AIAA  2002-0982

Large-Eddy Simulation of a Circular Cylinder on Unstructured Grids

Robert P. Hansen
United States Military Academy
West Point, NY

Lyle N. Long
The Pennsylvania State University
University Park, PA

40th AIAA Aerospace Sciences Meeting and Exhibit
14 – 17 January 2002 / Reno, NV
LARGE-EDDY SIMULATION OF A CIRCULAR CYLINDER ON UNSTRUCTURED GRIDS

R. P. Hansen
Department of Civil and Mechanical Engineering
United States Military Academy
West Point, New York 10996
Robert.Hansen@usma.edu

L. N. Long
Department of Aerospace Engineering
The Pennsylvania State University
University Park, PA 16802
lnl@psu.edu

ABSTRACT

A computational study of unsteady, separated, low-Reynolds number fluid flow is made using a second-order accurate, cell-centered finite-volume method on unstructured grids. The three-dimensional simulations include large-eddy simulation turbulence modeling using a fixed-coefficient Smagorinsky subgrid-scale model. The flow about a three-dimensional circular cylinder at Reynolds number 3900 and Mach number 0.2 was studied using a standard Navier-Stokes formulation with and without the subgrid-scale modeling. A third case was also tested using a modified form of the Navier-Stokes equations with subgrid-scale modeling. Comparisons are made with experiment and selected three-dimensional numerical calculations performed on structured grids. The frequency of the unsteady vortex shedding was in good agreement with experiment, however, the numerical dissipation inherent in the upwind flux calculations affected the details of the time-averaged flowfield. The results showed an increased base pressure, shortened recirculation zones in the cylinder wake, and excessive drag compared to experiment for all cases.

Introduction

While problems involving flow over complex geometries have already been attempted using unstructured grid methods1,2,3, the ability of unstructured methods to accurately simulate flow involving large-scale unsteadiness and turbulence has not been clearly demonstrated. The allure of accurately solving complex problems using the very efficient cell or element distribution offered by unstructured grids motivates continued research in this area.

The effectiveness of unstructured grid solutions for flow over standard geometries such as cubes and cylinders has not been systematically documented for both the large-scale mean properties and turbulent properties. The unsteady flow over a square cylinder4 and cube5 using unstructured grids has been studied as well as two-dimensional solutions for circular cylinders6 using unstructured methods. Numerous studies have provided critical insights into the advantages and disadvantages of various algorithms applied to three-dimensional cylinders using structured grids7,8,9,10. The study of the three-dimensional circular cylinder using unstructured grid methods, however, is either scarce or non-existent. This paper presents results for a simulation of flow over a three-dimensional cylinder using an unstructured, cell-centered, finite-volume method.

The physics of flow about a circular cylinder is far more complex than its relatively simple shape might suggest. Despite being one of the most experimentally tested geometries in fluid mechanics, three-dimensional numerical studies of the circular cylinder have only been attempted in the last decade. Parallel computation, as well as more powerful vector computations, has made greater grid resolution possible, which in turn has made the use of LES feasible for problems requiring a large number of grid points. Direct Numerical Simulation of flow over a circular cylinder at a Reynolds number of 300 has even been accomplished11. Much higher Reynolds numbers have been achieved using Detached - Eddy Simulation. Travin et al.12 have computed results for the three-dimensional cylinder up to Reynolds number 3 x 10^6.

Several experiments provide a very complete data set for flow over a circular cylinder at Reynolds number 3900. Norberg13 studied the effect of freestream turbulence on the measured flow, and provides surface pressure data as well as frequency of shedding for a range of Reynolds numbers. Krothapalli et al.14 and Lourenco and Shih15 measured time-averaged velocities and Reynolds stresses in the cylinder near-wake at Reynolds number 3900. Ong and Wallace16 provide experimental data at the same Reynolds number at positions further downstream of the cylinder, beyond the recirculation zone. These more recent, lower Reynolds number experiments have provided computational
researchers with a test case that is easily within the capability of current computing power. At the Reynolds number of 3900 the attached boundary layer on the cylinder surface is laminar, the separated shear layers are in the early stages of transition, and the wake is fully turbulent. At this Reynolds number, the boundary layer is thick enough to be easily resolved by the computational grid. The experimental off-body data (pressure, velocities, Reynolds stresses) are also close enough to the cylinder so that an excessive number of computational cells in the cylinder wake can be avoided. Despite the fact that some part of the flowfield around a cylinder at Reynolds number 3900 is in transition, a condition that is obviously more difficult to compute, the overall advantages of using this Reynolds number are judged to outweigh any difficulties caused by the weakly transitional shear layers.

The computational results of Breuer, Beaudan, and Mittal for Reynolds number 3900 have added great insight into the advantages and disadvantages of various numerical schemes available on structured grids. Breuer used the Smagorinsky LES model with five different flux calculation schemes on a structured grid. He showed that the type of numerical scheme had a large impact on features of the time-averaged flowfield and parameters such as drag. Central difference approximation schemes produced more accurate results, while the schemes employing upwinding tended to shorten the recirculation bubble and predict excessive drag compared to experimental results.

Beaudan used a finite difference approach with fifth- and seventh-order upwind biased schemes to discretize the convective terms. His simulations for three cases -- no turbulence model, Smagorinsky LES model, and dynamic LES model -- all over-predict the length of the recirculation bubble, with the dynamic model falling within only two percent of the experimental value. Other characteristics of the recirculation bubble, such as the location and value of the maximum streamwise velocity along the wake centerline, were less accurate. Beaudan also evaluated the one-dimensional turbulence spectra at several downstream locations for both the fifth- and seventh-order accurate simulations.

While the main effort of this research is to evaluate a very difficult test case using unstructured grids, improvements in the solution accuracy were attempted by using a novel splitting method. Several researchers have used the technique of expressing the Navier-Stokes equations in terms of steady and unsteady components. Huia et al. used a decomposition of instantaneous variables similar to that done in the present research for investigating the evolution of a vortex in a three-dimensional boundary layer. Huia et al. decomposed each variable into a time-independent base flow and a time-dependent disturbance. By substituting this decomposition into the Navier-Stokes equations and subtracting out the base flow, they formed what they called the disturbance equations.

The splitting approach used here, called the nonlinear disturbance equations (NLDE), was first proposed by Morris et al. to compute solutions to high-speed circular and rectangular jet noise problems. The NLDE have since been successfully applied to a variety of viscous and inviscid problems. Hansen et al. found the frequency of unsteady shedding over a two-dimensional cylinder to be more accurate using the NLDE formulation compared to a standard Navier-Stokes solution.

### Governing Equations and Numerical Implementation

#### Governing Equations

The spatially filtered, time-dependent, Navier-Stokes equations are used. In integral form they are:

\[
\frac{\partial}{\partial t} \iiint_V \tilde{Q} \, dV + \iint_S \tilde{F} \cdot \hat{n} \, dS = 0
\]

The conserved vector variable is given by:

\[
\tilde{Q} = \begin{bmatrix}
\tilde{p} \\
\tilde{p} \tilde{u} \\
\tilde{p} \tilde{v} \\
\tilde{p} \tilde{w} \\
\tilde{p} \tilde{E}
\end{bmatrix}
\]

where the overbar indicates a spatially filtered quantity and the tilde (~) denotes Favre (density weighting) averaging. The fluid density is given by \(\rho\), the Cartesian velocity components are \(u, v, \) and \(w\), and \(E\) is the energy per unit mass. The flux vector consists of:

\[
\tilde{F} = F_x \hat{i} + F_y \hat{j} + F_z \hat{k}
\]

where

\[
\tilde{F}_s = \begin{bmatrix}
\tilde{p} \tilde{u} \\
\tilde{p} \tilde{u}^2 + \tilde{p} - \tilde{\tau}_{ux} + \tau'_{ux} \\
\tilde{p} \tilde{u} \tilde{v} - \tilde{\tau}_{uv} + \tau'_{uv} \\
\tilde{p} \tilde{u} \tilde{w} - \tilde{\tau}_{uw} + \tau'_{uw} \\
\tilde{p} \tilde{E} + \tilde{p} \tilde{v} - \tilde{\tau}_{ux} \hat{i} - \tilde{\tau}_{uy} \hat{j} - \tilde{\tau}_{uz} \hat{k} + \tilde{q}_s + \Theta_s
\end{bmatrix}
\]
where $Pr_t$ is the turbulent Prandtl number and is assigned a value of 0.1 in this study. The resolved deformation rate tensor is defined by:

$$\tilde{S}_y = \frac{1}{2} \left( \frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} \right)$$

The model for the SGS heat fluxes is:

$$q''_i = \frac{\rho \mu}{\gamma - 1 Pr_t} \frac{\partial T}{\partial x}$$

where $Pr_t$ is the turbulent Prandtl number and is commonly chosen in the range of 0.3 to 0.5.

The last term in the filtered momentum equation is the SGS stress. The SGS stresses are defined by:

$$\tilde{F}_i = \begin{cases} \rho \tilde{v} \tilde{v} \\
\rho \tilde{u} \tilde{v} - \tilde{\tau}_{xy} + \tilde{\tau}_{xy}' \\
\rho \tilde{v}^2 + p - \tilde{\tau}_{yy} + \tilde{\tau}_{yy}' \\
\rho \tilde{w} \tilde{v} - \tilde{\tau}_{yz} + \tilde{\tau}_{yz}' \\
(\rho \tilde{E} + \tilde{p}) \tilde{v} - \tilde{\tau}_{xy} u - \tilde{\tau}_{xy} v - \tilde{\tau}_{yz} w + \tilde{q}_y + \Theta_y \\
\rho \tilde{w} \tilde{v} - \tilde{\tau}_{yz} + \tilde{\tau}_{yz}' \\
\rho \tilde{w}^2 + p - \tilde{\tau}_{zz} + \tilde{\tau}_{zz}' \\
(\rho \tilde{E} + \tilde{p}) \tilde{v} - \tilde{\tau}_{xy} u - \tilde{\tau}_{xy} v - \tilde{\tau}_{yz} w + \tilde{q}_y + \Theta_y \end{cases}$$

$$\Theta_i = \tau_{i'} u + \tau_{j'} v + \tau_{k'} w - q_i'^*$$

The subgrid-scale (SGS) stresses and heat fluxes appear as $\tau_{ij}'$ and $q_i'^*$, respectively, in the above. Using the Smagorinsky model, the SGS stresses are defined by:

$$\tau_{ij}' = -2 \mu \tilde{S}_{ij}$$

where $C_s$ is the Smagorinsky constant and is assigned a value of 0.1 in this study. The resolved deformation rate tensor is defined by:

$$\tilde{S}_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} \right)$$

The decomposition of each variable is substituted into the filtered equations, and the terms containing purely mean flow variables are placed on the right-hand side, so that equation 1 may now be written as:

$$\frac{\partial}{\partial t} \int V \{ Q + F^* \} dV + \int_S F^* \cdot \hat{n} dS = R_s$$

where $A^*$ is a constant equal to 25.0, and $y^*$ is the distance, in viscous wall units, from the wall to the field point where eddy viscosity is to be evaluated. The viscous wall unit is defined by:

$$y^* = \frac{d}{\sqrt{\nu \rho_w}}$$

where $d$ is the distance from a cell center to the nearest face that is on a viscous surface, $\tau_w$ is the shear stress at the wall, $\rho_w$ is the fluid density at the wall, and $\nu$ is the coefficient of dynamic viscosity. Calculation of $y^*$ on a partitioned, unstructured grid requires particular care and is discussed in the section on numerical implementation.

Use of the nonlinear disturbance equations (NLDE) requires the resolved-scale variables to be decomposed into a mean quantity that is independent of time and a time-dependent perturbation. For example:

$$\rho(x,t) = \rho_o(x) + \rho'(x,t)$$

$$u(x,t) = u_o(x) + u'(x,t)$$

The decomposition of each variable is substituted into the filtered equations, and the terms containing purely mean flow quantities are placed on the right-hand side, so that equation 1 may now be written as:

$$\frac{\partial}{\partial t} \int V \{ Q + F^* \} dV - \int_S F^* \cdot \hat{n} dS = R_s$$

where

$$R_s = -\frac{\partial}{\partial t} \int V \{ Q + F^* \} dV - \int_S F^* \cdot \hat{n} dS$$

The term that is to be modeled is:

$$F^* = \left\{ \begin{array}{l} \rho' \\
\rho u' u + \rho' v' \\
\rho w' w + \rho' w' \\
\rho' - \tilde{\tau}_{ij} \\
\rho' - \tau_{ij}' \end{array} \right\}$$

The coefficient of dynamic viscosity is defined by:

$$\nu = \frac{\mu}{\rho}$$

The typical grid is too coarse in the boundary layer, the "large" scale eddies approach the Smagorinsky model is its behavior close to the wall. In the boundary layer, the "large" scale eddies approach the Smagorinsky model is its behavior close to the wall. Use of a van Driest damping function with LES provides the required reduction in eddy viscosity near the wall. The Smagorinsky coefficient in equation 4 is replaced by $C_s^*$ defined as:

$$C_s^* = C_s \sqrt{1 - e^{|\gamma - 1/2 A^*|}}$$

The model for the SGS heat fluxes is:

$$q''_i = \frac{\rho \mu}{\gamma - 1 Pr_t} \frac{\partial T}{\partial x}$$

American Institute of Aeronautics and Astronautics
\[
F_i = \begin{bmatrix}
0 \\
\tau_{xx} \\
\tau_{yy} \\
\tau_{zz}
\end{bmatrix}
\]

\[
\Theta_i = \tau_{xx}' (u_x + u') + \tau_{yy}' (v_y + v') + \tau_{zz}' (w_z + w') - \tau_{xy}' w_x + \tau_{xz}' w_y + \tau_{yz}' w_z.
\]

The remaining components of \( F' \) and \( F^- \) are similarly expanded. The variable \( Q' \) in equation 10 is a new vector of conserved variables consisting of only interactions between the perturbations themselves or the perturbations and the mean flow. \( F^- \) contains all of the SGS stresses and heat fluxes, and \( Q_o \) and \( F_o \) are the same as defined by equations 2 and 3 except all terms are now resolved mean quantities. Depending on how the mean flow is obtained, the value of \( R_o \) may or may not be zero. When \( R_o \) is a constant it is calculated only once at the beginning of a computation. A steady, uniform flow is chosen as the mean flow in this study, reducing \( R_o \) to zero.

Numerical Implementation

The governing equations are discretized using a second-order accurate, cell-centered finite-volume method with upwinding for stability. The code used in this research is FLOUS (FLow UnStructured), originally written by Long in Connection Machine FORTRAN (CMF). Weinberg and Long later modified this code to use adaptive meshing. The present research included updating of the FLOUS code to FORTRAN90 and the addition of Message Passing Interface (MPI) library subroutine calls to allow parallel operation. Large-eddy simulation, in the form of a constant coefficient Smagorinsky model, was also added for turbulence modeling. The upwinding technique employs Roe’s flux difference splitting. A standard 4th order Runge-Kutta integration is used to explicitly advance the solution in time. Unstructured grids composed of tetrahedral cells were generated using the Gridtool/VGRIDns 3.3 software. The grid was decomposed into subdomains or zones using the METIS 4.0 partitioning software. Several utilities were also written for FLOUS to handle a variety of pre- and post-processing tasks.

Parallel operation of the code using MPI requires the exchange of all conserved variables between the cells on either side of a face that defines a zonal boundary. This exchange occurs several times per time step. Additional message passing requirements are incurred when calculating the eddy viscosity damping given by equation 7. Equation 8 implies an association between each cell center and the nearest solid, viscous face. Each cell identifies the location of and distance from its nearest wall face once as the code initializes. In a partitioned, unstructured grid the cell and its nearest solid surface may not be part of the same zone. This requires information, specifically the shear stress and the density, at the wall be sent from the face center to the cell where eddy viscosity is to be evaluated. This additional message passing takes place each time step.

Simulations and Results

Simulations

A circular cylinder in a crossflow at Reynolds number 3900 has been simulated at Mach = 0.2. The cylinder is four diameters in span. A symmetry boundary condition is placed at the end walls, creating the effect of an inviscid wall. Figure 1 shows the surface grid of the cylinder. Approximately 28 cells define the cylinder surface in the spanwise(y) direction, or seven cells per diameter. The farfield boundaries are placed 10 diameters from the cylinder surface except for the downstream exit plane, which is 20 diameters from the cylinder. The complete grid consists of 308,208 tetrahedral cells. All time-averages are computed using ten cycles of shedding. Spanwise averaging is also performed using 13 points across the span.

All computations were done on the NASA Ames Research Center 512-processor Origin 2000 system (LOMAX cluster) using 64 processors. Each cycle of shedding required approximately 2.8 hours of wall time, or about 0.25 seconds per iteration per cell.

Figure 1. Surface Mesh of the Circular Cylinder

Three cases are compared – the baseline FLOUS code without LES turbulence modeling, the baseline...
A summary of key time-averaged parameters is given in Table 1. The computed coefficient of drag, $C_D$, is substantially higher in all cases than the experimental value. The base pressure coefficient, $C_{pb}$, is defined as the coefficient of pressure at the point on the cylinder surface furthest downstream. The magnitude of $C_{pb}$ is also larger than the experimental value for all cases. The Strouhal number is slightly above the experimental value for the baseline code, but falls within the experimental range for the cases with LES. The case B2 from Breuer7 is used for comparison because that case used an upwind approximation for flux calculations. The Strouhal number for this case was not provided, but Breuer stated it was within the range of experiment. The length of the recirculation bubble, $L_r$, is non-dimensionalized by the cylinder diameter as $L_r/D$ in the last column of Table 1. There is a direct correlation between the drag coefficient and the length of the recirculation zone – the shorter the recirculation zone the higher the drag coefficient.

Figure 2 shows the time-averaged streamwise velocity along the wake centerline for all cases. Also shown are experimental results and selected results from Beaudan8 and Breuer7. The mean flow is in the positive $x$-direction in all figures and the distance along the wake is non-dimensionalized by the cylinder diameter. In all the present cases, the recirculating zone is significantly shorter than it should be. The cases using LES are very similar to the Breuer case B2 that uses a Smagorinsky subgrid-scale model and an upwind scheme for the flux calculations.

The addition of LES to the baseline code has the desirable effect of correcting the minimum velocity in the recirculating zone to a value consistent with experiment. The length of the recirculating zone is, however, slightly shortened when LES is added. Use of the NLDE formulation shortens the recirculating zone even further and reduces the minimum velocity in the bubble to a value below the experimental value.

Figure 3 shows the time-averaged, span-averaged streamlines for the NLDE model. This case has the shortest time-averaged recirculation zone, but is otherwise qualitatively the same as the non-NLDE cases.

Figure 4 shows the time-averaged streamlines for a three-dimensional numerical computation from Breuer7.
(case D3) which used a central difference approximation scheme with a dynamic subgrid-scale LES turbulence model. All parameters of this simulation are very close to the experimental ranges, and the time-averaged streamlines are also accepted to be an accurate representation of the experimental data. Comparison of Figure 3 and Figure 4 show the time-averaged recirculation zone of the present study to be less than half the length of the Breuer case D3, consistent with the data in Table 1. Additionally, the two smaller, counter-rotating vortices seen in Figure 3 are absent from Figure 4.

Figure 3. Time-Averaged Streamlines for the NLDE Case

Figure 4. Time-Averaged Streamlines from Breuer (reference 7) Case D3

Figure 5 reveals the strong three-dimensional flow that results from performing the cylinder computation in three dimensions. Shown is an iso-surface of absolute vorticity. Such three-dimensionality was absent when similar computations were performed during the present research using grids only two diameters in span.

The reason for the poor performance of all cases in the areas of drag, base pressure coefficient, and length of the recirculating zone appears to be related to the use of upwinding. Breuer\textsuperscript{7} was able to obtain significantly better results than the B2 case used here for comparison by using central difference approximations. The inherent numerical dissipation in Roe’s upwinding scheme, used in the present research, appears to be excessive, causing the wake to be dominated by the largest scales. This results in a wake pressure that is too low and a drag that is too high.

Figure 6 shows the instantaneous vorticity structure on the center-plane perpendicular to the cylinder axis obtained from the baseline FLOUS code. The structure of the vorticity in the wake is very similar to that obtained from a purely two-dimensional simulation. The wake is dominated by large coherent structures. Much smaller scale motion should be superimposed on the periodic vortex shedding. The oversized large-scale motion also appears responsible for inducing the counter-rotating smaller vortex adjacent to the cylinder surface.

Figure 6. Instantaneous Vorticity Magnitude (darker shading indicates higher vorticity magnitude)

The drag and lift coefficient time-histories for approximately six and one-half shedding cycles are shown in Figure 7 and Figure 8. While the time-
A comparison of the Reynolds stresses with experimental data was also made. All Reynolds stress components are normalized by $U_\infty^2$. Figure 9 compares the streamwise normal Reynolds stress ($\overline{u'u'}$) of the different cases. The near wake appears very similar for all cases. Only further downstream are some differences observable.

Figure 10 shows the streamwise Reynolds normal stress component ($\overline{u'u'}$) at the location $x/D = 1.54$ behind the cylinder. The streamwise components computed by the baseline code and the baseline code with LES under-predict the peak Reynolds stresses by as much as 12%. The NLDE version predicts the peaks to within one percent, but the locations of the peaks are slightly displaced from the experimental locations. It should be noted that there is a degree of uncertainty of the exact peak of the experimental data based on the fact that a number of curves could fit the data providing different peaks. Percentage differences used in this discussion are relative to the largest values plotted.

Figure 11 shows the $\overline{u'w'}$ Reynolds shear stress component for the NLDE case in the $xz$ plane. Figure 12 shows the shear stress value at $x/D = 1.54$ for all cases. All the cases over-predict the shear Reynolds stress by as much as 36%. The baseline FLOUS code using LES appears to be slightly more accurate than the other cases, but is still 27% over the experimental value. The $\overline{w'w'}$ lateral normal Reynolds stress component is shown in Figures 13 and 14. All calculations significantly over-predict the $\overline{w'w'}$ component by at least 100%. The calculations of Mittal\textsuperscript{18} also significantly over-predict the magnitude of this component. Breuer\textsuperscript{7} successfully captures this component when using a dynamic LES model with central difference approximations.
x/D = 1.54 for the NLDE case against two other numerical simulations. The results of Beaudan\textsuperscript{8} are used as the benchmark because experimental data is not available for this component. The peak stress value from Beaudan is 35% higher than the NLDE case or Mittal\textsuperscript{18} calculation. Figure 16 indicates a reduced level of three-dimensionality in the near-wake for the present study. The reduced spanwise stresses contribute to the higher values of the in-plane $\overline{u'w'}$ and $\overline{w'v'}$ components. This precise condition has been shown by Mittal and Balachandar\textsuperscript{35} to lead to a higher value of base pressure coefficient and higher drag.

Both Beaudan\textsuperscript{8} and Mittal\textsuperscript{18} used grids with approximately 15 cells per diameter along the span of the cylinder, or roughly twice the spanwise density used in this study. Despite the differences in grid density between the present study and Mittal, who used a second-order finite difference method, the peak value for spanwise Reynolds stress is the same. This indicates that the seventh-order scheme used by Beaudan accounts for the better accuracy of his results, and not the grid density.
CONCLUSIONS

The accuracy of the three-dimensional cylinder drag, base pressure, shedding frequency, and length of the recirculation zone were all degraded by application of the NLDE method. The only benefit from using the NLDE form was a modest improvement in the peak normal Reynolds stress in the streamwise direction.

The application of Large Eddy Simulation using a fixed-coefficient Smagorinsky model had a noticeable effect on key parameters of the three-dimensional cylinder tests. Relative to the baseline FLOUS code, the Strouhal number was more accurate and the minimum velocity in the recirculating zone was noticeably improved when LES was used. The overall poor quality for all three-dimensional cylinder cases is attributed primarily to the upwinding used in the finite-volume formulation. The use of upwinding introduces undesirable levels of artificial dissipation. Grid refinements studies should be conducted, however, to separate the effects of grid resolution and dissipation caused by upwinding on the accuracy of these results.

The artificial dissipation inherent in upwinding is necessary, to an extent, for numerical stability. Other researchers7,18 have shown that central difference approximation schemes are superior on structured grids. The central difference schemes, however, require the explicit addition of artificial dissipation for numerical stability. Explicit artificial dissipation is not a current capability on three-dimensional unstructured grids, limiting unstructured methods to the use of upwinding. While the structured, higher-order, upwinded method used by Beaudan8 performed better than the method used in the present research, dependable higher order methods for unstructured finite-volume grids have not been developed. The ultimate effect of excessive dissipation experienced in the three-dimensional cylinder cases is the domination of large-scale vortices in the cylinder near-wake, leading to a pressure that is too low and a drag prediction that is excessive.

References


