Numerical Computation of Pressure Distributions over Conical Forebody at High Angles of Attack

Yu Yang\(^1,\)*, Borui Zheng\(^2,\), Wei Yang\(^1,\)\(^\dagger\), Chao Gao\(^2,\)\(^\ddagger\)
\(^1\)AVIC ChengDu Aircraft Design & Research Institute, Cheng du, 610041, China
\(^2\)Northwestern Polytechnical University, Xi’an 710072, China

Pressure distributions over a 20° conical forebody are computed by the commercial CFD software ANSYS CFX. Free stream velocity is 30 m/s, Reynolds number based on the forebody base diameter is 0.3×10\(^6\), and the angle of attack ranges from 10° to 35°. The C-O structural grids are used to compute the entire flow field. The governing equations are steady Reynolds-Averaged Navier-Stokes Equations for turbulent flow, and steady Navier-Stokes equation for laminar flow. Different grids are investigated at the angle of attack of 35° for turbulent flow. The grid of 161×121×360 (axial × radial × circumferential) is chosen by the consideration of grid independence. The numerical solutions are compared with experimental data available in the literatures. The numerical results using laminar Navier-Stokes equations compare favorably with experimental measurements and the computation of transitional Navier-Stokes equations is required.

I. Introduction

The high angle of attack aerodynamics of a symmetric body under symmetric flight conditions is problem of both academic interest and practical significance because the symmetric body can produce an asymmetric flow and hence experience a side force which directly affects the maneuverability of an aircraft or missile. A great deal of experimental, theoretical, and computational effort has been spent regarding the understanding, prediction, and control of the vortex asymmetry. The subject has been reviewed by Hunt,\(^1\) Ericsson and Reding,\(^2\) and Champigny.\(^3\)

The basic physical mechanism of the force asymmetry is not clear. At least two possible causes for the force asymmetry were suggested mainly based on experimental investigations: (1) inviscid hydrodynamic instability of the symmetrically separated vortices (Keener and Chapman)\(^4\) and (2) asymmetric flow separation and/or asymmetric flow re-attachment on each side of the body (Ericsson).\(^5\) There is at present no general agreement on the mechanism involved in the creation of the force asymmetry.

There seems little doubt that the apex of the pointed slender body plays a decisive role in determining the flow pattern over the entire body. Since the pointed nose is locally conical in shape, the flow may be regarded as locally equivalent to that about a tangent cone. The basic features of the asymmetric flow about pointed slender bodies of revolution can be displayed by studying the flows over a circular cone.

Keener et al.\(^6\) measured the forces and moments acting on a cone of semi-apex angle 10° at Reynolds numbers based on base diameter ranging from 0.3×10\(^6\) to 4.6×10\(^6\) and Mach numbers ranging from 0.1 to 0.7 by a force balance in a large subsonic wind tunnel. Angle of attack was varied from 0° to 88° at zero sideslip. The cone with

---

* Graduate Student, Department of General Design.
† Graduate Student, Department of Fluid Mechanics.
‡ Researcher, General Office.
§ Professor and Vice Director, National Key Laboratory of Science and Technology on Aerodynamic Design and Research.
# Professor, Department of Mechanical and Aerospace Engineering, Associate Fellow AIAA.
** Researcher, Department of Mechanical and Aerospace Engineering.
pointed nose experienced steady side force at high angles of attack at zero sideslip. A large side force developed at angles of attack between 20° and 75°. The side force changed from side to side as the angle of attack increased and was accompanied by dynamic oscillations. The direction and magnitude of the side force was sensitive to the body geometry near the nose. The maximum side force reached about 0.9 times the normal force. The angle of attack of onset of side force was about 20° and not strongly influenced by Reynolds number or Mach number.

Lowson et al.\textsuperscript{7} reported the flow asymmetry development over circular cone as function of angle of attack, semi-apex angle and wind speed by smoke-laser-sheet visualization. The cone models had semi-apex angle of 5°, 10° and 20°. The flow over the 5° cone could be seen to be essentially conical at high angles of attack. For the 20° cone, the separated flow was found to be non-conical at lower angles of attack.

Fiddes et al.\textsuperscript{8} reported that a pressure measurement of a nose cone-fairing segment-after cylinder model had been conducted in the RAE 5 m low-speed pressurized wind tunnel. The nose cone has 10° semi-apex angle. The model was rolled in 10° intervals through the entire roll range. There were six pressure-tapping stations in which four stations were located on the nose cone and the rest two stations on the fairing segment. Each station housed 36 holes equally-spaced around the circumference. Part of the data on the front station, 148.5 mm from the apex at 35° angle of attack was reproduced in Ref. 8.

An RAE unpublished force measurement by A.R.G. Mundell (1982) was reported in Ref. 8. A large side force was obtained for a 20° cone at a high angle of attack, zero sideslip and low speed by a balance, while the separation positions were not greatly asymmetric as recorded by oil flow method.

Water flow about a circular cone of 12° semi-apex angle inclined at 16° is seen to be sensibly conical.\textsuperscript{9} The slender conical flow of an incompressible inviscid fluid has been analyzed by many authors. Dyer et al.\textsuperscript{10} found non-unique stationary (symmetric and asymmetric) vortex-pair positions even when symmetric separation positions are postulated with respect to the incidence plane. The stability of the stationary solutions were studied by Pidd et al.\textsuperscript{11} and Cai et al.\textsuperscript{12, 13} Pidd et al. showed that the stationary conical solutions are convectively stable under certain conditions. Cai et al. found that no stationary conical solutions are stable based on the global stability analysis.

Meng et al.\textsuperscript{14} reported that a pressure measurement of a circular cone-cylinder model had been conducted in a low-turbulence 3.0 × 1.6 m low-speed wind tunnel. The semi-apex angle of the cone is 10°. The results consist of detailed pressure distributions over nine stations along the cone at angles of attack of 0 ~ 35°, Ma = 0.09, and Re = 0.3 × 10\textsuperscript{6} based on the cone base diameter. The local and overall forces and moments are calculated from the measured pressures. Three angles of attack stages of side force variation with roll angle had been classified in the range of 0° ≤ α ≤ 35°. And their results are agreed well with the measurements for a 20° cone performed by Keener et al.\textsuperscript{15}

Jia et al.\textsuperscript{16} reported the results consist of detailed pressure distributions over nine stations on the cone at 35° angle of attack and Reynolds number of 0.9 × 10\textsuperscript{6} based on the diameter of the cone base, and the wind tunnel and model are the same as Ref. 14. Their results show good agreement with the characteristic pressure obtained by Fiddes et al.\textsuperscript{8}

To the authors’ knowledge, there has little investigation on pressure measurement of conical body as the flow field changing from symmetrically separated flow to asymmetrically separated flow, so we choose the pressure distributions in Ref. 14 as the comparison object, whose experimental data are also available to us. The flow conditions of α=0 ~ 35°, Re = 0.3 × 10\textsuperscript{6} in Ref. 14 are in the transition section between laminar and turbulent flow, and the flow seems more close to the laminar, which can be predicted by the flow regime classification of Lamont\textsuperscript{17}.

In the current work, a numerical investigation using 3-D Reynolds-averaged Navier–Stokes solver has been done to compare with the experimental results in Ref. 14. Comparisons have been made between the computational results and pressure distributions at the same free stream velocities and angles of attack. Results show that the numerical computations can predict the flowfield from stable to bi-stable regimes on the circular cone-cylinder model at typical flow conditions, which agrees well with the experimental results. In the following sections, the numerical method is described and employed to compute the flowfield around the cone-cylinder model. The computational results are compared with the experimental pressure distributions and then conclusions are drawn.

II. Numerical Method

A. Computational Software CFX

All CFD computations in the present paper have been performed with ANSYS CFX. Simulations were performed using both the laminar Navier-Stokes equations and fully turbulent formulation.

ANSYS CFX is an integrated software system capable of solving diverse and complex three-dimensional fluid flow problems. The multigrid based fluid flow solver provides solutions for incompressible or compressible, steady-
state or transient, laminar or turbulent, single- and multiphase and thermal fluid flow in complex geometries. The software uses unstructured and block-structured non-orthogonal grids with grid embedding and grid attaching to discretize the domain.

ANSYS CFX utilizes a finite-volume based, unstructured, parallelized, coupled, algebraic, multi-grid solver with a second order advection scheme with second order overall accuracy. The computations have been performed with the incompressible version of the Reynolds Averaged Navier-Stokes (RANS) equations and the shear-stress transport (SST) turbulence model. A control volume is constructed around each nodal point of the mesh and the fluxes are computed at the integration points located at the sub faces between two control volumes. The discrete equations are evaluated using a bounded high resolution advection scheme similar to that of Barth and Jesperson. The mass flow is evaluated such that a pressure-velocity coupling is achieved by the Rhie and Chow algorithm.

B. Computational Grids

A computational model of a circular cone-cylinder, which is the same size as Meng et al., has been adopted to validate the results produced in the wind tunnel. The difference is that the aft circular cylinder in the present work extends to the downstream farfield boundary and the computational model is smoother.

The accuracy of a numerical solution is directly related to the grid density. Four C-O structural grids have been constructed in Grid Tools Gridgen by specifying the boundary conditions, number of points, distance into the far field, initial and end spacing, and smoothing parameters. The volume grid resolution is shown in Fig. 1. The grid points are uniformly distributed in the circumferential direction, gradually stretched in the axial direction, and clustered in the radial direction to concentrate near body surface to capture the viscous flow. The first axial grid spacing at the conical apex is 0.1% of the conical section length, and hyperbolic tangent function is adopted to stretch subsequent mesh points in axial direction with a stretch ratio of 1.1. The first radial grid spacing away from body surface is 0.001 mm, which is $10^{-5}$ times the radius of the cylinder section of the cone-cylinder body, and subsequent grid points outward from the body surface is stretched by hyperbolic tangent function with a ratio of 1.1. Far field boundary in the radial direction is 40 times of cylinder’ radius and the cylinder stretch to the downstream outlet boundary in the axial direction.

(a) Meridian plane grid
(b) Cross section grid
(c) Surface grid

Figure 1. Local of C-O computational structured grid ($161 \times 121 \times 120$)

C. Computational Residuals

Because the pressure distributions of conical forebody are very complex, so the grids in the circumferential direction should be dense enough. The tests for different axial and radial number have also been conducted, but the results change little. Four sets of grid have been adopted, and we define $161 \times 121 \times 120$, $161 \times 121 \times 240$, $161 \times 121 \times 360$ and $161 \times 121 \times 480$ as Grid 1 – 4 for brevity.
Figure 2. The residuals of the C-O structural grids with SST turbulence model at 30 m/s, $\alpha = 35^\circ$, Station 3

Table 1. The volume grid resolution and the reduced order of residuals of the four C-O structural grids with turbulence model at 30 m/s, $\alpha = 35^\circ$

<table>
<thead>
<tr>
<th>Name</th>
<th>Axial $\times$ Radial $\times$ Circumferential</th>
<th>Reduced order of Residuals</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>$161 \times 121 \times 120$</td>
<td>3</td>
</tr>
<tr>
<td>Grid 2</td>
<td>$161 \times 121 \times 240$</td>
<td>2</td>
</tr>
<tr>
<td>Grid 3</td>
<td>$161 \times 121 \times 360$</td>
<td>2</td>
</tr>
<tr>
<td>Grid 4</td>
<td>$161 \times 121 \times 480$</td>
<td>2</td>
</tr>
</tbody>
</table>

In the present computations the flow at the far field boundary of the computational grid is assumed to be the undisturbed freestream condition. The initial flow conditions are simply the freestream flow. The no-slip condition is applied on the body surface. Steady state applications are computed by time stepping iteration until a user defined convergence level is reached. The Reynolds stresses in the momentum equations are computed using an automatic wall treatment. For Grid 1 to calculate turbulent flow, typically a maximum number of 1,000 iteration steps or 12 hours of eight CPUs of a Linux cluster are needed to make the residuals to reduce 2 to 3 orders, which is approximate as Ref. 22. For Grid 3, typically 500 iteration steps are enough for the residuals to reach the same level. Several runs were carried to 10,000 iterations to check for any divergences in the flow. None were discovered. As the number of grid points increases, the time necessary to get the solution also increases. The cases have also been calculated by laminar Navier-Stokes equations, and the residuals reduce two orders of magnitude, but the vibration is more violent than cases calculated by turbulence model.

Figure 3. The residuals of the C-O structural grids with laminar Navier-Stokes equations at 30 m/s, $\alpha = 35^\circ$, Station 3
D. Grid Independence

The analysis of grid independence has been performed with the incompressible version of the Reynolds Averaged Navier-Stokes (RANS) equations and the shear-stress transport (SST) turbulence model. Four sets of grid have been utilized to simulate the flow field at 30 m/s, $\alpha = 35^\circ$. Figure 4 illustrates the result of Grid 1 ($161 \times 121 \times 120$) is symmetry, but the pressure distributions of other grids are asymmetry and nearly coincide. Four grids are compared and Grid 3 ($161 \times 121 \times 360$) is selected as the grid used in the subsequent sections based on the consideration of grid independence, wherein the circumferential grid number is 360 which has more than 240 used in Ref. 23.

III. Experimental Results

All the experimental results are obtained in the NF-3 wind tunnel at the Aerodynamic Design and Research National Laboratory, Northwestern Polytechnical University. The test section has a 3.0 m $\times$ 1.6 m cross section, and a length of 8.0 m. The contraction ratio is 20:1. The free-stream turbulence level is 0.045% for wind speeds of 20 - 130 m/s. The cone-cylinder model is tested at $\alpha = 0^\circ$ ~ $35^\circ$, $V = 30$ m/s, $M = 0.09$ and $Re = 0.3 \times 10^6$.

Figure 5. The cone-cylinder model
The model comprises a nose cone of 10° semi-apex angle faired to a cylindrical afterbody as shown in Fig. 5. The longitudinal distance from the rear pressure-measurement station to the front support axle is approximately twice as large as the measurement length. No flow distortion is detected over the pressure-measurement stations by the presence of the support axle.

The fore-body’s roll orientation is facilitated by clamping the fore-body to the axis of a remotely controlled motor which is housed in the after cylinder. The model can be set at any roll angle between 0 and 351 from the chosen datum in 9° intervals. The accuracy of the roll-angle set is approximately 1%. The pressure instrumentation is confined to the nose cone and is well forward of the model support. The pressure transducers are placed at 9 stations along the model’s axis as shown in Fig. 5. Stations 1 and 2 have 12 and 18 pressure transducers, respectively, and the rest stations have 36 pressure transducers. The pressure transducers in each station are equally-spaced around the circumference and arranged from the same datum for all stations. The total number of the pressure transducers is 282.

![Figure 6. The model in the wind tunnel](image)

Figure 6 shows the model rigidly supported in the wind tunnel. The static pressure at the each pressure transducer is transmitted by a rubber tube passing through the base of the after cylinder to the pressure-measurement system outside the test section. The system consists of 24 scan-valves each of which has 16 channels and one pressure transducer of modulus 9816 of the PSI Company with an accuracy of ±0.05%. The pressure-measurement readings for each test case were taken 115 times in 0.05 s intervals and then time-averaged. The fluctuations of the readings are small and the time-averaged data are meaningful. A thorough job of cleaning the model was done prior to each run of the wind tunnel. The measurements of overall force and moment coefficients for a 20° cone were performed by Meng et al. and Keener et al. at angles of attack from 10° to 35° as shown in Fig. 7.
IV. Comparison of Computational Results with Experimental Data

Because the flow is conical flow, the pressure distributions difference between neighbor sections is very small, and the pressure transducers of station 1 and 2 are not enough for pressure distributions capture, so we take the experimental results of station 3 for comparison. The pressure distributions are predicted by the laminar Navier-Stokes equations and Reynolds averaged Navier-Stokes equations (RANS) with the grid of $161 \times 121 \times 360$, and the computational results are compared with the experimental data in Ref. 14. To match with the computational result pressures, the roll angle of experimental results are chosen, which are 198° at $\alpha = 10^\circ$, 72° at $\alpha = 15^\circ$, 144° at $\alpha = 20^\circ$, 207° at $\alpha = 25^\circ$, 243° at $\alpha = 30^\circ$ and 180° at $\alpha = 35^\circ$.

A. $\alpha = 10^\circ$

Figure 8 shows that the computed laminar and turbulent pressure distributions are nearly coincided. At $\alpha=10^\circ$, the flow on the leeward side of model is almost parallel with the freestream. The adverse pressure gradient is not large enough to make the boundary layer separate. Thus, the boundary layer is attached to the body surface. Since the boundary layers for both laminar and turbulent flows are thin, their effects on the pressure distribution are negligible. Both laminar and turbulent computational results agree well with the experimental data.
Figure 8. Comparison between experimental and computational results at 30 m/s, $\alpha = 10^\circ$, Station 3

B. $\alpha = 15^\circ$

The computed laminar pressure distribution agrees well with the experimental data as shown in Fig. 9. One additional suction peak appears near the symmetry plane on each side of the body. The additional suction peaks indicate that the boundary layers over the two sides of the body are separated there at $\alpha = 15^\circ$. The pressure distribution remains symmetric, which means that the flow separates symmetrically. The laminar results compared favorable with the experimental results. The flow is nearly laminar, rather than turbulent. The computed turbulent result also shows one additional suction peak on each side of the body, but they appear further closer to the symmetry plane.

Figure 9. Comparison between experimental and computational results at 30 m/s, $\alpha = 15^\circ$, Station 3
C. $\alpha = 20^\circ$

Figure 10 illustrates that the flow remains separated and essentially symmetric. The computed laminar pressure distribution depicts two additional pressure suction peaks on each side of the body near the symmetry plane, which agrees favorable with the experimental results. For the computed turbulent result, there appears only one additional suction peak on each side and closer to the symmetry plane.

D. $\alpha = 25^\circ$

The flow remains separated symmetrically at $\alpha = 25^\circ$ as shown in Fig. 11. Two additional pressure suction peaks appear on each side near the symmetric plane in the computed laminar and experimental results, while only one additional pressure suction peak appears on each side and closer to the symmetry plane in the computed turbulent solution.
E. $\alpha = 30^\circ$

Figure 12 shows that the computed laminar and turbulent pressure distributions are essentially symmetric, but the experimental result takes on asymmetry. The computed laminar solution on the port side of the body agrees fairly with the experimental data, while the starboard side solution deviates from the experiments remarkably. The computed turbulent solution deviates from the experiments remarkably.

![Figure 12. Comparison between experimental and computational results at 30 m/s, $\alpha = 30^\circ$, Station 3](image)

F. $\alpha = 35^\circ$

The computed laminar and turbulent pressure distributions are clearly asymmetric as shown in Fig. 13. The asymmetric solutions exhibit the intrinsic hydrodynamic instability of the symmetric separated flow over a slender circular conical body. However, the agreements between the computed and experimental results are not good, although they agree fairly in some parts of the body surface. The flows studied in the present paper may be located in the transitional region, i.e., between the laminar and fully turbulent regions, while closer to the laminar region. The laminar and fully turbulent models have been used in the present computations. A transitional model will be used in our future work.

![Figure 13. Comparison between experimental and computational results at 30 m/s, $\alpha = 35^\circ$, Station 3](image)
V. Conclusions

Floors over a 20° circular cone forebody at angles of attack up to 35°, speed 30 m/s and Reynolds number based on the forebody base diameter of 0.3 million are computed by using a commercial CFD software ANSYS CFX and the C-O structural grids for the entire flow field. The flow is computed as it is assumed laminar and turbulent. The after cylindrical body is extended to far field. The far field boundary is located at 40 times the forebody base diameter. The initial condition is the free stream flow. The computation residuals drop by two to three orders of magnitude. The numerical solutions are compared with experimental data available in the literatures.

At $\alpha = 10^\circ$, the flow is attached. The computed laminar and turbulent pressure distributions are almost identical and agree with the experimental results. It indicates that the viscosity effect on the pressure distribution is small.

At $\alpha = 15^\circ$, the flow is separated symmetrically. The computed laminar pressure distributions agree well with the experimental result. An additional pressure suction peak appears on each side near the symmetric plane. The additional suction may be caused by separated vortex. The computed turbulent pressure distribution differs from them. It means that the flow is close to laminar rather than turbulent.

At $\alpha = 20^\circ, 25^\circ$, the flow is separated and remains essentially symmetric. The computed laminar pressure distribution has two additional suction peaks on each side near the symmetric plane, and its agreement with the experimental result is much better than the computed turbulent pressure distribution.

At $\alpha = 30^\circ, 35^\circ$, the separated flow is asymmetric. Both computed laminar and turbulent pressure distributions at $\alpha = 35^\circ$ are clearly asymmetric. The computed pressures differ from the experimental results, though the computed laminar pressures are closer to the experimental. The disagreement may indicate the flow is transitional, although it is closer to laminar rather than turbulent.

Acknowledgments

The present work is supported by the Specialized Research Fund for Doctoral Program of Higher Education, SPFDP-20080699003, and the Foundation for Fundamental Research of the Northwestern Polytechnical University, NPU-FFR-W018102 and JC201103. The authors would like to thank Professors Jinsheong Cai, Zhengke Zhang and Xuanshi Meng in NPU for their valuable contribution and to acknowledge Peijia Si, Yushuai Wang of Northwestern Polytechnical University and Ya Liu of University of California, Irvine for their technical assistance.

References


American Institute of Aeronautics and Astronautics


