Numerical Analysis of the ONERA-M6 Wing with Wind Tunnel Wall Interference*

Taisuke NAMBU,1) Atsushi HASHIMOTO,2) Takashi AOYAMA,2) and Tetsuya SATO3)

1)Department of Applied Mechanics, Waseda University, Tokyo 169–8555, Japan
2)Aerospace Research and Development Directorate, Japan Aerospace Exploration Agency (JAXA), Tokyo 182–8522, Japan

The flow around an ONERA-M6 wing, including the effect of wind tunnel wall interference, is computed using CFD analysis with a porous wall model. The computational domain sets porous walls at the top and bottom of the wing section similar to actual wind tunnel experiments. The computational result captures almost the same shock wave shape as the wind tunnel. This could not be computed exactly in previous works that did not include wall effects. The interference of the porous walls reduces the Mach number and attack angle of the flow, and these effects alter the swept angle of front shock wave and the location of rear shock wave. The aerodynamics coefficients are also affected by the wall interference. The lift coefficient becomes smaller due to the reduction in attack angle. The lower Mach number decreases the drag coefficient, while the reduction in attack angle causes additional drag by a mechanism similar to induced drag.

Key Words: Wall Interference, Porous Wall, ONERA-M6 Wing, Computational Fluid Dynamics

Nomenclature

\( x, y, z \): coordinate (streamwise, normal to wing surface, spanwise)
\( u, v \): velocity perturbation of \( x, y \) direction
\( b \): wing span
\( l \): chord length of each cross section
\( c \): length of mean aerodynamic chord
\( M \): Mach number
\( \alpha \): attack angle
\( M_l \): Mach number at the leading edge
\( P_o \): porosity (ratio of hole to wall area)
\( C_p \): pressure coefficient
\( C_L \): lift coefficient
\( C_D \): drag coefficient
\( \rho \): density
\( \Delta C_p \): differential pressure coefficient of the wall
\( L \): depth of hole
\( D \): diameter of hole
\( \delta^* \): displacement thickness of boundary layer
\( \phi \): potential of velocity perturbation
\( U \): streamwise velocity of free stream
\( \beta \): shock angle of oblique shock wave
\( \theta \): turning angle of oblique shock wave

1. Introduction

Today, computational fluid dynamics (CFD) plays a significant role in aircraft development. The precision required for today’s aircraft development is quite high, a typical target precision for drag prediction being about 1 drag count \((C_D = 0.0001)\). In order for CFD analysis to meet these high requirements, its validation must be strict. Ideally, validation should be conducted by comparing CFD results with actual flight data. However, flight data measurement methods are not yet sufficiently mature to give the required accuracy, so wind tunnel test data are used for CFD validation. For strict validation of CFD solvers, these data need to be reliable, with experimental errors minimized as far as possible. However, it is difficult to exactly match the flow conditions in a wind tunnel with CFD computation, one of the most serious problems being the wind tunnel walls, which cause differences between wind tunnel flow conditions and actual flight conditions. This problem is known as “wind tunnel wall interference.” Although many past studies have attempted to correct wall interference, it is very difficult to remove completely from experimental data.

On the other hand, there have been attempts to include wind tunnel wall interference in CFD results.1–4) Today’s advanced mesh generation methods can easily construct computational meshes that include wind tunnel walls. The authors have also researched the wall interference problem using CFD analysis. However, in the case of a transonic wind tunnel, it is not easy to compute the wall interference strictly because the test section walls are permeable, such as porous walls. The effect of the flow through porous walls is non-negligible, so it is necessary to be modeled properly. We have therefore developed a new porous wall model for transonic wind tunnels by analyzing the flow through a single hole using computational and experimental approaches.5,6) This is the first model that can compute the effect of porous walls without requiring any empirical parameters. In our previous study, we attempted to apply the new model to compute the flow around a two-dimensional airfoil with wind tunnel walls. We succeeded in computing almost the same pressure distributions on the airfoil as wind tunnel test results, showing that the model was able to simulate the wall interference accurately.7)
In the present study, we analyze the wind tunnel wall interference in a three-dimensional manner. The flow field over an ONERA-M6 wing, which is a three-dimensional swept-back wing, including wall interference is analyzed by CFD analysis using the porous wall model. Wind tunnel data of the ONERA-M6 wing\(^8\) is one of the most popular references for CFD solver validation. The measured pressure distributions on the wing surface have shown relatively good agreement with many CFD results,\(^9\)-\(^17\) and so the data have been recognized as reliable and thus suited for validation. However, it is known that there are several cross-sections at which there are differences between experimental and CFD results. Figure 1 compares the pressure distributions at the cross-sections \(z/b = 0.2, 0.65, 0.8\) computed by three well-known CFD solvers: CFL3D,\(^9\) OVERFLOW,\(^10\) and Wind-US.\(^11\) The cross-sections that show relatively large differences are seen. In transonic flow conditions, a “lambda shock,” which is the characteristic shock wave structure for sweptback wings, arises on the suction surface. The shape of the lambda shock shows differences between computational results and experimental measurements. There are also differences around the suction surface near the root. Many CFD solutions in past studies have shown the same error tendencies, so it is doubtful to claim that these errors are caused only by issues related to computational processes. Generally, wall interference increases as the test model size grows relative to the test section size. The ratio of test section height to mean sonic wind tunnel tests. Wall interference seems to have a large influence on the ONERA-M6 test measurements, but the experiment values are not corrected for the wall interference. Therefore, it is important to examine whether this is the cause of the discrepancies seen between the CFD and experimental results. The S2MA wind tunnel, which is the transonic tunnel used for the ONERA-M6 testing, has porous walls at the test section, and the experiment conditions are recorded precisely, including the wall conditions. This data and our porous wall model will allow us to simulate the wall interference by accurately modelling the actual ONERA-M6 testing conditions.

There are two main purposes in the present study. The first one is to verify the extent to which wall interference effects contribute to ONERA-M6 test errors. The second one is to analyze the wall interference effect of a three-dimensional wing as a step to develop a new wall interference correction method for aircraft testings. At first, the flow of the ONERA-M6 wing is computed under the same conditions as the actual wind tunnel measurements including the wind tunnel walls, and it is compared with the experimental data. Then, the wall interference effect is discussed by comparing the flows with and without wind tunnel walls. In our previous study, we investigated how to change the flow field of a two-dimensional airfoil by wall interference and showed the wall interference reduced the Mach number and flow angle.\(^7\) Also in the present study, the interference values for the Mach number and flow angle are computed, and we investigate the wall interference effect in the three-dimensional flow field. Then, in order to understand the wall interference in more detail, the effect on the lambda shock and aerodynamics coefficients are discussed in terms of the interference values of Mach number and flow angle.

2. Analytical Methods

2.1. CFD methods

In the present computations, FaSTAR (FAST Aerodynamic Routines) is used as a CFD solver.\(^18\) In FaSTAR, the full Navier-Stokes equations are solved on an unstructured grid by a cell-centered finite volume method. HLLEW (Harten-Lax-van Leer-Einfeldt-Wada scheme)\(^19\) is used for the numerical flux computation, and the LU-SGS (Lower/Upper Symmetric Gauss-Seidel) implicit method is applied for time integration. Time integration is carried out by local time stepping. Second-order spatial accuracy is realized by linear reconstruction of the primitive variables with the MUSCL scheme. The Spalart-Allmaras model\(^20\) is used as the turbulence model.

2.2. Computational domain

In the present study, we compute the flow of the ONERA-M6 wing using computational grids with and without wind tunnel walls. Four types of wall configurations are employed: top and bottom wall only, side wall only, full wall (top, bottom and side walls), and no wall cases. Figure 2 shows the computational grid for the full wall case. The top and bottom walls at the test section are porous and are set at the same locations as in the actual experiment. The boundary condition in the porous wall region is computed using the porous wall model. There is no bleeding hole on the side wall and so it is computed using non-slip boundary condition. The top and bottom wall case and the side wall case have additional domains in the direction of the lacking wall. In the actual wind tunnel testing, a diverter is set at the root of the wing to avoid the boundary layer, and so that wall is computed...
using a slip wall boundary condition. Except for the full wall case, all primitive values at the inflow boundary condition and the static pressure at the outflow boundary condition are far-field values. In the full wall case, the total pressure and total temperature are fixed at the inflow boundary condition, and the static pressure at the outflow boundary condition is adjusted to realize the same flow conditions as the other cases forward of the airfoil. Figure 3 shows computational grids on the wing surface. Two types of grid, coarse and fine, are used to check grid dependence. The dimensions of the coarse mesh are 100 × 50 × 60 grid points (streamwise, normal, spanwise) and those of the fine mesh are 200 × 75 × 120 grid points in the block surrounding the wing. The attack angle is varied by rotating the computation grids near the wing. The computation grid around the wing is same for all cases to compute the flow under the same numerical conditions.

### 2.3. Porous wall model

At the porous wall boundary condition, the perpendicular velocity is computed by the porous wall model and other components are zero. The porous wall is modeled by the relationship between flow rate and differential pressure through the wall. Details of the modeling method can be found in Nambu et al.\(^5\)\(^6\) In this study, the porous wall is modeled by the following equations:

\[
\frac{(\rho u)_{\text{porous}}}{(\rho u)_{\infty}} = A \cdot P_O \cdot \Delta C_p
\]

\[
A = F_{LD} \cdot F_{BL}
\]

where \((\rho u)_{\text{porous}}\) is the mass flow rate through the wall, \((\rho u)_{\infty}\) is the mass flow rate of the free stream, \(\Delta C_p\) is the differential pressure coefficient of the wall, \(L/D\) is the ratio of each hole’s depth to diameter, and \(\delta^*\) is the displacement thickness of the boundary layer near the wall. In the present computation, the values of these parameters are same as those of the actual wind tunnel testing given in Schmitt and Charpin\(^8\) and Reference \(21\); \(\delta^* = 15\) [mm], \(P_O = 0.06\), \(D = 18\) [mm] and \(L/D = 1\). The values of the coefficient \(A\) is determined as 1.029 from these values.

### 3. Comparison with Experiment

In this part, the CFD results with wind tunnel walls are compared with actual wind tunnel test data. The experimental data at \(M = 0.8395\), \(\alpha = 3.06\), which is the most frequently used for validation, is the reference for comparison. All experimental values are not corrected by wall interference correction methods.

#### 3.1. Effect of top, bottom and side walls

At first, the flow around the ONERA M6 wing is computed for the different wall configuration cases. Figures 4 and 5 show the pressure coefficient distributions at the cross-sections \(z/b = 0.2, 0.65\) and 0.8. Computations for four cases, top and bottom walls, side wall, full wall and no wall, are conducted. The cross-section \(z/b = 0.8\) shows a two-step shock wave in the experiment, but the computational results of the no wall and side wall cases do not capture such a shock. On the other hand, the full wall case and top and bottom wall case capture shock waves similar to those observed by experiment. This means that interference of the top and bottom walls causes the cross point of the front shock and rear shock to move toward the wing tip. Past computations have not been able to capture the two-step shock wave exactly at \(z/b = 0.8\), so the present computations suggest that wall interference is the main cause of this discrepancy. Additionally, at \(z/b = 0.65\), the computational results of the full wall case and top and bottom wall case capture similar locations of the front and rear shocks as experiment. However, all computations still show differences with experiment on the suction surface at \(z/b = 0.2\), and so this might not be caused by wall interference.

The shock wave shapes computed by the full wall case and the top and bottom wall case show better agreement with experiment, and these two cases have almost the same distributions. This means that the top and bottom walls mainly affect the flow field. Therefore, the top and bottom wall case is focused on in the following discussion.
3.2. Grid dependence

Although the present computations come close to the experiment by including wind tunnel wall effects, there remain some differences. In this part, the grid density is increased and grid dependence is examined. Figures 6 and 7 show the pressure coefficient distributions at \( z/b = 0.2 \), 0.65 and 0.8 computed by coarse and fine meshes. The fine mesh captures the shock wave more sharply and its result shows better agreement with experiment at \( z/b = 0.8 \). However, the result at \( z/b = 0.2 \) is not improved even by the fine mesh. Therefore, this error is likely to be due to other factor. In the experiment, there is a diverter to avoid the boundary layer effect at the root of the wing, and so the condition of the root is not the exactly same between computation and experiment. We believe this is one of the reasons for the difference.

3.3. Other flow conditions

Comparisons are conducted with other flow conditions. Figure 8 shows the comparison with \( M_\infty = 0.8359 \), \( \alpha = 4.08 \). Additionally, in this flow, the computation with top and bottom walls captures the two-step shock wave more faithfully than the no wall case. On the other hand, the similar difference with the previous case shows up at \( z/b = 0.2 \). Figure 9 shows the results for \( M_\infty = 0.6990 \), \( \alpha = 3.06 \). In this case, there is no shock wave on the wing surface. Compared to the transonic case, no visible difference is seen between the results with and without walls, and both results agree with experiment. Therefore, the subsonic case seems to be less sensitive to wall interference than the transonic case.

4. Effect of Wind Tunnel Wall Interference

The interference of the top and bottom walls changes the flow around the wing, especially the shock wave. In this part, we discuss the wall interference by comparing the flows with and without wind tunnel walls.
4.1. Evaluation of interference quantity

In the present study, the linearized small perturbation potential equation is used to evaluate the wall interference. Using linear equations, velocity perturbations caused by wall interference and other factors can be solved separately. Therefore, the degree of interference the wind tunnel walls is specified explicitly. Many wall interference correction methods are based on this equation.22,23) The linearized small perturbation potential equation for wall interference correction can be written as:

\[
(1 - M^2) \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0
\]  \hspace{1cm} (5)

\[
\phi = \phi_{\text{wall}} - \phi_{\text{no wall}}
\]  \hspace{1cm} (6)

where \(\phi_{\text{wall}}\) and \(\phi_{\text{no wall}}\) are the perturbation potentials of flows with and without wind tunnel walls. The difference of these potentials means the velocity perturbation potential of the wall interference. This equation is differentiated with respect to \(x\) and \(y\), and the equation of velocity perturbation is then derived as:

\[
(1 - M^2) \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} = 0
\]  \hspace{1cm} (7)

\[
\Delta u = u_{\text{wall}} - u_{\text{no wall}}
\]  \hspace{1cm} (8)

\[
(1 - M^2) \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} = 0
\]  \hspace{1cm} (9)

\[
\Delta v = v_{\text{wall}} - v_{\text{no wall}}
\]  \hspace{1cm} (10)

where \(\Delta u\) and \(\Delta v\) are velocity perturbations caused by wall interference in streamwise and normal directions. The wall interference can be evaluated quantitatively by solving these equations using the CFD results as boundary conditions. An advantage of the present method is that the interference quantity on the wing chord can be specified where there is no CFD solution. Although previous research showed that the wall interference quantity computed by the linearized potential equation has some error in transonic flow cases,7) its accuracy is sufficient for the present qualitative discussion. In the present study, Eqs. (7)–(10) are solved numerically. Figure 10 shows the computational domain. The equations are spatially-discretized using the central difference and are solved by the Gauss-Seidel method. The computational grid has 100 points in all directions.

Figures 11 and 12 show the velocity perturbations on the chord from the leading edge to the trailing edge. Three types of porous wall are employed: high porosity, low porosity and no bleeding cases (\(Po = 0.2, 0.06, 0.0\)). The low porosity case, \(Po = 0.06\), corresponds to the porosity of the ONERA-M6 wing testing. In the high porosity case, both velocity components, \(\Delta u\) and \(\Delta v\), are decreased. On the other hand, the no bleeding case shows increases of both velocity components. The low porosity case results are between the high porosity case and the no bleeding case, and its interference is relatively small. The changes of streamwise and normal velocity almost correspond to the changes of Mach number and attack angle (\(\Delta M\) and \(\Delta \alpha\)). Therefore, interference by the porous wall causes \(\Delta M < 0\) and \(\Delta \alpha < 0\), while the wall without bleeding causes opposite interference, \(\Delta M > 0\) and \(\Delta \alpha > 0\).
and $\Delta \alpha > 0$. In the following sections, wall interference is discussed in terms of the changes of these values.

### 4.2. Effect on the shock wave

In this section, the wall interference effect on the shock wave is discussed. Figure 13 shows the pressure coefficient distributions of the cross-section at $z/b = 0.8$. The flow condition is $M_\infty = 0.8395$, $\alpha = 3.06$ and the porosities of these computations are 0.2, 0.06 and 0.0. The two-step shock wave is shown in each case, but the locations of the front and rear shock waves are altered by the change of porosity. Figure 14 shows the contours of pressure coefficient on the suction surface. In this figure, there are two visible changes: the swept angle of the front shock and the location of the rear shock. We first discuss the front shock. Figure 15 shows a schematic of the front shock. In the case of sweepback wings such as the ONERA-M6 wing, only the velocity component normal to the leading edge is accelerated, and the flow line is deflected towards the wing root. However, the flow line must be parallel to the symmetric plane, and so an oblique shock wave appears. The swept angle between the symmetric plane and the oblique shock wave, $\beta - \theta$, increases as the porosity become larger. As mentioned above, the Mach number and attack angle are reduced by the interference of porous wall (see Figs. 11 and 12). This means the Mach number at the leading edge $M_l$ decreases. The turning angles $\theta$ are nearly constant as the porosity varies, and the shock angle $\beta$ is a function of $M_l^{24}$ in that case. In the weak shock cases, the
shock angle increases as $M_1$ become small, and so $\beta - \theta$ also becomes large. On the other hand, the interference in the no bleeding wall case increases the Mach number and attack angle, and so it shows the opposite change.

Furthermore, the location of the rear shock wave is changed by wall interference. The front shock wave is a weak shock (i.e. supersonic in the downstream) and so the rear shock shows up at end of the supersonic region as a strong shock. Generally, that shock wave moves downstream as the Mach number increases. The interference of porous wall reduces the Mach number and attack angle, and this means that Mach number of the suction surface decreases. Therefore the rear shock wave is moved to the upstream by the interference.

### 4.3. Effect on lift and drag coefficients

In this part, the wall interference effect on the lift and drag coefficients is discussed. Table 1 shows the changes of the lift and drag coefficients due to wall interference. The values, $\Delta C_L$ and $\Delta C_D$, are difference between the results with and without walls ($\Delta C = C_{\text{wall}} - C_{\text{no wall}}$). The lift and drag coefficients of the no wall case are 0.25772 and 0.01952 respectively. In the no bleeding case, both coefficients increase; here wall interference increases the Mach number and attack angle as mentioned previously (see Figs. 11 and 12). In transonic flow conditions, the drag coefficient is sensitive to changes of Mach number and a small increment of Mach number causes a large increase in drag (i.e. drag divergence).

Obviously, the lift coefficient increases with the increment of the attack angle. In the high porosity case, the wall interference reduces both the Mach number and the attack angle. Obviously, the lift coefficient is reduced by the change of attack angle. The drag coefficient becomes smaller due to decrease of the Mach number. On the other hand, the decrease of the attack angle causes additional drag by a similar mechanism to induced drag. The change of the drag coefficient includes these two effects.

### 5. Conclusion

In the present study, the flow around the ONERA-M6 wing was computed including the effects of wind tunnel wall interference. In order to compute the flow field in conditions that closely match the actual wind tunnel testing, the effects of the wind tunnel walls at the top and bottom of the test section were computed using a porous wall model. First, we compared the computation results with actual wind tunnel test data. Previous computations for the ONERA-M6 wing without the effect of the wind tunnel wall have shown some differences between the computed shock wave shape and the experiment. However, the present computation, which included the effect of the top and bottom walls, captured almost the same shock wave shape as the experiment data. The wall interference was evaluated by solutions of the linearized small perturbation potential equation computed using the CFD results as boundary conditions. The wall interference of a porous wall reduced the Mach number and attack angle. On the other hand, the wall interference in the case of no bleeding increased these values. Due to these effects, the swept angle of the front shock wave and the location of the rear shock wave were changed. In the porous wall case, the swept angle between the front shock wave and the symmetric plane were increased and the rear shock wave was moved upstream. The wall interference also affected the lift and drag coefficients. The lift coefficient was affected by the change of attack angle. The change of Mach number altered the drag coefficient, and the smaller attack angle caused additional drag by a similar mechanism to induced drag.

### References

1) Hashimoto, A., Aoyama, T., Kohzai, M., and Yamamoto, K.: Transonic Wind Tunnel Simulation with Porous Wall and Support Devices, AIAA Paper 2010-4201, 2010.
2) Heidebrecht, A.: Simulation and Model Support Correction for Slotted Wall Transonic Wind Tunnels, 47th International Symposium of Applied Aerodynamics (CD-ROM), 2a-02.
3) Hantrais-Gervois, J. L., Mouton, S., and Pinat, J. F.: RANS Simulations to Compute Wind Tunnel Wall Corrections, 47th International Symposium of Applied Aerodynamics (CD-ROM), 2a-04.
4) Krynytzky, A., Plenum, M., Sommerfield, D., and Li, P.: Computational Modeling of a Slotted Wall Test Section, AIAA Paper 2012-2863, 2012.
5) Nambu, T., Hashimoto, A., Aoyama, T., and Sato, T.: Numerical Analysis of Flow through a Hole for Modeling of Wind Tunnel Porous Wall, 1388.1. Jpn. Soc. Aeronaut. Space Sci., 54 (2011), pp. 221–228.
6) Nambu, T., Hashimoto, A., Aoyama, T., and Sato, T.: Analysis and Modeling of Flow through Wind Tunnel Porous Wall, AIAA Paper 2010-4858, 2010.
7) Nambu, T., Hashimoto, A., Murakami, K., and Sato, T.: Numerical Analysis of Wind Tunnel Wall Interference on Two-dimensional Airfoil by New Porous Wall Model, AIAA Paper 2012-3229, 2012.
8) Schmitt, V. and Charpin, F.: Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers, AGARD AR, 138 (1979), pp. B1-1–B1-44.
9) CFL3D Version 6 Home Page, http://cfl3d.larc.nasa.gov/, CFL3D Test Cases.
10) User’s Manual for OVERFLOW 2.2, http://people.nas.nasa.gov/˜pallim/OVERflow/test2.1z.html/onera.m6/HOWTO/onera.m6.html
11) NPARC Alliance Validation Archive, http://www.grc.nasa.gov/WWW/wind/m6wing/m6wing01/m6wing01.html
12) Bonhaus, D. and Wornom, S.: Relative Efficiency and Accuracy of Two Navier-Stokes Codes for Simulating Attached Transonic Flow over Wings, AIAA Paper 1990-3078, 1990.
13) Hashimoto, A., Murakami, K., Aoyama, T., and Lahur, P.: Lift and Drag Prediction Using Automatic Hexahedra Grid Generation Method, AIAA Paper 2009-1365, 2009.
14) Durrani, N. and Qin, N.: Comparison of RANS, DES and DDES Results for ONERA M-6 Wing at Transonic Flow Speed Using an In-House Parallel Code, AIAA Paper 2011-190, 2011.
15) Collercandy, R.: An Improved Approach for the Computation of Transonic/Supersonic Flows with Applications to Aerospace Configurations, AIAA Paper 1992-2613, 1992.
16) Yang, J., Yen, R., and Peng, Y.: Three-Dimensional Wing Flow Computations Using Implicit Weno Euler Solvers, J. Aircraft, 39 (2002), pp. 181–184.
17) Wild, J.: Smooth Mixed Meshes for Acceleration of RANS CFD in
Aircraft Analysis and Design, AIAA Paper 2011-1267, 2011.

18) Hashimoto, A., Murakami, K., Aoyama, T., Ishiko, K., Hishida, M., Sakashita, M., and Lahur, P.: Toward the Fastest Unstructured CFD Code ‘FaSTAR,’ AIAA Paper 2012-1075, 2012.

19) Obayashi, S. and Guruswamy, G. P.: Convergence Acceleration of a Navier-Stokes Solver for Efficient Static Aeroelastic Computations, AIAA J., 33 (1995), pp. 1134–1141.

20) Spalart, P. R. and Allmaras, S. R.: A One-equation Turbulence Model for Aerodynamic Flows, AIAA Paper 1992-439, 1992.

21) ONERA, Web site of the Wind Tunnel Division, http://windtunnel.onera.fr/s2ma-continuous-flow-wind-tunnel-variable-pressure-mach-01-mach-31

22) Mokry, M. and Ohman, L. H.: Application of the Fast Fourier Transform to Two-dimensional Wind Tunnel Wall Interference, J. Aircraft, 17 (1980), pp. 402–408.

23) Ulbrich, N.: The Application of Panel Method Code ANTARES to Wind Tunnel Wall Interference Problems, AIAA Paper 2002-307, 2002.

24) Liepmann, H. W. and Roshko, A.: Elements of Gasdynamics, John Wiley & Sons, New York, 1957, pp. 84–93.