment Criteria for Computational Fluid Dynamics Model Validation Experiments. *Journal of Verification, Validation and Uncertainty Quantification*, *2*. https://doi.org/10.1115/1.4037887

Pearse, J. R. (1982). Wind Flow Over Conical Hills in a Simulated Atmospheric Boundary Layer. *Journal of Wind Engineering and Industrial Aerodynamics*, *10*, 303–313. https://doi.org/10.1016/0167-6105(82)90004-6

Robbins, M. L., Samuell, M., Annamalai, H., & Williams, O. J. (2021). Overview of validation completeness for gaussian speed-bump separated flow experiments. *AIAA SciTech 2021 Forum*, AIAA 2021-0969. https://doi.org/10.2514/6.2021-0969

Simpson, R. L., Long, C. H., & Byun, G. (2002). Study of vortical separation from an axisymmetric hill. *International Journal of Heat and Fluid Flow*, *23*(5), 582–591. https://doi.org/10.1016/S0142-727X(02)00154-6

Williams, O. J., Samuell, M., Robbins, M. L., Annamalai, H., & Ferrante, A. (2021). Characterization of separated flowfield over Gaussian speed-bump CFD validation geometry. *AIAA SciTech 2021 Forum*, AIAA 2021-1671. https://doi.org/10.2514/6.2021-1671

Williams, O., Samuell, M., Sarwas, S., Robbins, M., & Ferrante, A. (2020). Experimental study of a CFD validation test case for turbulent separated flows. *AIAA SciTech 2020 Forum*, AIAA 2020-0092.